KISSSOFT

Calculation programs for machine design

KISSSOFT RELEASE 03/2014 USER MANUAL

Issue 1.4

Copyright Notice

© 2014 KISSsoft AG
Rosengartenstrasse 4
CH-8608 Bubikon Schweiz

All rights reserved
This documentation may not be copied without the express written approval of KISSsoft AG.

Table of Contents

I General I-38

1 Installing KISSsoft I-39

1.1 Basic installation I-40

1.2 Downloading a license file I-41

1.3 Licensing I-42

1.3.1 Test version I-42

1.3.2 Student version I-42

1.3.3 Single user version with dongle (protection key) I-42

1.3.4 Single user version with license code I-43

1.3.5 Network version with dongle (protection key) I-43

1.3.6 Network version with the license code I-44

2 Setting Up KISSsoft I-45

2.1 Directory structure I-46

2.2 Language settings I-47

2.3 System of units I-48

2.4 Defining your own default files I-49

2.5 Rights I-50

2.6 Global settings - KISS.ini I-51

2.6.1 Definitions in [PATH] I-51

2.6.2 Definitions in [SETUP] I-52

2.6.3 Definitions in [REPORT] I-53

2.6.4 Definitions in [GRAPHICS] I-54

2.6.5 Definitions in [LICENSE] I-54

2.6.6 Definitions in [CADEXPORT] I-55

2.6.7 Definitions in [INTERFACES] I-55

2.6.8 Definitions in [SOLIDEDGE] I-56

2.6.9 Definitions in [SOLIDWORKS] I-56

2.6.10 Definitions in [INVENTOR] I-56

2.6.11 Definitions in [CATIA] I-56
2.6.12 Definitions in [PROENGINEER] I-57
2.6.13 Definition in [COCREATE] I-57
2.6.14 Definitions in [THINK3] I-58
2.6.15 Definitions in [HICAD] I-58
2.7 User-defined settings I-59
2.7.1 Configuration tool I-59
2.8 Rules I-63

3 Starting KISSsoft I-66
3.1 Initial parameters I-67
3.2 Disconnect license from the network I-68

4 Elements of the KISSsoft User Interface I-69
4.1 Menus, context menus and the Tool bar I-70
4.2 Docking window I-72
4.2.1 The module tree I-72
4.2.2 The project tree I-73
4.2.3 The Results window I-73
4.2.4 The Messages window I-73
4.2.5 The info window I-73
4.2.6 Manual and Search I-74
4.3 Graphics window I-75
4.3.1 Tool bar and context menu I-76
4.3.2 Comment field I-78
4.3.3 Context menu I-78
4.3.4 Properties I-78
4.3.5 Toothing I-80
4.4 Main input area I-82
4.4.1 Report Viewer I-82
4.4.2 Helptext viewer I-83
4.5 Tooltips and status bar I-84

5 KISSsoft Calculation Modules I-85
5.1 Standard and special tabs I-86
5.2 Input elements I-87
5.2.1 Value input fields I-87
5.2.2 Formula entry and angle input I-87
5.2.3 Unit switch I-88
5.2.4 Tables I-88
5.3 Calculating and generating a report I-89
5.4 Messages I-90

6 Project Management I-91
6.1 Creating, opening and closing projects I-92
6.2 Adding and deleting files I-93
6.3 The active working project I-94
6.4 Storage locations I-95
6.5 Project properties I-96

7 Results and Reports I-97
7.1 Results of a calculation I-98
7.1.1 Add your own texts in the results window I-98
7.2 Calculation reports I-99
7.3 Drawing data I-100
7.4 Report settings I-101
7.4.1 General I-101
7.4.2 Page layout I-101
7.4.3 Header and footer I-101
7.4.4 Start and end block I-102
7.5 Report templates I-104
7.5.1 Storage locations and descriptions I-104
7.5.2 Scope of a report I-105
7.5.3 Formatting I-105

8 Database Tool and External Tables I-114

8.1 Viewing database entries I-116
8.2 Managing database entries I-119
8.2.1 Generating a database entry I-119
8.2.2 Deleting a database entry I-120
8.2.3 Restoring a database entry I-120
8.3 Import and export data with the database tool I-121
8.4 External tables I-122
8.4.1 Functions tables I-123
8.4.2 Range tables I-125
8.4.3 List tables I-126
8.4.4 List of key words used I-128
8.5 Description of database tables I-130
8.5.1 Center distance tolerances I-130
8.5.2 Machining allowance for cylindrical gear I-130
8.5.3 Reference profiles I-130
8.5.4 Compression springs standard I-130
8.5.5 Selection of hobbing cutters I-131
8.5.6 Basic material Glued and Soldered joints I-131
8.5.7 Manufacturing process for bevel and hypoid gears I-131
8.5.8 V-belt Standard I-131
8.5.9 Spline Standard I-132
8.5.10 Chain profiles ISO606 ..... I-132
8.5.11 Adhesives I-132
8.5.12 Modifications I-132
8.5.13 Load spectra I-132
8.5.14 Solders I-133
8.5.15 Surface roughness of shafts and shaft-hub connections I-134
8.5.16 Key standard I-134
8.5.17 Polygon standard I-134
8.5.18 Woodruff Key standard I-134
8.5.19 Bolts/pins I-134
8.5.20 Lubricants I-134
8.5.21 Bolts: Tightening factor I-136

8.5.22 Bolts: Bore I-136
8.5.23 Bolts: Strength class I-136
8.5.24 Bolts: Nuts strength class I-137
8.5.25 Bolts: Coefficients of friction classes I-137
8.5.26 Bolts: Thread type I-137
8.5.27 Bolts: Nuts I-138
8.5.28 Bolts: Type I-138
8.5.29 Bolts: Washer I-138
8.5.30 Selection of pinion type cutters I-138
8.5.31 Disc spring standard I-138
8.5.32 Tolerances standard I-139
8.5.33 Beam profiles I-139
8.5.34 Multi-Spline standard I-139
8.5.35 Materials I-139
8.5.36 Roller bearing I-144
8.5.37 Roller bearing tolerance I-152
8.5.38 Roller bearing Tolerance classes I-152
8.5.39 Tooth thickness tolerances I-152
8.5.40 Toothed belt standard I-153

9 Description of the public interface I-155

9.1 Interfaces between calculation programs and CAD - Overview I-156
9.1.1 Efficient interfaces I-156
9.1.2 Open interfaces concept in KISSsoft I-157
9.2 Defining input and output I-159
9.2.1 Preamble I-159
9.2.2 Requirements placed on the third party program I-160
9.2.3 Used files I-160
9.2.4 Service life of files I-161
9.2.5 Explicitly reading (importing) and generating data I-161
9.3 Example: Interference fit calculation I-162
9.4 Geometry data I-164
9.5 COM Interface I-165

9.5.1 Registering the server I-165
9.5.2 Server functionality I-165
9.5.3 Example of a call from Excel I-167

10 3D interfaces I-173
10.1 Overview of the available CAD interfaces and their functionality I-174
10.2 Generation of 3D gears I-175
10.3 Generating 3D shafts I-177
10.4 Viewer with neutral format interface I-179
10.4.1 Parasolid Export of 3D Shafts I-180
10.4.2 Face gear - 3D geometry I-180
10.4.3 Bevel gear - generating a 3D model I-182
10.4.4 Worm wheel - generating a 3D model I-183
10.5 3D interface to Solid Works I-184
10.5.1 Gear teeth in the case of an existing basic solid I-184
10.5.2 Integrating the KISSsoft Add-in (menu items in CAD) I-186
10.5.3 Add-in functions (calls) I-189
10.6 3D interface to Solid Edge I-192
10.6.1 Changes of the parameters for generation I-192
10.6.2 Gear teeth in the case of an existing basic solid I-192
10.6.3 Integrating the KISSsoft Add-in (menu items in CAD) I-194
10.6.4 Add-in functions (calls) I-198
10.6.5 Opening the calculation file for the created gear I-199
10.7 3D interface to Autodesk Inventor I-200
10.7.1 Gear teeth in the case of existing shaft data I-200
10.7.2 Add-in (menu items in CAD) I-201
10.7.3 Add-in functions (calls) I-204
10.7.4 Opening the calculation file for the created gear I-205
10.8 3D interface to Unigraphics NX I-206
10.8.1 Add-in (menu items in CAD) I-207
10.8.2 Running KISSsoft via an add-in I-209
10.8.3 Add-in functions (calls) I-216
10.9 3D interface to Creo Parametric (ProEngineer) I-217

10.9.1 Integrating the KISSsoft Add-in I-221
10.9.2 Modifying the selected 3D model I-224
10.9.3 Cutting teeth on an existing shaft I-225
10.9.4 Modifying the teeth on an existing shaft I-227
10.9.5 Changing base settings in the interface I-228
10.10 3D interface to CATIA I-230
10.10.1 Registering the interface I-230
10.11 3D interface to CoCreate I-233
10.12 3D interface to ThinkDesign I-235
10.12.1 Integrating the KISSsoft Add-in I-237
10.13 3D interface to ASCON Kompas I-238

11 Answers to Frequently Asked Questions I-239
11.1 Change the output of angles in reports I-240
11.2 Input materials for gear calculations in the database I-241
11.3 How can I test the software? I-242
11.4 What licenses are available? I-243
11.5 Add your own texts in the results window I-244
11.6 Restore previous stages of the calculation I-245

II Toothing II-246

12 Introduction II-247

13 Cylindrical gears II-248
13.1 Basic data II-250
13.1.1 Normal module II-250
13.1.2 Pressure angle at normal section II-250
13.1.3 Hand of gear for gear teeth II-250
13.1.4 Helix angle at reference circle II-251
13.1.5 Center distance II-251
13.1.6 Number of teeth II-252
13.1.7 Facewidth II-253

13.1.8 Profile shift coefficient II-253
13.1.9 Quality II-256
13.1.10 Geometry details II-259
13.1.11 Material and lubrication II-260
13.2 Load II-266
13.2.1 Calculation method II-266
13.2.2 Service life II-271
13.2.3 Application factor II-272
13.2.4 Power, torque and speed II-273
13.2.5 Strength details II-273
13.2.6 Strength details (AGMA) II-286
13.2.7 Define load spectrum II-287
13.2.8 Calculate scuffing II-289
13.3 Factors II-291
13.3.1 Transverse coefficient II-291
13.3.2 Dynamic factor II-291
13.3.3 Load distribution coefficient II-292
13.3.4 Alternating bending factor II-293
13.3.5 Face load factor II-296
13.3.6 Z-Y coefficients and the technology factor II-309
13.3.7 General calculation procedure for KHbeta as specified in ISO 6336-1, Appendix E. II-311
13.4 Reference profile II-312
13.4.1 Configuration II-312
13.4.2 Preliminary treatment and grinding allowance II-319
13.4.3 Tip alteration II-320
13.5 Tolerances II-322
13.5.1 Tooth thickness tolerance II-322
13.5.2 Tip diameter allowances II-324
13.5.3 Root diameter allowances II-324
13.5.4 Center distance tolerances II-325
13.5.5 Settings II-325
13.6 Modifications II-326

13.6.1 Define grinding wheel for gear n dialog window II-327
13.6.2 Type of modification II-328
13.6.3 Underlying principles of calculation II-329
13.6.4 Profile modifications II-331
13.6.5 Tooth trace modifications II-336
13.6.6 Sizing modifications II-342
13.6.7 Notes about profile modification II-346
13.7 Tooth form II-347
13.7.1 Context menu II-348
13.7.2 Operations II-349
13.8 Flank breaking II-369
13.9 Contact Analysis II-371
13.9.1 Theory of Contact Analysis II-373
13.9.2 Discretized model II-376
13.9.3 Smoothing the tooth form curvature to calculate Hertzian pressure in the contact analysis II-376
13.9.4 Reduced stiffness on the side edges II-378
13.9.5 Linking the individual slices II-379
13.9.6 Wear iteration II-379
13.9.7 Calculation of contact analysis taking the shafts into account .. II-380
13.9.8 Contact analysis of a planet system II-385
13.10 Gear pump II-393
13.11 Operating backlash II-396
13.11.1 Reference temperature II-398
13.11.2 Relative water absorption during swelling II-398
13.11.3 Coefficient of thermal expansion for housing II-399
13.12 Master gear II-400
13.13 AGMA 925 ..... II-402
13.14 Rough sizing II-404
13.15 Fine Sizing II-409
13.15.1 Necessary entries in the input window II-410
13.15.2 Conditions I II-410
13.15.3 Conditions II II-412

13.15.4 Results II-417
13.15.5 Graphics II-419
13.15.6 Geometry-Fine Sizing for 3 gears II-420
13.15.7 Additional strength calculation of all variants II-420
13.16 Measurement grid II-421
13.17 Profile modification optimization II-424
13.17.1 Conditions I/II II-424
13.17.2 Results II-424
13.17.3 Graphics II-425
13.17.4 Report II-425
13.18 Settings II-428
13.18.1 General II-428
13.18.2 Plastic II-431
13.18.3 Planets II-433
13.18.4 Sizings II-434
13.18.5 Calculations II-435
13.18.6 Required safeties II-441
13.18.7 Contact analysis/Face load factor II-442
13.18.8 Summary II-443
13.18.9 Generation of 3D model II-443
13.19 Tooth thickness II-447

14 Bevel and Hypoid gears II-448

14.1 Underlying principles of calculation II-449
14.1.1 General II-449
14.1.2 Overview of the bevel gear manufacturing process and the terminology used in it II-449
14.1.3 Calculation in accordance with Klingelnberg, Gleason and Oerlikon II-450
14.2 Basic data II-452
14.2.1 Type II-452
14.2.2 Normal module (middle) II-457
14.2.3 Reference diameter gear 2 ..... II-458

14.2.4 Pressure angle at normal section II-458
14.2.5 Pressure angle driving/driven flank: Hypoid gears II-458
14.2.6 Helix angle II-460
14.2.7 Shaft angle II-461
14.2.8 Offset (Center dist.) II-462
14.2.9 Number of teeth II-462
14.2.10 Facewidth II-463
14.2.11 Profile shift coefficient II-463
14.2.12 Tooth thickness modification factor II-463
14.2.13 Quality II-464
14.2.14 Addendum angle and root angle II-465
14.2.15 Angle modifications II-466
14.2.16 Geometry details II-467
14.2.17 Manufacturing process II-468
14.3 Manufacture II-469
14.3.1 Cutter radius II-469
14.3.2 Number of blade groups, tool II-469
14.4 Load II-470
14.4.1 Methods used for strength calculation II-470
14.4.2 Required service life II-474
14.4.3 Power, torque and speed II-474
14.4.4 Strength details II-475
14.4.5 Application factor II-476
14.5 Factors II-477
14.5.1 Bearing application factor II-477
14.5.2 Dynamic factor II-478
14.5.3 Bevel gear factor at flank and root II-478
14.6 Reference profile II-480
14.6.1 Default values for tip base clearance II-480
14.6.2 Default values for addendum coefficients II-480
14.7 Rough sizing II-481
14.7.1 Face width ratio II-481
14.7.2 Module ratio II-482

14.8 Notes on calculations in accordance with the Klingelnberg standard II-483
14.8.1 Bevel gears with cyclo-palloid® gear teeth II-483
14.8.2 Hypoid gears with cyclo-palloid gear teeth..... II-483
14.8.3 Normal module ranges for Klingelnberg machines (cyclo-palloid) II-484
14.8.4 Bevel gears with Palloid toothing II-485
14.8.5 Definitions and dimensions of standard cutters for palloid toothing II-486
14.8.6 Minimum safeties II-487
14.8.7 Surface roughness at tooth root II-487
14.8.8 Accuracy grade bevel gears..... II-487
14.8.9 Characteristic number..... II-487
14.9 Settings II-489
14.9.1 Calculations II-489

15 Face gears II-490
15.1 Underlying principles of calculation..... II-491
15.2 Basic data..... II-494
15.2.1 Normal module..... II-494
15.2.2 Pressure angle at normal section II-496
15.2.3 Helix angle at reference circle..... II-496
15.2.4 Axial offset..... II-497
15.2.5 Profile shift coefficient (on the pinion) II-497
15.2.6 Quality..... II-498
15.2.7 Geometry details II-499
15.2.8 Material and lubrication II-500
15.3 Load..... II-501
15.3.1 Methods used for strength calculation..... II-501
15.3.2 Service life..... II-503
15.3.3 Power, torque and speed..... II-503
15.3.4 Application factor..... II-503
15.4 Factors II-505
15.4.1 Face load factor II-505

15.5 Modifications II-506
15.5.1 Addendum reduction II-506
15.5.2 Type of modification II-506
15.6 Settings II-507
15.6.1 General II-507
15.6.2 Sizings II-508
15.7 Notes on face gear calculation II-509
15.7.1 Dimensioning II-509
15.7.2 Pinion - Face gear with Z1 > Z2 ..... II-510

16 Worms with enveloping worm wheels II-511

16.1 Underlying principles of calculation II-512
16.2 Basic data II-514
16.2.1 Axial/transverse module II-514
16.2.2 Pressure angle at normal section II-514
16.2.3 Lead angle at reference circle II-514
16.2.4 Center distance II-515
16.2.5 Number of teeth II-515
16.2.6 Facewidth II-516
16.2.7 Profile shift coefficient II-516
16.2.8 Tooth thickness modification factor II-517
16.2.9 Quality II-517
16.2.10 Geometry details II-518
16.2.11 Material and lubrication II-519
16.3 Load II-521
16.3.1 Strength calculation methods II-521
16.3.2 Service life II-522
16.3.3 Application factor II-522
16.3.4 Permissible decrease in quality II-522
16.3.5 Power, torque and speed II-523
16.3.6 Strength details II-523
16.4 Tolerances II-526
16.5 Settings II-527

16.5.1 General II-527
16.5.2 Reference gearing II-528
16.5.3 Calculations II-529
16.5.4 Required safeties II-530

17 Crossed helical gears and precision mechanics worms II-532
17.1 Underlying principles of calculation II-533
17.2 Basic data II-534
17.2.1 Normal module II-534
17.2.2 Pressure angle at normal section II-534
17.2.3 Helix angle reference circle gear 1 ..... II-534
17.2.4 Center distance II-535
17.2.5 Facewidth II-535
17.2.6 Profile shift coefficient II-535
17.2.7 Quality II-535
17.2.8 Define details of geometry II-536
17.2.9 Material and lubrication II-537
17.3 Load II-538
17.3.1 Methods used for strength calculation II-538
17.3.2 Service life II-541
17.3.3 Application factor II-542
17.3.4 Power, torque and speed II-542
17.3.5 Strength details II-543
17.4 Settings II-548
17.5 Notes II-549
17.5.1 Checking the contact pattern II-549

18 18. Beveloid gears II-550

19 Non circular gears II-551
19.1 Input data II-552
19.1.1 Geometry II-552
19.1.2 Tolerances II-555

19.1.3 Reference profile II-555
19.2 How to use KISSsoft II-557
19.2.1 Angle error II-557
19.2.2 Checking the meshing II-557
19.2.3 Improve tooth form II-558
19.2.4 Accuracy of the tooth form II-558
19.2.5 Export individual teeth II-559
19.2.6 Report II-560
19.2.7 Temporary files II-560

20 Report menu II-562
20.1 Drawing data II-563
20.2 Manufacturing tolerances II-564
20.3 Summary II-565
20.4 Service life II-566
20.5 Sizing of torque II-567
20.6 Proposal for the hardening depth EHT II-568

21 Graphics menu II-569
21.1 AGMA 925 ..... II-573
21.1.1 Thickness of lubrication film and specific oil film thickness II-573
21.2 2D geometry II-574
21.2.1 Gear tooth forms II-574
21.2.2 Gear tool II-575
21.2.3 Manufacturing a gear II-575
21.2.4 Meshing II-575
21.2.5 Profile and tooth trace diagram II-576
21.2.6 Flank curvature radii II-580
21.2.7 Angle of flank normal II-581
21.2.8 Drawing II-581
21.2.9 Assembly II-581
21.2.10 Manufacturing drawing II-581
21.3 3D geometry II-584

21.3.1 Tooth system II-585
21.3.2 Tooth form II-585
21.4 Evaluation II-586
21.4.1 Specific sliding II-586
21.4.2 Flash temperature II-587
21.4.3 Hardening depth II-588
21.4.4 Woehler line for material II-589
21.4.5 Safety factor curves II-590
21.4.6 Oil viscosity, depending on temperature II-590
21.4.7 Theoretical contact stiffness II-591
21.4.8 Contact line (face gear) II-592
21.4.9 Stress curve (face gear) II-593
21.4.10 Scuffing and sliding velocity (face gear) II-594
21.5 Contact analysis II-596
21.5.1 Axis alignment II-596
21.5.2 Transmission error II-596
21.5.3 Transmission error acceleration II-597
21.5.4 Transmission error amplitude II-598
21.5.5 Contact lines on tooth flank II-598
21.5.6 Normal force curve II-598
21.5.7 Normal force distribution II-598
21.5.8 Torque curve II-599
21.5.9 Single contact stiffness II-599
21.5.10 Stiffness curve II-599
21.5.11 Amplitude of contact stiffness II-600
21.5.12 Bearing force curve and direction of the bearing forces II-600
21.5.13 Kinematics II-600
21.5.14 Specific sliding II-601
21.5.15 Power loss II-601
21.5.16 Heat development II-601
21.5.17 Stress curve II-601
21.5.18 Flash temperature II-602
21.5.19 Safety against micropitting II-602

21.5.20 Wear II-604
21.6 Gear pump II-607
21.7 3D export II-608
21.8 Settings II-609

22 Graphics list II-610

23 Answers to Frequently Asked Questions II-611
23.1 Answers concerning geometry calculation II-612
23.1.1 Precision mechanics II-612
23.1.2 Deep toothing or cylindrical gears with a high transverse contact ratio II-612
23.1.3 Pairing an external gear to an inside gear that has a slightly different number of teeth II-613
23.1.4 Undercut or insufficient effective involute II-613
23.1.5 Tooth thickness at tip II-614
23.1.6 Special toothing II-614
23.1.7 Calculating cylindrical gears manufactured using tools specified in DIN 3972 ..... II-614
23.1.8 Variations in rolling as defined in DIN 58405 ..... II-615
23.1.9 Automatic change of reference profiles II-616
23.1.10 Non-identical (mirrored symmetry) tooth flanks II-616
23.1.11 Internal teeth - differences in the reference profile if you select different configurations II-617
23.1.12 Effect of profile modifications II-618
23.1.13 Number of teeth with common multiples II-619
23.1.14 Allowances for racks II-620
23.2 Answers to questions about strength calculation II-621
23.2.1 Differences between different gear calculation programs II-621
23.2.2 Difference between cylindrical gear calculation following ISO 6336 or DIN 3990 ..... II-621
23.2.3 Calculation using Methods B or C (DIN 3990, 3991) II-622
23.2.4 Required safeties for cylindrical gears II-622

23.2.5 Insufficient scuffing safety II-623
23.2.6 Material pairing factor (strengthening an unhardened gear) II-624
23.2.7 Defining the scoring load level (oil specification) II-624
23.2.8 The influence of the face load factor KHß for tooth trace deviation fma is due to a manufacturing error. II-624
23.2.9 Load spectrum with changing torque II-625
23.2.10 Strength calculation with several geometries on one gear II-627
23.2.11 Bevel gears: – Determine permitted overloads II-628
23.2.12 Take shot peening data into account in calculating the strength of toothed gears II-629
23.2.13 Calculation in accordance with AGMA 421.06 (High Speed Gears) II-630
23.2.14 Comparison of a FEM calculation with spiral-toothed gear wheel calculation II-631
23.2.15 Estimate the strength of asymmetrical spur gear toothings II-631
23.2.16 Determine the equivalent torque (for load spectra) II-632
23.2.17 Check changes in safeties if the center distance changes II-632
23.2.18 Warning: "Notch parameter qs …. outside RANGE (1.0 to 8.0) ..." II-633
23.3 Abbreviations used in gear calculation II-634

III Shafts and Bearings III-641

24 Defining Shafts III-642
24.1 Input window III-645
24.1.1 Shaft editor III-645
24.1.2 Elements tree III-647
24.1.3 Elements list III-648
24.1.4 Elements editor III-649
24.2 Element overview III-650
24.2.1 The Shaft element III-650
24.2.2 Outer contour III-655
24.2.3 Inner contour III-662

24.2.4 Forces III-662
24.2.5 Bearing III-667
24.2.6 Connection elements III-670
24.2.7 Cross sections III-672
24.3 Basic data III-673
24.3.1 Position of shaft axis in space III-673
24.3.2 Number of eigenfrequencies III-674
24.3.3 Number of buckling cases III-674
24.3.4 Speed III-674
24.3.5 Sense of rotation III-675
24.3.6 Reference temperature III-675
24.3.7 Temperature of housing III-676
24.3.8 Lubricant temperature III-676
24.3.9 Load spectra III-676
24.3.10 Gears III-677
24.3.11 Roller bearing III-677
24.3.12 Tolerance field III-678
24.3.13 Modified rating life according ISO 281 ..... III-678
24.3.14 Consider weight III-679
24.3.15 Consider gyroscopic effect III-679
24.3.16 Housing material III-679
24.3.17 Lubrication III-679
24.3.18 Impurity III-679
24.4 Module specific settings III-681
24.4.1 Non-linear shaft III-681
24.4.2 Take into account deformation due to shearing and shear correction coefficient III-682
24.4.3 Standard radius at shoulder III-682
24.4.4 Node density III-683
24.4.5 Iterative calculation of load distribution III-684
24.4.6 Input different load cycles for bending and torsion (for finite life calculations) III-684
24.4.7 Axial clearance III-684

24.4.8 Failure probability III-685
24.4.9 Required service life III-685
24.4.10 Maximum service life coefficient III-685
24.4.11 Display critical bearing III-685
24.4.12 Surface roughness of housing III-686
24.4.13 Calculation method for friction III-686
24.4.14 Type of oil lubrication III-686
24.4.15 Bearing manufacturer III-686
24.4.16 Show coordinate system III-686
24.4.17 Show automatic dimensioning III-686
24.4.18 Equivalent stress for sizings III-686
24.4.19 Maximum deflection for sizings III-687

25 Calculating Shafts III-688

25.1 Deflection and Bearing Forces, Distribution and Force of Torque III-690
25.1.1 Calculating force on bearings with a contact angle III-692
25.2 Eigenfrequency III-694
25.2.1 Bending critical speed III-695
25.2.2 Torsion critical speed III-695
25.3 Buckling III-696
25.4 Strength III-697
25.4.1 Calculation method III-698
25.4.2 Type of calculation III-702
25.4.3 Service life III-704
25.4.4 Strength parameters in accordance with Hänchen and Decker III-704
25.4.5 Strength parameters in accordance with FKM III-705
25.4.6 Strength parameters in accordance with DIN III-707
25.4.7 Stress III-708
25.4.8 Stress ratio III-708
25.4.9 Load factor for static analysis III-709
25.4.10 Load factor for endurance calculation III-710
25.4.11 Cross sections III-710
25.4.12 Sizing III-712

25.4.13 Cross section types III-712
25.4.14 General entries III-718
25.5 Tooth trace modification III-719
25.6 Campbell diagram III-722

26 Bearing calculation General III-724
26.1 Classification of bearings III-725
26.1.1 Properties III-725

27 Roller bearing III-727
27.1 Selecting the type of roller bearing III-728
27.1.1 Properties of the most important bearing types III-728
27.1.2 Comparing types III-730
27.2 Load capacity of roller bearings III-733
27.2.1 Dynamic load capacity III-733
27.2.2 Permissible static stress III-733
27.2.3 Bearing calculation with inner geometry III-734
27.3 Thermally permissible service speed III-735
27.3.1 Thermal reference speed III-735
27.3.2 Process for calculating thermally permitted operating speed (DIN 732-2) III-737
27.4 Moment of friction III-739
27.4.1 Calculation in accordance with SKF Catalog 2004 ..... III-739
27.4.2 Calculation in accordance with SKF Catalog 1994 ..... III-742
27.5 Maximum Speeds III-743
27.6 Service life III-744
27.6.1 Extended service life calculation in accordance with Supplement to DIN ISO 281 (2007) III-744
27.6.2 Service life calculation with load spectra III-745
27.7 Failure probability III-747
27.8 Bearings with radial and/or axial force III-747
27.9 Calculating axial forces on bearings in face-to-face or back-to-back arrangements III-748

27.10 Oil level and Lubrication type III-750

28 Roller Bearings (Inner Geometry) III-751

28.1 Bearing data tab III-752

28.1.1 File interface III-752

28.1.2 Bearing data III-754

28.2 Load tab III-757

28.2.1 Load III-757

28.2.2 Enhanced service life calculation in accordance with ISO 281 III-757

28.3 Graphics III-759

28.3.1 Load distribution III-759

28.3.2 Pressure curve III-759

28.3.3 Stiffness curve III-761

28.3.4 Pressure curve for each rolling body III-762

29 Hydrodynamic plain radial bearing III-763

29.1 Calculation methods III-764

29.2 Module specific inputs III-765

29.3 Thermal expansion coefficients III-766

29.4 Average surface pressure III-767

29.5 Lubrication arrangement III-768

29.6 Heat transfer surface III-772

29.7 Heat transfer coefficient III-773

29.8 Oil temperatures III-774

29.9 Sizing the bearing clearance III-775

29.10 Sommerfeld number III-776

29.11 Bearing width III-777

29.12 Permissible thickness of lubrication film III-778

30 Hydrodynamic plain thrust bearing III-779

30.1 Calculation III-782

30.2 Sizings III-783

30.3 Calculation of volume specific heat III-784

30.4 Limiting values in the calculation.....III-785

31 Answers to Frequently Asked Questions III-786

31.1 Intersecting notch effects.....III-787

31.2 Notch effects on hollow shafts.....III-788

31.2.1 Notches on the outer contour.....III-788

31.2.2 Notches on the inner contour.....III-788

31.3 Fatigue Limits for New Materials.....III-789

31.4 Taking double helical gearing into account in the shaft calculation.....III-790

IV Connections IV-791

32 Cylindrical interference fit.....IV-792

32.1 Inputting Tolerances IV-795

32.2 Coefficients of friction.....IV-796

32.3 Variable hub external diameter.....IV-798

32.4 Convert external pressure with multiple interference fit IV-799

32.5 Materials IV-800

32.6 Settings IV-801

32.7 Sizings IV-803

33 Conical interference fit IV-804

33.1 Calculation.....IV-806

33.2 Application factor IV-807

33.3 Axial spanning with nut.....IV-808

33.4 Variable external diameter of the hub.....IV-810

33.5 Conicity IV-811

33.6 Materials IV-812

33.7 Settings IV-813

33.8 Sizings IV-814

34 Clamped connections IV-815

34.1 Calculations IV-816
34.2 Sizings IV-817
34.3 Settings IV-817
34.4 Materials IV-818

35 Key.....IV-819
35.1 Main screen.....IV-821
35.1.1 Additional inputs for DIN 6892 Method B IV-822
35.2 Application factor IV-824
35.3 Load factor.....IV-826
35.4 Own inputs.....IV-827
35.5 Permissible pressure IV-828
35.6 Materials IV-829
35.7 Settings IV-830
35.8 Sizings IV-831

36 Spline shaft IV-832
36.1 Standard profiles.....IV-833
36.2 Application factor IV-834
36.3 Torque curve/Frequency of change of load direction IV-835
36.4 Occurring flank pressure.....IV-836
36.5 Length factor.....IV-837
36.6 Share factor.....IV-838
36.7 Permissible pressure IV-839
36.8 Materials IV-840
36.9 Settings IV-841
36.10 Sizings.....IV-842

37 Spline (strength) IV-843
37.1 Standard profiles.....IV-844
37.2 Application factor IV-846
37.3 Torque curve/Frequency of change of load direction IV-847
37.4 Occurring flank pressure.....IV-848

37.5 Length factor.....IV-849
37.6 Share factor.....IV-850
37.7 Permissible pressure IV-851
37.8 Materials IV-852
37.9 Settings IV-853
37.10 Sizings.....IV-854

38 Spline (geometry and strength) IV-855

38.1 Underlying principles of calculation.....IV-856
38.1.1 General IV-856
38.1.2 Calculation of spline connections as described in DIN 5480 with diameter centering IV-856
38.2 Basic data.....IV-858
38.2.1 Geometry standards.....IV-858
38.2.2 Normal module.....IV-859
38.2.3 Pressure angle at normal section an IV-859
38.2.4 Number of teeth.....IV-860
38.2.5 Profile shift coefficient.....IV-860
38.2.6 Quality.....IV-861
38.2.7 Geometry details IV-862
38.2.8 Strength calculation method.....IV-863
38.2.9 Application factor.....IV-863
38.2.10 Resulting shearing force.....IV-864
38.2.11 Define details of strength IV-865
38.2.12 Materials.....IV-868
38.3 Tolerances.....IV-869
38.3.1 Tooth thickness tolerance.....IV-869
38.3.2 Effective/Actual IV-870
38.3.3 Ball/pin diameter shaft/hub IV-871
38.4 Templates.....IV-872
38.5 Tooth form.....IV-873

39 Polygon.....IV-874

39.1 Standard profiles IV-875
39.2 Application factor IV-876
39.3 Torque curve/Frequency of change of load direction IV-877
39.4 Occurring flank pressure IV-878
39.5 Permissible pressure IV-880
39.6 Materials IV-881
39.7 Settings IV-882
39.8 Sizings IV-883
39.9 Graphics IV-884

40 Woodruff Key IV-885

40.1 Standard profiles IV-886
40.2 Application factor IV-888
40.3 Torque curve/Frequency of change of load direction IV-889
40.4 Occurring flank pressure IV-890
40.5 Length factor IV-891
40.6 Share factor IV-892
40.7 Permissible pressure IV-893
40.8 Materials IV-894
40.9 Settings IV-895
40.10 Sizings IV-896

41 Bolts and pins IV-897

41.1 Influence factors IV-899
41.2 Materials IV-900
41.3 Settings IV-901
41.4 Permitted values IV-902
41.5 Sizings IV-903

42 Bolts IV-904

42.1 Special features in KISSsoft IV-906
42.2 Basic data inputs IV-907
42.2.1 Operating data IV-907

42.2.2 Bolt data IV-917
42.2.3 Type of bolted joint IV-920
42.2.4 Washers IV-922
42.2.5 Tightening technique IV-922
42.3 Clamped parts inputs IV-924
42.3.1 Geometry of clamped parts IV-924
42.3.2 Distances for eccentric clamping/load IV-927
42.3.3 Load application IV-927
42.4 Constraints data IV-929
42.4.1 Technical Explanations IV-930
42.4.2 Coefficients of friction IV-931
42.4.3 Swing angle controlled tightening IV-932
42.5 Stripping strength IV-933
42.6 Settings IV-934

43 Welded joints IV-937
43.1 Welded joints IV-938
43.2 Seam length IV-940
43.3 Welded seam equivalent stress IV-941
43.4 Weld seam boundary stress IV-942
43.5 Part safety coefficient IV-943
43.6 Boundary safety coefficient IV-944
43.7 Materials IV-945

44 Glued and soldered joints IV-946
44.1 Basic materials IV-948
44.2 Settings IV-949
44.3 Sizings IV-950
44.4 Bracket connection IV-951
44.5 Shaft connections IV-952

45 Retaining rings (self-locking rings, Seeger rings) IV-953
45.1 Basic data IV-954

45.2 Automatic calculation of load factor q IV-956
45.3 Automatic calculation of the dishing angle ψ IV-957
45.4 Module specific settings IV-958

46 Answers to Frequently Asked Questions IV-959
46.1 Adding new bolt types to the database IV-960
46.1.1 Extending an existing bolt series.....IV-960
46.1.2 Creating a new bolt type.....IV-962

V Springs V-963

47 Compression springs V-964
47.1 Strength values V-965
47.2 Shear stress values V-965
47.3 Bearings coefficient V-966
47.4 Materials V-966
47.5 Tolerances V-967
47.6 Relaxation.....V-968
47.7 Drawing data V-969
47.8 Sizings V-969

48 Tension springs V-970
48.1 Strength values V-971
48.2 Shear stress values V-971
48.3 Manufacturing type V-972
48.4 Eyes screen V-972
48.5 Materials V-974
48.6 Settings V-975
48.7 Tolerances V-975
48.8 Relaxation.....V-976
48.9 Drawing data V-977
48.10 Sizings.....V-978

49 Leg springs V-979
49.1 Strength values V-980
49.2 Bending stress values V-981
49.3 Spring design V-981
49.4 Assumptions made for the calculation V-982
49.5 Materials V-982
49.6 Tolerances V-983
49.7 Drawing data V-983
49.8 Sizings V-984

50 Disc springs V-985
50.1 Strength values V-986
50.2 Stress values V-986
50.3 Materials V-987
50.4 Calculate number V-988
50.5 Limit dimensions V-989

51 Torsion-bar springs V-990
51.1 Head forms V-992
51.2 Strength values V-992
51.3 Shear stress V-993
51.4 Limiting values V-993
51.5 Sizings V-994

VI Belts and chain drives VI-995

52 V-belt VI-996
52.1 V-belts data VI-997
52.2 V-belt standards VI-997
52.3 Configuring Tensioning Pulleys VI-998
52.4 Application factor f1 ..... VI-998
52.5 Center distance VI-998
52.6 Belt length VI-999

52.7 Effective number of V-belts VI-999
52.8 Tensioning pulley diameter VI-999
52.9 Position of tensioning pulley (x/y).....VI-1000
52.10 Inspecting V-belts VI-1001

53 Toothed belts VI-1002
53.1 Technical notes (toothed belts) VI-1003
53.2 Toothed belt standard VI-1004
53.3 Possible Sizings/Suggestions.....VI-1005
53.4 Configuring Tensioning pulleys VI-1005
53.5 Application factor and summand for operational behavior VI-1006
53.6 Center distance.....VI-1006
53.7 Belt length and number of teeth on belt.....VI-1007
53.8 Effective belt width.....VI-1007
53.9 Tensioning pulley tooth number VI-1007
53.10 Position of the tensioning pulley x/y.....VI-1009

54 Chain drives VI-1010
54.1 Sizings VI-1010
54.2 Tensioning pulleys.....VI-1011
54.3 Standard.....VI-1011
54.4 Chain type.....VI-1011
54.5 Number of strands.....VI-1011
54.6 Application factor VI-1012
54.7 Speed/number of teeth/transmission ratio.....VI-1012
54.8 Configuration VI-1012
54.9 Center distance VI-1013
54.10 Polygon effect VI-1013
54.11 Number of links VI-1014
54.12 Geometry of chain sprockets.....VI-1015

VII Automotive VII-1016

55 Synchronization VII-1016
55.1 Geometry VII-1017
55.2 Operating data VII-1018

56 Friction clutches VII-1018
56.1 Calculation VII-1021
56.2 Definition of spring forces VII-1024
56.3 Definition coefficients of sliding friction and velocities VII-1025
56.4 Graphics VII-1026
56.5 Settings VII-1027

VIII Various VIII-1028

57 Tolerance calculation VIII-1029

58 Strength verification with local stresses VIII-1030
58.1 General VIII-1031
58.1.1 Functionality of the software VIII-1031
58.1.2 Areas of application for the FKM guideline VIII-1031
58.2 Background VIII-1033
58.2.1 The FKM Guideline: Rechnerischer Festigkeitsnachweis für Maschinenbauteile VIII-1033
58.2.2 Usefulness of the service life calculation VIII-1033
58.3 Implementation in KISSsoft VIII-1037
58.3.1 Main screen VIII-1037
58.3.2 Load cases VIII-1039
58.3.3 Woehler line VIII-1039
58.3.4 Number of load cycles VIII-1039
58.3.5 Temperature VIII-1040
58.3.6 Temperature duration VIII-1040
58.3.7 Protective layer thickness, aluminum, chapter 4.3.4, Figure 4.3.4 VIII-1040
58.3.8 Stress ratios VIII-1040

58.3.9 Spectra VIII-1042
58.3.10 Surface factor KV, section 4.3.3, Table 4.3.7 ..... VIII-1042
58.4 Materials VIII-1043
58.4.1 Surface roughness VIII-1043
58.4.2 Settings VIII-1044

59 Hertzian pressure VIII-1049
59.1 Settings VIII-1051

60 Hardness conversion VIII-1052

61 Linear drive train VIII-1053
61.1 Calculation VIII-1056
61.2 Sizings VIII-1061
61.3 Settings VIII-1061
61.4 Materials VIII-1062

IX KISSsys IX-1064

62 KISSsys: Calculation Systems IX-1065
62.1 General IX-1066
62.1.1 Structure of KISSsys IX-1066
62.1.2 Ways in which KISSsys can be used IX-1066
62.2 The user interface IX-1068
62.2.1 Tree view IX-1068
62.2.2 Diagram view IX-1069
62.2.3 Table view IX-1069
62.2.4 3D view IX-1070
62.2.5 Message output IX-1070
62.3 Creating Models in KISSsys IX-1071
62.3.1 Classic method IX-1072
62.3.2 Element Assistant IX-1073

62.3.3 System Assistant IX-1074
62.3.4 Creating and modifying tables IX-1075
62.3.5 Adding variables in tables IX-1076
62.3.6 Individual names for elements IX-1078
62.3.7 Menus, context menus and the Tool Bar IX-1079
62.4 Extended functionality for developers IX-1080
62.4.1 Properties dialog.....IX-1080
62.4.2 Table view IX-1081
62.5 The existing elements IX-1083
62.5.1 Variables.....IX-1083
62.5.2 Calculation elements IX-1084
62.5.3 Elements for shafts IX-1086
62.5.4 Connection elements IX-1087
62.5.5 Displaying elements in 3D graphics.....IX-1088
62.5.6 System settings IX-1089
62.6 Programming in the Interpreter.....IX-1090
62.6.1 Expressions in variables IX-1090
62.6.2 Functions IX-1091
62.6.3 Important service functions IX-1093
62.6.4 Variable dialogs.....IX-1093
62.6.5 Defining 2D graphics IX-1102

X Bibliography and Index X-1105

63 Bibliography.....X-1106

XI Index XI-1113

Part I

General

Chapter 1
Installing KISSsoft

1.1 Basic installation

After you have inserted the KISSsoft CD in the appropriate disk drive, the setup program starts automatically. If it does not, you can run the setup.exe file directly in the CD root directory by double-clicking on it.

The setup program guides you through the installation process step by step. All you need to do is select an installation folder and the required language for the installation. If you change the default installation folder, it is advisable to include the version descriptor as part of the directory name of the other installation folder (e.g. C:/Programs/KISSsoft xx-20xx).

At the end of the installation we recommend that you install the latest Service Pack (patch). Download the latest patch http://www.kisssoft.ch/patches.php from our website. You can choose between an installation program (*.exe) and zipped files (*.zip). The installation program automatically copies the necessary files after you specify which installation folder it is to use. However, not all companies permit exe files to be downloaded. In this case, you must unpack the ZIP file and manually copy the files it contains into your installation folder. Any files that are already present must be overwritten by the ones contained in the patch.

After you have installed KISSsoft you need to license (see page I-42) it. If KISSsoft is not licensed, it will only run as a demo version.

NOTE:

If you are installing KISSsoft on a server, we recommend that you perform the installation from a client (workstation computer). Consequently, all necessary directory entries will automatically be added to the KISS.ini (see page I-51) file correctly. Otherwise, you will have to change these directory entries from the local drive name (e.g. C:/...) to the appropriate share name in the network, later, manually, using an editor.

1.2 Downloading a license file

1. Go to our website, www.KISSsoft.ch, and click on the Service/Support page link on the left. There, you will find a link to the "customer zone". Click on the link. You will see the Customer Zone web page. In that page, on the top right-hand side, enter your license number in the License Number field, and click on "Open".
2. A login window will open, in which you enter your license number, and also your download password, again. If you do not have this password, please get in touch with your commercial contact representative or contact directly KISSsoft via email on info@KISSsoft.CH or phone number +41 55 254 20 53.
3. You are now in your personal download area. Save the lizenzxxxx.lic file in the license directory of your KISSsoft installation.

NOTE:

It may be that your personal download area contains license files for different versions of KISSsoft. Please make sure you select the correct license file for the system version you have just installed.

1.3 Licensing

After you have performed the KISSsoft Installation (see page I-40) you must license the software either by downloading a license file or activating the program's license. Please read the relevant section for your license type.

1.3.1 Test version

1. If you start KISSsoft from the client (workstation computer), the user account for the test version will become active.
2. Select `License tool` in the `Extras` menu and click on the Activate license tab.
3. **Activate online:** If your computer has Internet access, and you have received an online code from us, enter this code under the `Release Test or Student version` option and then click on `Activate license`.
4. **Direct activation:** Under the `Activate test version by phone` option you see a question code. Call the telephone number you see there and tell us this code. We will then give you the appropriate answer code. Input this in the corresponding field and click the `Activate license tab`.

1.3.2 Student version

1. Copy your license file (you will usually be given this by your high school) to your License directory (see page I-52).
2. Select `License tool` in the `Extras` menu and click on the Activate license tab.
3. Input your online code (which you will also be given by your high school) under the `Activate test or student version` option and click on `Activate license tab`.

1.3.3 Single user version with dongle (protection key)

1. Copy your license file (see page I-41) to your license directory (see page I-52).
2. Now, simply plug in the dongle supplied with the system.

NOTE

The single user version of KISSsoft can also be installed on a central server. Local clients (workstation computers) can then run the software directly from this server. Please note here that the dongle must always be plugged into each particular client.

1.3.4 Single user version with license code

1. Start KISSsoft from the client (workstation computer) for which the software is to be licensed.
2. Select `License tool` in the `Extras` menu and click on the Activate license tab.
3. Enter your contact data under the `Request license file` option and click on `Send` to send your computer-specific access data directly to us. Alternatively, you can first save this access data in a file and then send us this file by email.
4. You will receive an email as soon as we have created your license file.
5. Download your License file (see page I-41) and copy it to your License directory (see page I-52).

1.3.5 Network version with dongle (protection key)

For the network version with dongle a server program has to be installed in addition to the licensing of the KISSsoft installation.

1.3.5.1 Installation on the server

1. Copy the KISSsoft dongle/MxNet installation directory onto a server.
2. Start MxNet32 on the server. You will see a dongle icon in the task bar.
3. Double-click this icon to start the user interface.
4. Now enter `Application`: KISSsoft and any file with the file extension *.mx as the `server file`. The clients must have both read and write access to this file. Now click `New Entry` to add this entry.
5. Then click the `Active Users` button to check who is using KISSsoft. You can also reactivate a license that has already been used.

1.3.5.2 Licensing the KISSsoft system.

1. Copy your license file (see page I-41) to your license directory (see page I-52).
2. Complete the necessary details in the "ServerFile: serverfilepath" line after the checksum line in the license file. The "serverfilepath" is the path to the server file that is defined in the server program.

NOTE

The KISSsoft installation will also run if the client is not connected to the network and if the dongle is inserted in the client instead of in the server. You can also "check out" the license if you remove the dongle.

1.3.6 Network version with the license code

1. Start KISSsoft from a client (workstation computer).
2. Select `License tool` in the `Extras` menu and go to the General tab.
3. Select an access directory on a server. Please note: If you change this, you will need a new license.
4. Go to the Activate license tab.
5. Enter your contact data under the `Request license file` option and click on `Send` to send your computer-specific access data directly to us. Alternatively, you can first save this access data in a file and then send us this file by email.
6. You will receive an email as soon as we have created your license file.
7. Download your License file (see page I-41) and copy it to your License directory (see page I-51).

Chapter 2
Setting Up KISSsoft

2.1 Directory structure

If there are several users it is advisable to store shared data (**databases**, user-defined **report templates** and **standard files**) on one server. This ensures that, if there are changes and upgrades, all users will be able to work with one uniform set of data. To set this up, move the KDB, EXT and TEMPLATE directories onto a server that can be accessed by all users, and then tailor the corresponding variables, KDBDIR, EXTDIR and TEMPLATEDIR, in the KISS.ini (see page I-51) file.

In contrast, the temporary directories should be defined locally on the workstations for several users. Otherwise, the interim results of individual users might overwrite each other. For each installation, KISSsoft uses the temporary user directory in accordance with the operating system. The CADDIR and TEMPDIR variables can, however, be tailored in the KISS.ini (see page I-51) file.

If you want to open or save a calculation file or a report, KISSsoft offers you your personal **User directory** as the first choice storage location. This property saves you frequent searches in the directories on your system. You can define this user directory via the USERDIR variable in the KISS.ini (see page I-51) file. The user directory will be ignored if you have selected an Active working project (see page I-94). In this case, KISSsoft offers you the project directory as the first choice storage location.

2.2 Language settings

KISSsoft is available in five languages: English, French, German, Italian and Spanish. When you select a language, the program differentiates between the language used for the user interface and the language used for the reports. It is therefore possible to operate KISSsoft in one language and to simultaneously display reports in a different language. Messages will be displayed either in the same language as the user interface or as the reports.

For global language settings, you need to edit the KISS.ini file (see page I-52). Additionally, you can also quickly toggle between languages in the program by selecting `Extras > Language`, and then the required language. The user can change the language used for reports by selecting `Report > Settings`.

2.3 System of units

KISSsoft recognizes two unit systems: the metric system and the US Customary Units system. For global language settings, you need to edit the KISS.ini file (see page I-52). You can also quickly toggle between systems of units in the program by selecting `Extras > System of units`. In addition to changing the system of units, it is possible to switch the unit used for a particular value input field (see page I-87).

2.4 Defining your own default files

Anyone who frequently carries out the same, or at least similar, calculations has to repeatedly enter the same values into selection lists and value input fields. Thanks to default files, KISSsoft makes your work considerably easier here. For each calculation module, there is an internal default setting for all values. If, however, you have defined your own default file, this default file will be used when you open a calculation module or load a new file.

To define a default file, you open a new file in the corresponding calculation module and enter your default settings. Click `File > Save as template` to transfer your values to the template file. All template files will be saved in the directory that has been defined as TEMPLATEDIR (see page I-51).

Default files can also be defined as project-specific. To define special standards for a project (see page I-91), select this project in the project tree (see page I-73) and open its properties by selecting `Project > Properties`. There, select `Use own templates for this project` and specify a directory for the default files. To define the default files you must select this project as the Active working project (see page I-94).

2.5 Rights

You can restrict the rights for selected areas of KISSsoft for some users.

Right	Implementation
Changes to the general settings	Write protect the KISS.ini (see page I-51) file
Changes or additions in the databases	Write protect databases (files of the type *.kdb) as well as the directories DAT and EXT/DAT (but write rights for KDBDIR (see page I-51) should be retained)
Changes to the report templates	Write protect RPT, EXT/RPT and EXT/RPU directories
Changes to the template files	Write protect the TEMPLATE directory

2.6 Global settings - KISS.ini

Global settings for KISSsoft are defined in the KISS.ini file, which is located directly in the installation folder. Most of these settings can also be defined directly in the software and are then saved to the KISS.ini file.

2.6.1 Definitions in [PATH]

Variable name	Explanation	Note
KISSDIR=<INIDIR>	The KISSsoft installation folder is generally defined with the INIDIR variable.	
HELPDIR	Directory for user manual and help figures	
DATADIR	Directory for files of the type *.dat	**Attention:** You should not carry out any upgrades or make any changes in this directory. Save your own files in the DAT subdirectory in the EXTDIR.
RPTDIR	Directory for report templates (*.rpt)	**Attention:** You should not carry out any upgrades or make any changes in this directory. Save your own files in the RPT subdirectory in the EXTDIR (described below).
USERDIR	Default directory for opening and saving	
CADDIR	Default directory for CAD export	Should be located locally on a workstation. %TEMP% sets the temporary directory to the default of the operating system.
TMPDIR	Directory for temporary files	Should be located locally on a workstation. %TEMP% sets the temporary directory to the default of the operating system.
KDBDIR	Directory for KISSsoft's databases (*.kdb)	If several users are using the system, we recommend you store the databases on one server to ensure data uniformity if there are changes and upgrades.
EXTDIR	Directory for user-defined report templates and additional DAT files	If there are several users, it is advisable to store this directory on one server.
TEMPLATEDIR	Directory for template files (STANDARD.*).	If there are several users, it is advisable to store this directory on one server.
LICDIR	Directory for the license files	You can install this directory on one server so that all the users can access the new license files.

Table 2.1: Table of variables used in the PATH environment

NOTE

You should have write permission for the directories set in these variables: TMPDIR, CADDIR, USRDIR and KDBDIR.

Depending on the configuration, you do not have write permission in the operating system directory: C:\Program Files\<KISSsoft Directory Name>. Any files you create are then diverted to the operating system's internal directories. Here, please select directories with write permission.

2.6.2 Definitions in [SETUP]

Variable name	Explanation	Values
USCUSTOMARYUNITS	Sets the system of units	0: metric, 1: US customary units
MATERIALSSTANDARD	Specifies the standard in which the materials are defined (configuration tool)	0 DIN, 1: BS, 2: AISI, 3: UNI, 4: AFNOR, 5: JIS, 6: CN
REPORTLANGUAGE	Sets the language in which reports are displayed	0: German, 1: English, 2: French, 3: Italian, 4: Spanish, 5: Russian, 6: Portuguese, 11: English with US Customary Units
SHOWCALCTIME	Shows the calculation time	0 No, 1: Yes
SHOWPROGRESSBAR	Shows the progress bar in time-intensive calculations	0 No, 1: Yes
DISPLAYLANGUAGE	Sets the language in which the user interface is displayed	0: German, 1: English, 2: French, 3: Italian, 4: Spanish, 5: Russian
DISPLAYFONTSIZE	Sets the font size in KISSsoft (FONT)	0 System size, otherwise the direct font size
MESSAGESINREPORTLANGUAGE	Sets the language in which messages are displayed	0: as interface, 1: as reports
MESSAGESSHOWSTATE	Defines which messages are to appear as a message box.	0: all, 1: Information only in message window, 2: Information and warnings only in message window
EDITOR	Path to the external editor	
USEEXTERNALEDITOR	Defines whether the external editor is to be used.	0 No, 1: Yes
DATEFORMAT	Date format, e.g. DD.MM.YYYY	

TIMEFORMAT	Time format, e.g. hh.mm.ss	
ENABLENETWORKING	Defines whether the network/Internet may be accessed (for example, to display innovations).	0 No, 1: Yes
CHECKFORUPDATES	Defines whether the system is to search for updates when the program starts.	0 No, 1: Yes
USETEMPORARYDATABASE	Defines whether the databases are to be copied to a temporary directory when the program starts	0 No, 1: Yes
RECENTFILESCOUNT	Number of most recently used files in the File menu	
FORCEEXCLUSIVEOPEN	Defines whether the files can only be opened exclusively.	0 No, 1: Yes
CALCONOPEN	Defines whether calculations are to be performed immediately on a file when it is loaded	0 No, 1: Yes, 2: no if KISSsoft. is started from KISSsys, otherwise yes
CALCINTERFACEOUT	Defines whether temporary reports for manufacturing data are to be written during the calculation	0 No, 1: Yes
ENABLEUSERSETTINGS	Defines whether the settings in kiss.ini can be overwritten by local settings.	0 No, 1: Yes
USEFILEEXPLORER	Defines whether the Explorer is to appear in the "View" menu list. This process will slow down KISSsoft considerably.	0 No, 1: Yes

Table 2.2: Table of variables used in the SETUP environment

2.6.3 Definitions in [REPORT]

Variable name	Description
SIZE	Number 0÷9 that specifies the length of the report
INCLUDEWARNINGS	0/1: Warnings are contained in the report
FONTSIZE	Number for the font size in the report
PAPERFORMAT	Paper format: A3, A4, A5, Letter, Legal
PAPERORIENTATION	0/1: Portrait/Landscape

PAPERMARGINLEFT	Distance from the left-hand page margin [mm]
PAPERMARGINRIGHT	Distance from the right-hand page margin [mm]
PAPERMARGINTOP	Distance from the top page margin [mm]
PAPERMARGINBOTTOM	Distance from the bottom page margin [mm]
COMPARE	0/1: Adds date/time to the report in comparison mode
SAVEFORMAT	0÷4: RTF, PDF, DOC, DOCX, TXT
LOGO	Picture file displayed in the header and footer
HEADER	Definition of the header
USEHEADERFORALLPAGES	0/1: header only on first page/on all pages
FOOTER	Definition of the footer
USEFOOTERFORALLPAGES	0/1: footer only on first page/on all pages

Table 2.3: Table of variables used in the REPORT environment

2.6.4 Definitions in [GRAPHICS]

Variable name	Description
BACKGROUND	0: black, 15: white (for more information, see Graphics > Settings)

Table 2.3b: Table of variables used in the GRAPHICS environment

2.6.5 Definitions in [LICENSE]

Variable name	Description
LOGGING	Number to activate the logging of license usage 0: no log file 1: Log in, Log out, No license, Used and Missing rights 2: Log in, Log out, No license 3: Log in, Log out, No license, Missing rights In network versions the user's uptime is also displayed in minutes when they log out.
LICENSELOGFILE	*.log file used to log the license usages
TIMEOUT	Duration until an unused floating license is activated on the network again

	[min]

Table 2.4: Table of variables used in the LICENSE environment

2.6.6 Definitions in [CADEXPORT]

Variable name	Description
USEDXFHEADER	0/1: DXF header will be used for DXF export
DXFVERSION	0/1: Version 12/15
INPUTLAYER	Name of the layer for import
OUTPUTLAYER	Name of the layer for export
DXFPOLYLINE	0/1/2: Uses polygonal course, lines or points for the export

Table 2.5: Table of variables used in the CADEXPORT environment

2.6.7 Definitions in [INTERFACES]

Variable name	Description
DEFAULT	Name of the CAD system:
	SolidEdge
	SolidWorks
	Inventor
	CATIA
	ProEngineer
	CoCreate
	Think3 interface
	HiCAD
GEAREXPORT3D	Displays the CAD system name in lists (see DEFAULT)
SYMMETRIC	0/1: Full tooth space/half tooth space mirrored (symmetrical) (default = 0)
SAVEFILENAME	0/1: Saves the entire file contents/Saves only the file name and the path (Default = 1)

Table 2.6: Table of variables used in the INTERFACES environment

2.6.8 Definitions in [SOLIDEDGE]

Variable name	Description
LIBRARY	Interface dll directory (kSoftSolidEdge.dll)
SMARTPATTERN	0/1: Fastpattern/Smartpattern
APPROXIMATION	1/2/3/4: Polygonal course (supported)/ Arcs (supported)/ Quadratic splines (supported) / Cubic splines (default)

Table 2.8: Table of variables used in the SOLIDEDGE environment

2.6.9 Definitions in [SOLIDWORKS]

Variable name	Description
LIBRARY	Interface dll directory (kSoftSolidWorks.dll)
SIMPLIFIEDPRESENTATIONNAME	Setting this variable generates a simplified gear with this name
APPROXIMATION	1/2/3/4: Polygonal course (supported)/ Arcs (supported)/ Quadratic splines (supported) / Cubic splines (default)

Table 2.9: Table of variables used in the SOLIDWORKS environment

2.6.10 Definitions in [INVENTOR]

Variable name	Description
LIBRARY	Interface dll directory (kSoftInventor.dll)
APPROXIMATION	1/2/3/4: Polygonal course (supported)/Arcs (default)/ Quadratic splines (not supported)/Cubic splines (not supported)

Table 2.10: Table of variables used in the INVENTOR environment

2.6.11 Definitions in [CATIA]

Variable name	Description
LIBRARY	Interface dll directory (kSoftCatia.dll)
LIBRARYSWMS	Directory of the interface manufacturer's *.dll files

LANGUAGEFILE	Directory of the interface manufacturer's *.ini files
DEBUG	Interface manufacturer's variable
DEBUGPATH	Interface manufacturer's variable
HELPFILE	Interface manufacturer's variable
LASTSETTING_CONSTRUCTION	Interface manufacturer's variable
LASTSETTING_GEARNAME	Interface manufacturer's variable
LASTSETTING_PRODUCTIONINFO	Interface manufacturer's variable
LASTSETTING_CALCINFO	Interface manufacturer's variable
LASTSETTING_FLAGINFO	Interface manufacturer's variable
APPROXIMATION	1/2/3/4: Polygonal course (not supported)/Arcs (not supported)/ Quadratic splines (default)/ Cubic splines (not supported)

Table 2.11: Table of variables used in the CATIA environment

2.6.12 Definitions in [PROENGINEER]

The ProEngineer interface has an individual subsection/menu for each version (for example, Wildfire 5, 32bit).

However, the definitions in "kiss.ini" are the same in every 3D interface to Creo Parametric (ProEngineer) chapters.

Variable name	Explanation
LIBRARY	Interface dll directory (kSoftProEngineer.dll)
INTERFACECOMMAND	Directory containing the interface manufacturer's *.exe files
USCUSTOMARYUNITS	0/1: System of units used in the metric or US Customary Units model
APPROXIMATION	1/2/3/4: Polygonal course (not supported)/arcs of circles (default)/quadratic splines (not supported)/cubic splines (not supported)

Table 2.12: Table of variables used in the PROENGINEER environment

2.6.13 Definition in [COCREATE]

Variable name	Explanation
LIBRARY	Interface dll directory (kSoftCoCreateCreo.dll)

INTERFACECOMMAND	Directory containing the interface manufacturer's *.exe files
APPROXIMATION	1/2/3/4: Polygonal course (not supported)/Arcs (not supported)/ Quadratic splines (default)/ Cubic splines (not supported)

Table 2.13: Table of variables used in the COCREATE environment

2.6.14 Definitions in [THINK3]

Variable name	Description
LIBRARY	Interface dll directory (kSoftThink3.dll)
INTERFACECOMMAND	Directory of the interface manufacturer's *.exe files
APPROXIMATION	1/2/3/4: Polygonal course (not supported)/Arcs (default)/ Quadratic splines (not supported)/Cubic splines (not supported)

Table 2.14: Table of variables used in the THINK3 environment

2.6.15 Definitions in [HICAD]

Variable name	Description
LIBRARY	Interface dll directory (kSoftHiCAD.dll)
APPROXIMATION	1/2/3/4: Polygonal course (not supported)/Arcs (default)/ Quadratic splines (not supported)/Cubic splines (not supported)

Table 2.15: Table of variables used in the HICAD environment

2.7 User-defined settings

User-defined settings can be reset via `Extras > Configuration tool`.

2.7.1 Configuration tool

In the `General` tab, you can select the older version's "kdb" database directory (`Update database` option). Click "Run" to transfer the data records you defined yourself in the older version to the current version, to ensure these records are available in the current version.

Click `Update external data` to select the "ext" directory of the older version. This then automatically copies the "dat", "rpt" and "rpu" subdirectories to the current release.

Click `Update settings` to transfer your personal settings from the previous version to the current release.

Select `Connect file extensions` to link all the KISSsoft files with the current version so that you can double-click on any particular file to open it in the current release.

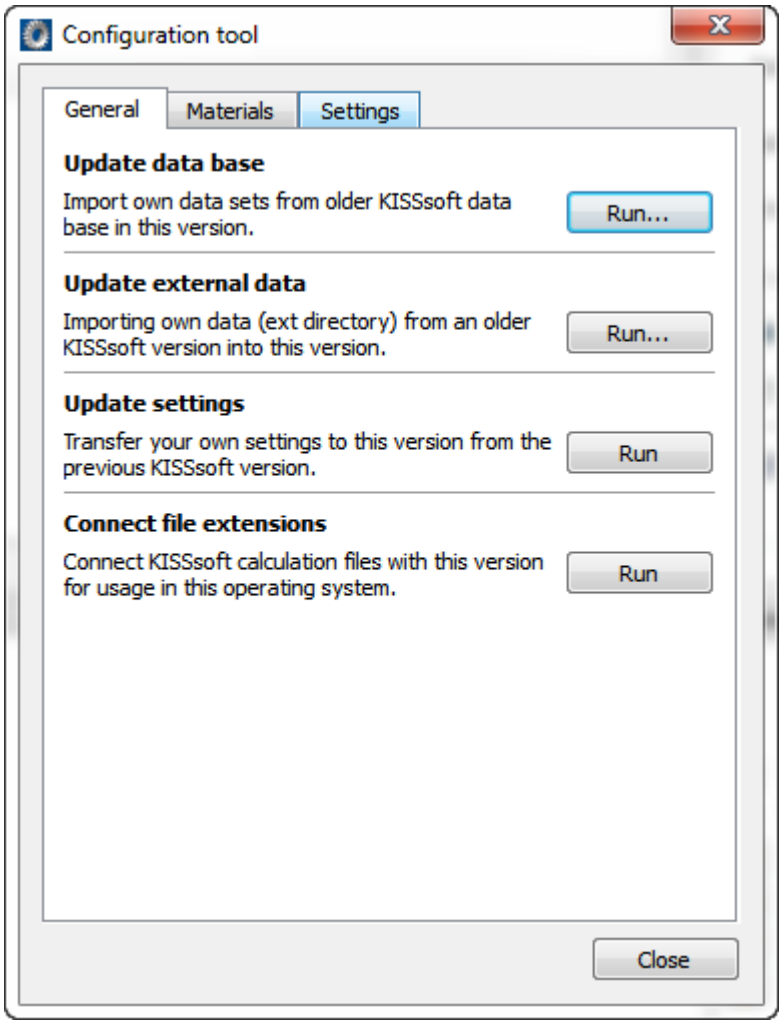


Figure <Kap3>.1: `General` tab in the `Configuration tool` window

In the `Materials` tab you can specify the standard with which the material descriptions in the database are to comply.

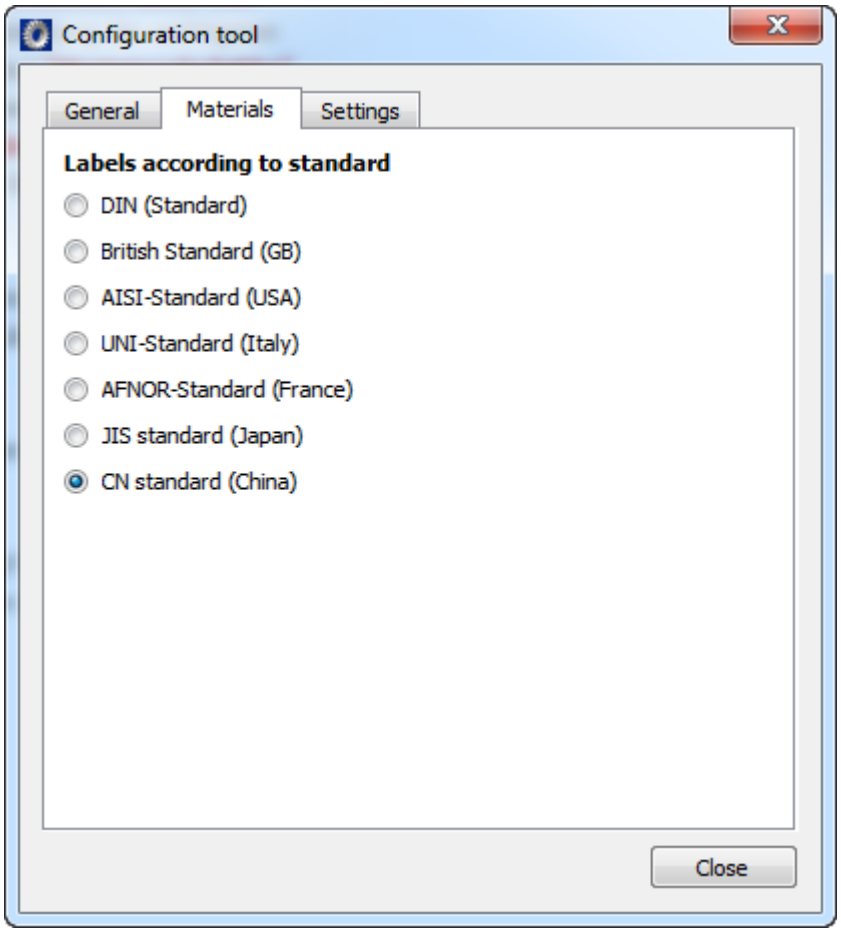


Figure <Kap3>.2: `Materials` tab in the `Configuration tool` window

In the `Settings` tab you can delete the user-defined settings (divided into groups). This reloads the default values.

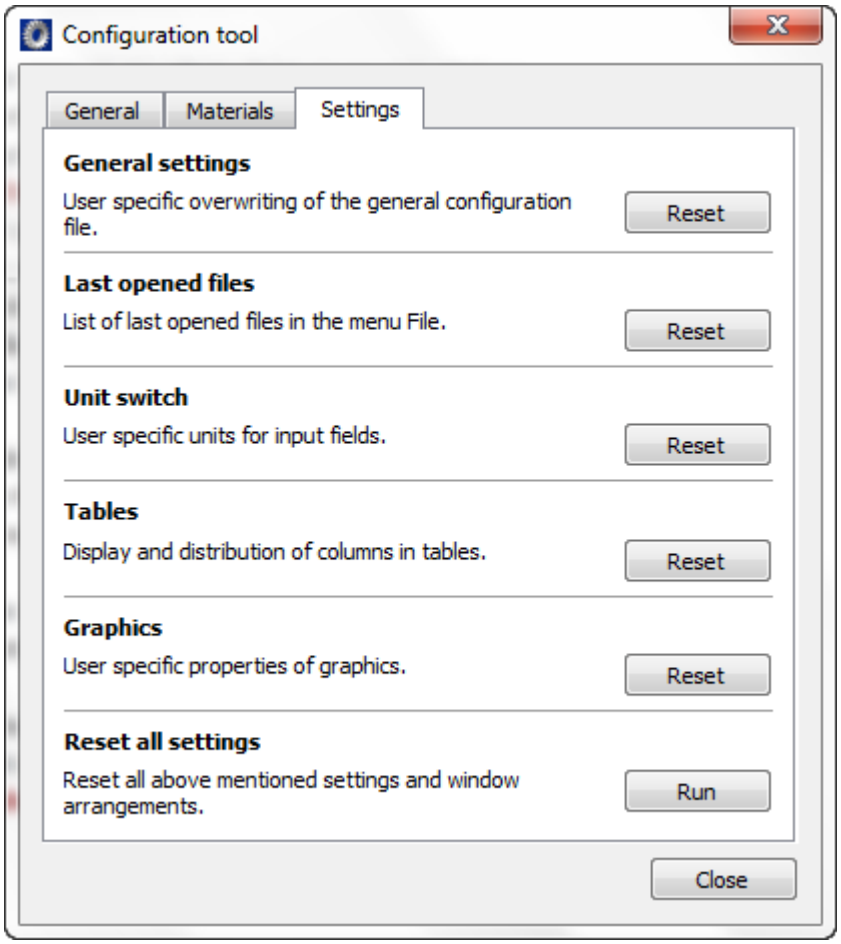


Figure <Kap3>.3: `Settings` tab in the `Configuration tool` window

2.8 Rules

Rules are used to ensure that in-house guidelines for the validity ranges of parameters are applied and adhered to. This typically concerns the maximum and minimum limits of input values, calculated values and the relationships between these values i.e. length-width relationships, length-diameter relationships or even the relationship between the module and the center distance.

These rules are defined by being stored in a <module>.rls file, where <module> stands for the in-house description of the calculation module, e.g. Z012 for cylindrical gear pairs.

These rules are subdivided into those that must be fulfilled before the calculation is performed and those that must be checked afterwards. If a rule is infringed, the appropriate messages can be displayed. In the case of rules that must be checked before the calculation, variables can also be set to constant or calculated values.

The following statements are possible in this situation:

precalc: marks the beginning of the rules that must be checked before a calculation is performed.

postcalc: marks the beginning of the rules that must be checked after a calculation.

assert(<Condition>): The <condition> is ensured. In this case, the <condition> usually represents a comparison, where both the right-hand and left-hand side of the comparison can also be calculated.

action msg <Message>: If the <condition> of the previous assert has not been fulfilled, the <message> is output. Here the <message> can include variables, in the same way as report templates.

action set <Assignment>: If the <condition> of the previous assert has not been fulfilled, the <assignment> is performed. The assigned value can be a constant or be calculated from variables, in the same way as for the report templates.

An assignment is only really useful in the precalc section because changing the contents of variables after the calculation merely leads to inconsistent results and has no other effects.

Here is an example file for a helical gear calculation:

precalc

assert (ZR[0].x.nul < 1)

```
  action msg "Profile shift for Gear 1 too big, is {ZR[0].x.nul}, maximum 1. It has been reset to 1."
  action set ZR[0].x.nul = 1

assert (ZR[1].x.nul < 1)
  action msg "Profile shift for Gear 2 too big, is {ZR[1].x.nul}, maximum 1. It has been reset to 1."
  action set ZR[1].x.nul = 1
postcalc
assert ((ZP[0].a/ZS.Geo.mn) < 200)
  action msg "Center distance too big for module (a={ZP[0].a}, mn={ZS.Geo.mn}, a/mn={ZP[0].a/ZS.Geo.mn})."
```

Explanations:

The "precalc" statement open the section of the rules that must be executed before the calculation.

The first "assert" statement checks whether the nominal profile shift of gear 1 is less than 1.0.

If this "assert" is not fulfilled, the "action msg" statement outputs the message that the profile shift is too big, displays the current value and tells the user that the profile shift has been set to 1.0.

The "action set" then sets the profile shift to 1.0.

The second "assert" statement checks the same values for gear 2.

The "postcalc" statement signifies the end of the set of rules to be executed before the calculation and opens the section of the rules to be checked after the calculation. The example shows a definition of an "assert" statement. This checks the ratio from the center distance to the module. If the rule is infringed, the "action msg" statement triggers a message. However, there is no point in converting one of these two values after the calculation and this is why the "action set" statement is not present here.

The "General – Results and Reports – Report templates – Formatting – Calculation variables" section details which operators and functions can be used in the formulae.

The file containing the rules is stored in the template directory (TEMPLATEDIR, usually the "template" subdirectory, see the "Directory structure" section in the "Setting Up KISSsoft" section). As the template directory can also be project-specific, you can also define project-specific rules.

Chapter 3
Starting KISSsoft

3.1 Initial parameters

KISSsoft can be called up from the input prompt with the following initial parameters:

Parameter	Description
INI=`directory`	The KISS.ini (see page I-51) file will be loaded from the specified location. You can transfer a file name including its directory path, or only a directory name.
START=`module`	The specified calculation module will be started. The module descriptor is, for example, `M040` for bolt calculation or `Z012` for cylindrical gear pair calculation.
LOAD=`file name`	The calculation module belonging to the file is started and the file is loaded. If the supplied file name does not include a path, the system looks for the file in the User directory (see page I-51).
LANGUAGE=`number`	KISSsoft starts with the language specified for the interface and reports. (0: German, 1: English, 2: French, 3: Italian, 4: Spanish, 11: English with US Customary Units)
DEBUG=`filename`	A log file with debug information will be written which can be very helpful for error-tracking. It is advisable to define the file name with a complete path, so that you can find the log file easily later.
`File name`	The calculation module belonging to the file is started and the file is loaded. This also provides a way to associate KISSsoft with the appropriate filename extensions in Windows.

3.2 Disconnect license from the network

If KISSsoft has not been properly shut down, it may be possible that users remain registered, in the case of a network version. This may lead to licenses being blocked even though some users are no longer working with KISSsoft. You can disconnect a license from the network by selecting the required license (the user and start time are also specified) under `Extras > License tool` in the `Network` tab, which deletes the appropriate cookie file and activates the blocked license on the network again.

Unused licenses will be activated after a certain time, as soon as the next user logs on. This time-span can be predefined via the TIMEOUT (see page I-54) variable in the kiss.ini (see page I-51) file.

NOTE

A user who has been disconnected from KISSsoft can no longer carry out calculations in the current session. The user must restart KISSsoft. However, data backups can still be carried out.

Chapter 4
User Interface

KISSsoft is a Windows-compliant software application. Regular Windows users will recognize the elements of the user interface, such as the menus and context menus, docking window, dialogs, Tooltips and Status bar, from other applications. Because the internationally valid Windows Style Guides are applied during development, Windows users will quickly become familiar with how to use KISSsoft.

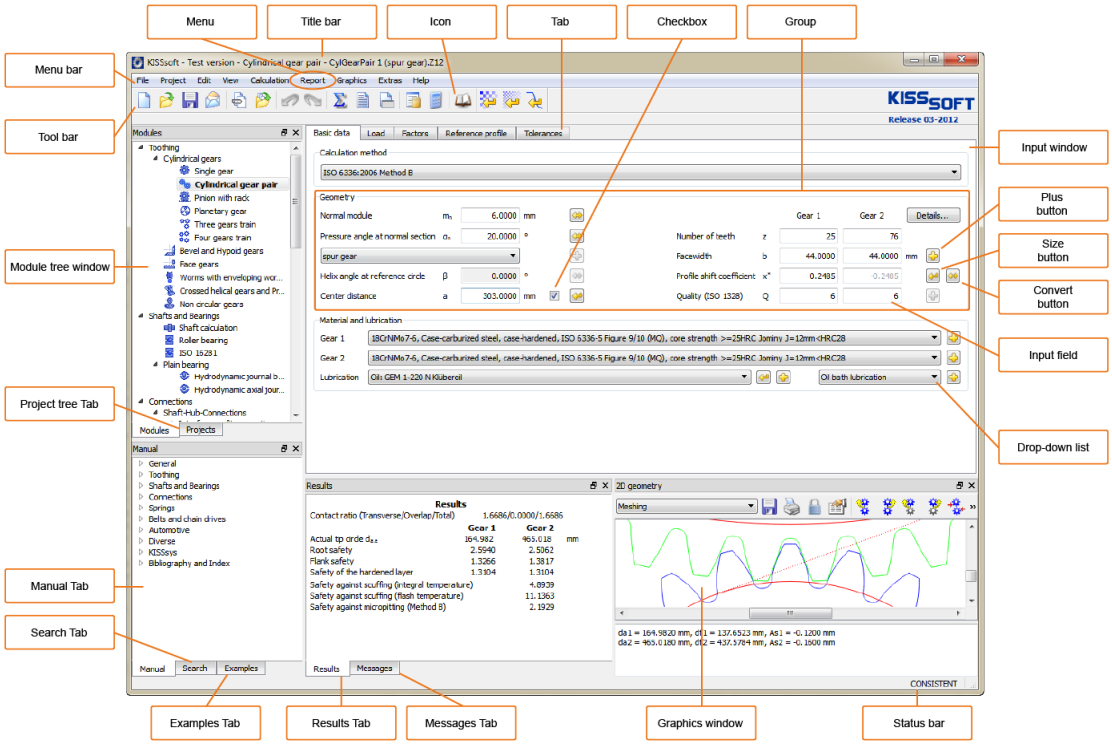


Figure 4.1: KISSsoft's user interface

4.1 Menus, context menus and the Tool bar

In the `File` main menu you can open, store, and send calculation files as email attachments, restore previous calculation stages, view file properties, and close KISSsoft. Click `File > Save as template` to retain user-defined default values (standard files (see page I-49)).

You can use the KISSsoft Project Management (see page I-91) functionality from both the `Project` main menu and the Project (see page I-73) tree. You can open, close and activate projects, insert files into a project, or delete them, and also view project properties.

Each individual docking window (see page I-72) in the user interface can be hidden or displayed in the `View` main menu. If you are in the report or helptext viewer, select `View > Input window` to return to the calculation module input dialog.

In the `Calculation` main menu you can run the current calculation (see page I-85), add more calculations to the calculation module as default or special tabs and call subcalculations as dialogs. Select `Calculation > Settings` to change the module-specific settings.

In the `Report` main menu you will find actions for generating and opening a report. The system always generates a report for the current calculation. Click `Report > Drawing data` to display Drawing data (on page I-100) for the element currently selected in the Report Viewer (see page I-82). Click `Report > Settings` to change the report's font size, page margins and scope. The actions for saving, sending and printing are only active if a report is open.

You can open and close the Graphics (see page I-75) window of a calculation module in the `Graphics` main menu.. Select `3D export` to access KISSsoft's CAD interfaces. Select `Graphics > Settings` to select the CAD system into which you want to export the selected element.

In the `Extras` menu you will find the license tool, the configuration tool and the database tool. In this main menu you can start the Windows calculator and change the Language (see page I-47) and System of units (see page I-48). In `Extras > Settings` you can change general program settings such as the formats for time and date values.

In accordance with Windows conventions, at the end of the menu bar you will find the `Help` icon which you can use to navigate in the KISSsoft manual. In `Help > Info` you will find information on the program version and on the support provided by KISSsoft.

In addition to the main menu, KISSsoft uses **context menus** in many locations. Context menus give you access to actions for a particular area or element of the software. Context menus are normally called up via the right-hand mouse button.

The **Tool bar** gives you faster access to actions from the menus that are used particularly frequently. You should also note the tool tips which display information about the actions in the Tool bar as well as other descriptions in the Status bar (see page I-84).

NOTE

The `Calculation`, `Report` and `Graphics` main menus are only active if a calculation module is open. The actions available in these menus may vary depending on the current calculation module.

4.2 Docking window

Beside the menu bar, Tool bar and Status bar, the docking windows are important elements in the KISSsoft user interface. Docking windows are windows that, can either be moved freely on the desktop, like a dialog, or can be docked onto the pages of the program, in any arrangement that suits you. Several docking windows can be placed on top of each other and be represented as tabs.

You can unlock a docking window by double-clicking in its title bar. You move a docking window by clicking with the left-hand mouse button in the title bar and moving the mouse with the key held down. If you move the mouse close to the edge of the main window, a new position for the docking window will be displayed. Release the mouse button to position the docking window. Docking windows can be displayed and hidden via the `View` `(see page I-70)` menu.

4.2.1 The module tree

The module tree shows all KISSsoft calculation modules in an easy to understand and logically structured list. Any calculation modules for which you have not purchased a license are grayed out. You open a module by double-clicking on it with the left-hand mouse button. The current calculation module will be shown in bold.

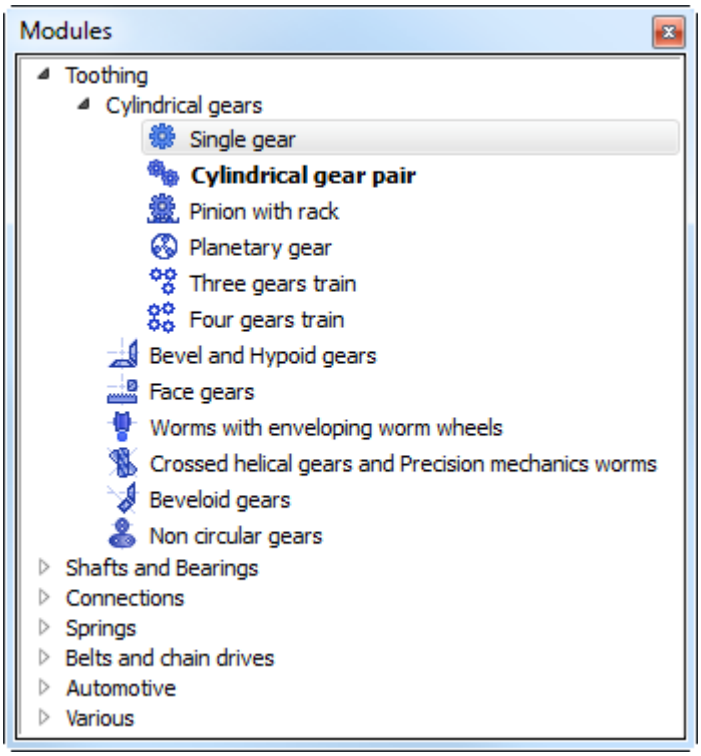


Figure 4.2: KISSsoft calculation modules

4.2.2 The project tree

The project tree gives you an overview of the open projects, and the files belonging to these projects, and highlights the active working project (see page I-94) in bold. You use the project management (see page I-91) functions via the `Project` menu or from a context menu (see page I-70).

4.2.3 The Results window

The KISSsoft results window displays the results of the last calculation.

Results

Results			
Contact ratio (Transverse/Overlap/Total)		1.6686/0.0000/1.6686	
	Gear 1	Gear 2	
Actual tip circle $d_{a.e}$	164.982	465.018	mm
Root safety	2.6022	2.5141	
Flank safety	1.3292	1.3844	
Safety of the hardened layer	1.2828	1.2828	
Safety against scuffing (integral temperature)		4.8971	
Safety against scuffing (flash temperature)		11.1757	
Safety against micropitting (Method B)		2.1823	

Figure 4.3: The KISSsoft results window

4.2.4 The Messages window

The messages window displays all information messages, warnings and errors. Generally, all additional messages are not only displayed, but also appear in a message box. You can change the way information and warnings are displayed in a message box by selecting `Extras > Settings`.

4.2.5 The info window

The Info window displays information that is displayed when the user clicks on an Info (see page I-87) button in the calculation module. You zoom and print the information via a context menu (see page I-70).

4.2.6 Manual and Search

The manual's Table of Contents and search function are also available as docking windows. When you select an entry by double-clicking on it, the Helptext viewer (see page I-83) opens and the relevant section in the manual is displayed.

4.3 Graphics window

In KISSsoft you can open as many graphics windows as you need at the same time and arrange them in the same way as the other docking windows (see page I-72). This means you can see all the graphics and diagrams you require for your calculations at a glance. To make working with graphics more effective you can use the Tool bar (see page I-76), the Comment field, the context menu (see page I-78) and the Properties (see page I-78).

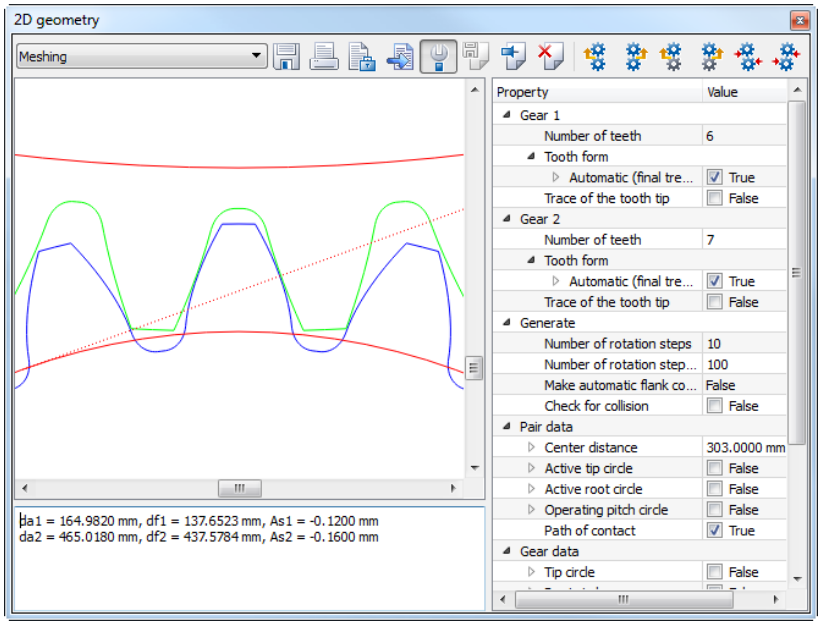


Figure 4.4: Components of the graphics window

4.3.1 Tool bar and context menu

Use the selection list in the Tool bar to switch from one graphic to another in a group. You will also see various icons for saving, printing and locking a graphic, as well as functions for highlighting and graying out its properties.

Save graphics as

This stores the graphics as DXF, IGES or other image or text formats under the name you enter here.

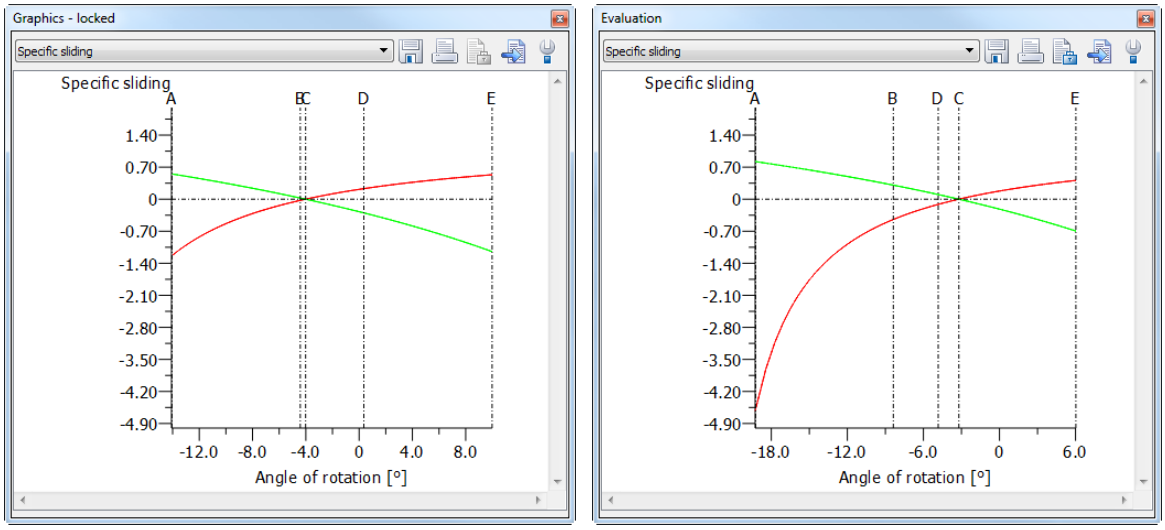
Saving diagrams in a DXF file usually creates a conflict between the diagram axis units and the unit used in the DXF file. For this reason, when you save a diagram, the program opens a dialog in which you can specify the drawing area to which the diagram is to be projected in the file.

Print

Prints the current section of the graphic. The information underneath the graphics is defined in the graph?.rpt report templates (see Report templates (on page I-104)).

Lock

This is useful for comparing two calculation results. In this way, you can, for example, generate a `specific sliding` graphic for a toothing scenario, lock this graphic and then, after having changed the gear parameters, open a new graphics window that shows the new calculation results. The locked window will no longer be updated.



(a) Locked window (b) Window with new calculation results

Figure 4.5: Locking graphics windows

When you lock a graphics window, a dialog will open in which you can enter a title for the window, which will make it easier for you when you are making comparisons.

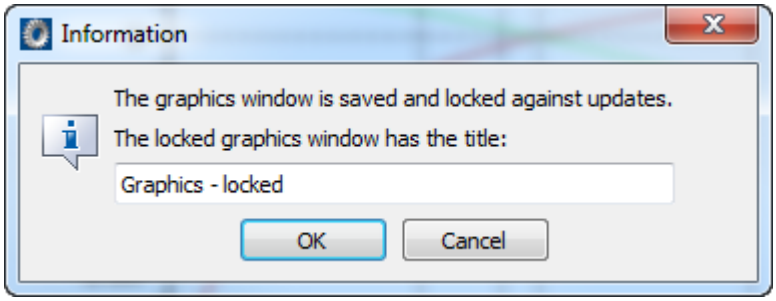


Figure 4.6: Dialog window for inputting the window title

Properties

This opens a list with the Properties (see page I-78) of the current graphic in the same window.

4.3.2 Comment field

In the Comment information is displayed about the graphic. You can change the Comment to suit your needs and it is included in the print output.

4.3.3 Context menu

Here, use the left-hand mouse button to select, move, zoom and measure elements in a graphic. You can permanently select which action is to be performed in the context menu. You can access this more quickly by using these combinations: Move: Shift, Zoom: Ctrl and Measure: Alt key with the left-hand mouse button.

Other actions in the context menu are: Zoom In (plus), Zoom out (minus) and Fit window (Pos1 or Home). Use the direction keys to move the current section of the graphic.

4.3.4 Properties

In Properties you can display or hide elements in a graphic and change its colors and line styles. You can make different modifications, depending on the graphic: for diagrams and such like, you can modify the value ranges and units to match the axes, or for a meshing you can change the center distance.

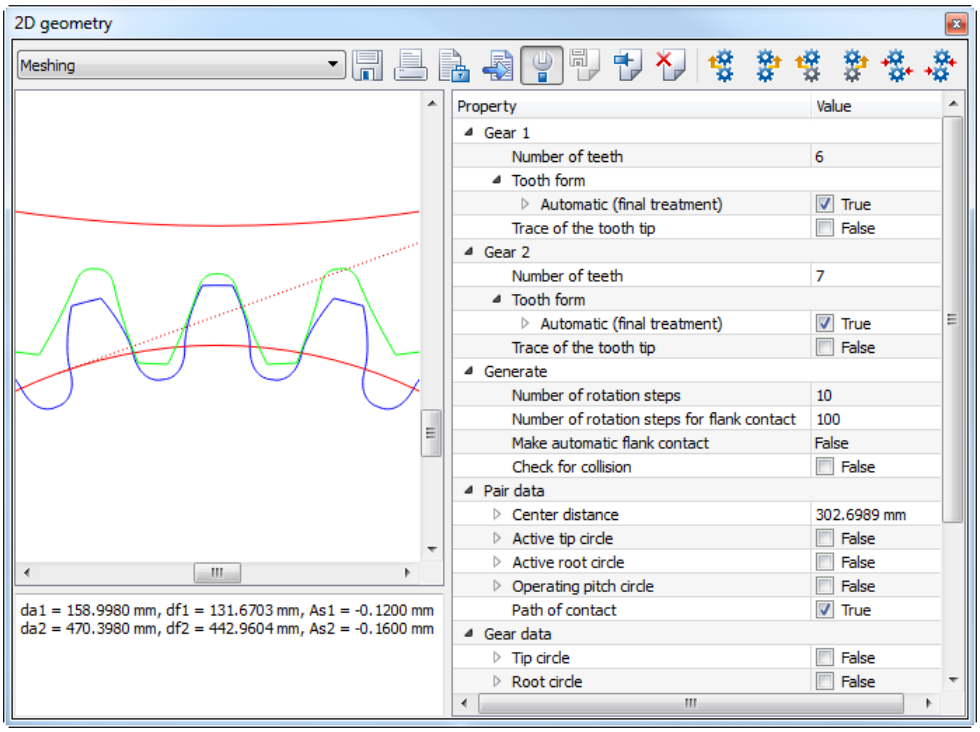


Figure 4.7: Graphic properties

If the properties are displayed, you will see three other icons in the Tool bar. You use them to store curves in a graphic as text, or in the graphic itself.

Save curve as text

Stores the coordinates of the curve selected in Properties in a text file. This makes it easy to transfer curves to, for example, an Excel file.

Save curve

Stores the curve selected in Properties in the graphic. This function is ideal for comparing the graphical outputs of a calculation whilst you change its parameters.

Delete memory

Deletes the curve from the memory.

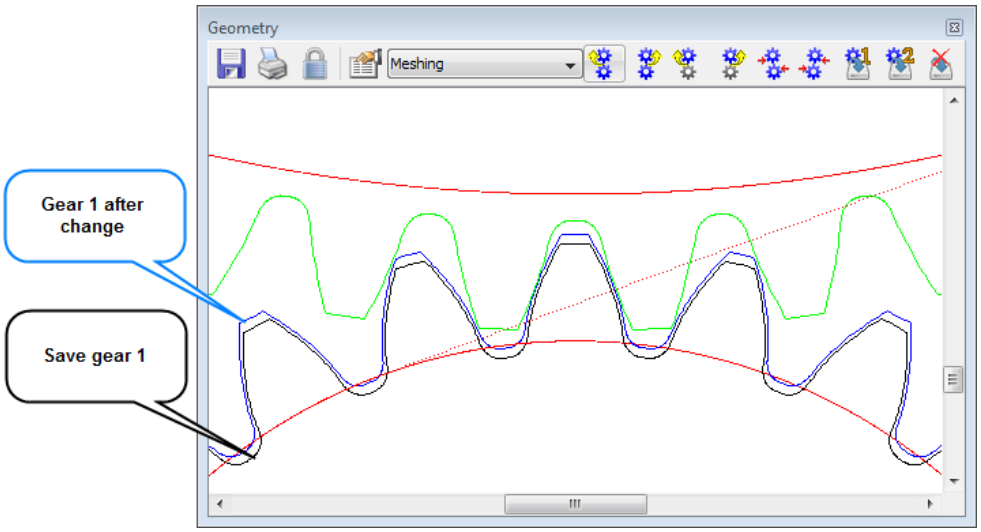


Figure 4.8: Graphics with saved and different curves

4.3.5 Toothing

If you select Toothing, additional icons are displayed for generating the gear pair and creating the flanks when you open the `Geometry` graphics window.

Rotate to the left

Turns the gear pair to the left.

Key combination: Ctrl + left direction key

Rotate to the right

Turns the gear pair to the right.

Key combination: Ctrl + right direction key

Rotate independently to the left

One gear remains static whilst the other is rotated to the left. The profiles overlap.

Key combination: Alt + left direction key

Rotate independently to the right

One gear remains static whilst the other is rotated to the right. The profiles overlap.

Key combination: Alt + right direction key

Make flank contact left

The gears are rotated until the flanks of both gears touch on the left.

Make flank contact right

The gears are rotated until the flanks of both gears touch on the right.

NOTE:
Hold down a rotate button to rotate the gears continuously (movie).

NOTE:
Click Properties (see page I-78) to specify the number of rotation steps for the rotation. The number of rotation steps here refers to the pitch.

4.4 Main input area

The main input area shows a calculation module's input window. In addition, it is used to display the internal report viewer or the internal help viewer.

4.4.1 Report Viewer

When you generate a report in KISSsoft, the report viewer in the main input area will open, the entries in the `Report` menu will be activated and the report viewer Tool bar will be displayed. The report viewer is a text editor that supports the usual functions for saving and printing a text file. In KISSsoft, you can save reports in Rich Text Format (*.RTF), in portable document format (*.PDF), in Microsoft Word format (*.doc) or as ANSII text (*.txt).

The report viewer's other functions are Undo/Redo, Copy, Cut and Paste, and Search with the usual shortcuts. You can zoom in on the view and later edit the report by changing the font size, bold, italics and underlining style. To change the general appearance of the report, select `Report > Settings`.

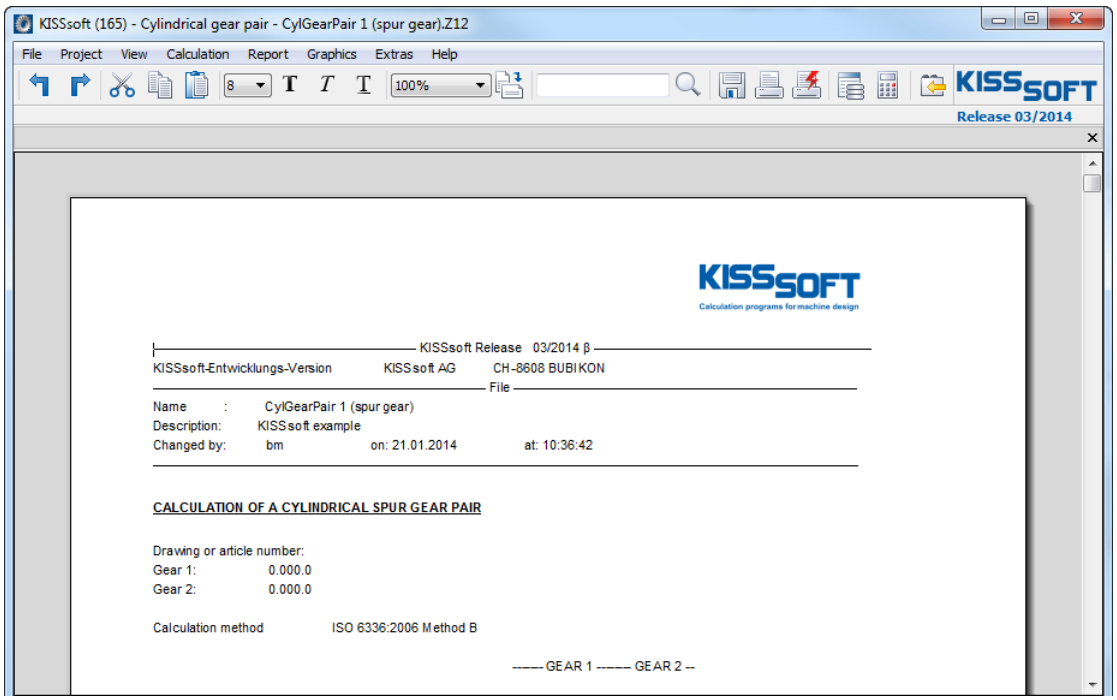


Figure 4.9: The KISSsoft report viewer

4.4.2 Helptext viewer

The KISSsoft manual is displayed in the Helptext viewer in HTML format. To open the manual, select something in the Table of Contents or the Search function. If you press function key `F1`, the system displays more information on the location in KISSsoft at which the cursor is currently is located.

4.5 Tooltips and status bar

Whenever it is useful, Tooltips are provided in KISSsoft to give you additional information about program elements. Tooltips appear automatically if you slowly move the mouse over a program element.

If you position the mouse over a particular menu item, the system will display detailed information on all actions available in that menu, in the left-hand area of the Status bar. If the mouse is positioned over a selection list, the currently selected list entry will be displayed in the Status bar. This is especially helpful if the display is restricted by the width of the selection list.

In the right-hand area of the Status bar the system will display the current status of the calculation. The flag is set to CONSISTENT if the results are current. INCONSISTENT shows that a new calculation needs to be carried out.

Chapter 5
KISSsoft Calculation Modules

5.1 Standard and special tabs

The input window for most calculation modules is subdivided into different tabs. This ensures that inputs are separated logically. For more complex calculations such as for a cylindrical gear pair, the system does not automatically display all existing tabs. When you open a new calculation, you only see the tabs that contain the absolutely necessary inputs (e.g., for a cylindrical gear pair this would be the Basic data, Reference profile and Tolerances tabs). In the `Calculation` menu you can add more tabs if needed (e.g., for a cylindrical gear pair you would need the "Modifications" and "Correction of the gears" tabs).

KISSsoft calculation modules use two types of tabs: Standard tabs and Special tabs, as shown in Figure 1.1.

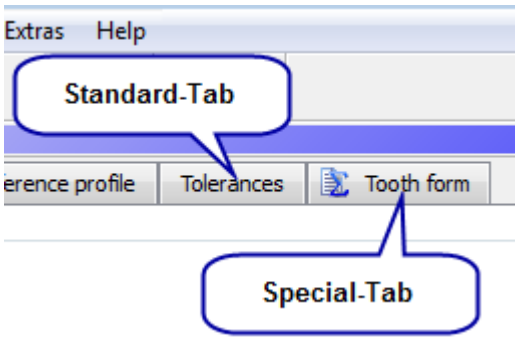


Figure 5.1: Standard and special tabs

If a standard tab (e.g. Basic data) is active when the calculation is run, then the standard calculation will be executed and the results of this standard calculation will be displayed in the Results window (see page I-73). When a report is generated, the default report is created.

Special tabs are marked with the icon. If a special tab is active when the calculation is run, then a special calculation will be executed in addition to the standard calculation, (e.g., for a cylindrical gear pair the calculation of the meshing line under load). The results of this additional calculation will then be displayed in the Results window, and when you generate reports you will get a report containing the results the additional calculation.

5.2 Input elements

All KISSsoft calculation modules use the same input elements for input. These input elements are described in more detail in the sections that follow.

5.2.1 Value input fields

In general, a value input field always includes the label of the variable, a formula character, the edit field and a unit. If the edit field is grayed out, this variable cannot be predefined. Instead it will be determined during calculation. One or more of the following buttons can follow a value input field:

You can retain a value by selecting the Check button.

You can set a radio button to specify which values in a group should be calculated and which should be retained

Click the Sizing button to calculate the value using calculation methods

Click the Convert button to calculate the value using conversion formulae

Click the Plus button to display additional data for a value

Click the Info button to display information in the Info window (see page I-73)

5.2.2 Formula entry and angle input

In some cases it is advisable to use a small auxiliary calculation to determine a value. Click the right-hand mouse button in the Edit field of a value input field (see page I-87) to open a formula editor. In it you can enter a formula, which must be one of the four basic calculation types: +, -, * and /. Additionally, you can use all the functions that are supported by the report generator (→ see Table on page I-110). Confirm the formula by pressing Enter. The system will evaluate the formula. The formula itself will be lost: if you return to the formula entry dialog, the calculated value will be shown there instead of the formula.

In value input fields (see page I-87) that display an angle, a dialog in which you can input degrees, minutes and seconds will be displayed instead of the formula editor.

5.2.3 Unit switch

In KISSsoft, you can switch all the units in the value input fields (see page I-87) and in the tables (see page I-87). To do so, click on a unit with the right-hand mouse button. A context menu will open, offering all possible units for the value. If you select a different unit from the one that is currently in use, KISSsoft converts the current value in the value input field into the new unit.

To switch between metric and US customary units globally, select `Extras > System of units`.

5.2.4 Tables

In some modules data will be displayed or entered in a table. You select a row by double-clicking, just like when you select a field for input. For tables, additional information is often displayed in a tooltip (see page I-84). In general, the following buttons come after tables so that you can input data:

Click the Add button to insert a row into the table

Click the Remove button to delete the selected row from the table

Click the Clear button to delete all entries in the table

5.3 Calculating and generating a report

Click on `Calculation` > Click `Run` to perform the current calculation. In addition, the tool bar and the `F5` function key give you quick, convenient access to this action. Here, please note that a calculation module can have other special calculations in addition to the standard calculation. These special calculations are only executed if the appropriate Special tab (see page I-86) is active.

Select `Report > Generate` to generate a report about the current calculation. Also note the differentiation here between the default report and the reports about the special calculations in the Special tabs (see page I-86).

The status of a calculation is consistent if it could be performed without error. As soon as you change data in the input window, the calculation becomes inconsistent, which means that the results of the calculation in the Results window and the graphics no longer match with the data in the interface. The current status of the calculation is displayed in the Status bar (see page I-84).

5.4 Messages

A calculation sends different types of messages to the input window: information, warnings and errors. Information and warnings should always be taken note of to ensure accurate results. If an error has occurred, the calculation is interrupted.

Normally, all the messages are displayed in a message box and in the Messages window (see page I-73). You can change the way information and warnings are displayed in a message box by selecting `Extras > Settings`.

Chapter 6
Project Management

KISSsoft contains its own project management system to help you organize your calculation files and your external files. The most important area in the project management system is the KISSsoft project tree (see page I-73). In it you can see which projects are currently opened or active, and you can see all the information about the files belonging to the individual projects.

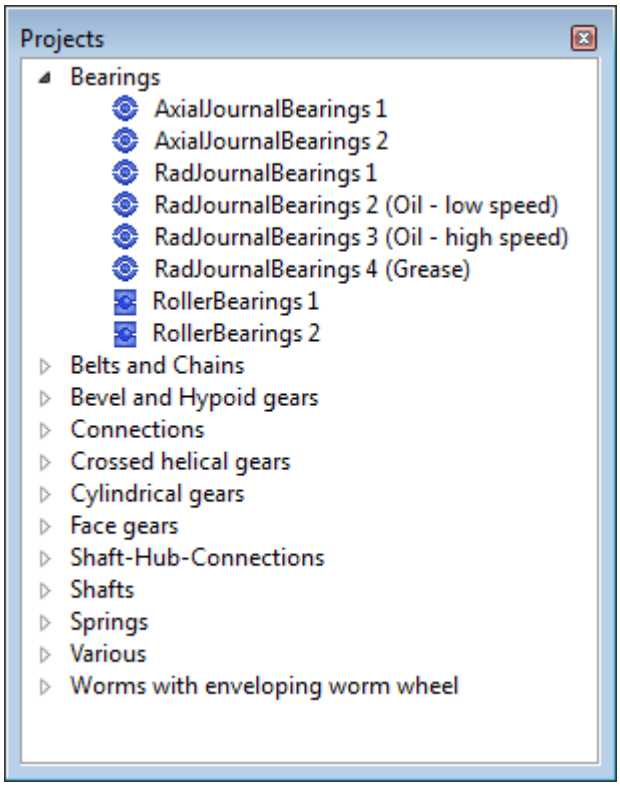


Figure 6.1: The KISSsoft project tree

6.1 Creating, opening and closing projects

Click on `Project > New...` to create a new project. A dialog opens in which you enter the name of the project, the project directory, descriptions and comments, and also the directory for the templates (see page I-49) that are to be used. The newly created project is inserted into the project tree and defined as the Active working project (see page I-94).

When you open an existing project (click on `Project > Open...`) this is also inserted into the project tree and defined as the Active working project (see page I-94).

You close a project by selecting it and then clicking `Project > Close`. You will also find this action in the project tree's Context menu (see page I-70). The project will still be retained, and you can open it again at any time.

6.2 Adding and deleting files

Files can be added and deleted either via the Project properties (see page I-96) or the Context menu (see page I-70). Not only can you insert calculation files from KISSsoft into a project, but also any external files.

6.3 The active working project

The project tree shows all opened projects, and it is not absolutely necessary to define an active working project. If you have defined an active working project, it is highlighted in bold. Click on `Project > Set as working project` to enable a project. Alternatively you can do this in the Context menu. If you select `Project > Work without project`, this deactivates the active working project.

The current calculation file does not necessarily have to belong to the active working project.

6.4 Storage locations

Files belonging to a project do not necessarily have to be stored in that project's directory. Consequently, files can also belong to several projects simultaneously. However, if you have defined an active working project (see page I-94) KISSsoft will prompt you with its project directory as the first choice storage location whenever you want to open or save a calculation file or a report. If you are working without a project, the system will display your personal user directory (see page I-51) as a default storage location, which you can change.

6.5 Project properties

Click on `Project > Properties` to display the project properties of the selected project. Alternatively, you can display this in the project tree's context menu (see page I-70).

Chapter 7
Results and Reports

7.1 Results of a calculation

KISSsoft displays the results of a calculation in the Results (see page I-73) window. If no results are displayed, an error has occurred during the calculation. In this case, you will be alerted to the error by a system message in a message box. An indicator in the status bar (see page I-84) shows whether the results are consistent, i.e. whether the results match up with the data in the user interface.

7.1.1 Add your own texts in the results window

To do this, define a new file in the KISSsoft installation folder in "…\ext\rpt\". This file must then be named like this: "Modulname + result.RPT" (e.g. for a cylindrical gear pair Z012result.RPT).

Then define the new parameters or values that are to be added. These values then appear at the end of the "Results" window.

The syntax corresponds exactly to the entries for the report templates.

7.2 Calculation reports

Select `Report > Generate` to generate reports about your calculations. In addition, the tool bar and the `F6` function key give you quick, convenient access to this action. The report contents depend on which tab (see page I-86) is currently active. The Length (see page I-105) and Appearance (see page I-105) of standard reports can be influenced by user-defined report templates (see page I-104).

A calculation module can contain further reports which you can access via the `Report` menu.

Reports are usually displayed in the KISSsoft Report Viewer (see page I-82). **Important**: The report is not saved when you return from the report viewer to the input window. To make it permanently available, you must save it under a new name!

NOTE

In general, a report should only be created if the calculation is consistent (see page I-89). If this is not the case, you can still generate the report, but the status of the calculation will then be noted in the report.

NOTE

When you generate a report, the system generates an RTF file with the module's label as its file name. The file is saved in the directory defined as the TEMPDIR (see page I-51) in the KISS.ini (see page I-51) file.

7.3 Drawing data

Depending on which calculation module you are using, click `Report> Drawing data` to create a report that can be used to output drawings.

7.4 Report settings

Under `Report > Settings`, you can tailor the automatic generation of reports. All the settings can also be defined globally in the KISS.ini (see page I-53) file.

7.4.1 General

Here you define the scope of the report (see page I-105) and whether warnings from the calculation are to be included in it. Further options are the font size and language, along with the standard format used to save the report.

The report can be viewed in two different modes: "update" or "compare".

If a report is generated and a previous report is still open, the data will be updated. The cursor in the editor will remain in the same line where it was left. This feature will help the user to analyze specific values using different inputs.

Change the report mode to "compare" if you need to compare two or more reports at a time. This mode can only by set if you are using KISSedit as the editor. You can also synchronize the reports and scroll through them all at the same time.

You can also set these properties directly in the KISS.ini file.

7.4.2 Page layout

Here you can define the paper size and the page margins used to create reports automatically.

7.4.3 Header and footer

In KISSsoft, reports are usually generated with headers and footers. You can define your own header and footer lines. There are a number of placeholders available for this.

Placeholder	**Description**
%logo	Picture file
%date	Dated
%time	Time
%pn	Number of pages
%pc	Number of pages
%t	Tab

The %logo placeholder uses the selected graphics file to integrate a user-defined logo (company label). The date and time are output in accordance with the details specified under `Extras > Settings`.

7.4.4 Start and end block

Reports in KISSsoft are usually generated with a start and an end block. You can define these start and end blocks yourself. The start and end blocks are defined in template files which are stored in the `rpt` directory in the installation folder.

Language	Start block file	End block file
German	kissd.rpt	kissfd.rpt
English	kisse.rpt	kissfe.rpt
French	kissf.rpt	kissff.rpt
Italian	kissi.rpt	kissfi.rpt
Spanish	kisss.rpt	kissfs.rpt
Russian	kissr.rpt	kissfr.rpt
Portuguese	kissp.rpt	kissfp.rpt

Commands that can be used in these templates and what they mean:

Command	Explanation
DATE	Date (set your own output format under "Extras/Settings")
TIME	Time (set your own output format under "Extras/Settings")
PROJECT	Project name
PROJECTDESCRIPTION	Description of the project
FILENAME/DESCRIPTION	File name
FILENAME.EXT	File name with extension (e.g. "Example1.Z12")
FILEPATH	Path with file name (e.g. "C:\Temp\GearPair.Z12")
DESCRIPTION	Description of the file
COMMENT	Comment for the file
CUSTOMER	Customer name as defined in the project
USER	User name (Windows user name)

RELEASE	Version number (e.g. "04-2010")
COMPANY	Company name (as defined in the license file)
NLINES	Number of lines in the report
IMPERIALUNITS	Whether US customary units are specified for IF statements
METRICUNITS	Whether metric units are specified for IF statements
PROJECTUSED	Whether projects are used for IF statements

7.5 Report templates

For each calculation module, KISSsoft provides report templates to define the form and content of the reports. You can use these supplied templates as the basis for generating user-defined templates to produce reports that meet your requirements. However, you must ensure the Formatting (see page I-105) and Storage locations (see page I-104) remain the same.

7.5.1 Storage locations and descriptions

The report templates supplied by KISSsoft are stored in the directory that has been set as RPTDIR (see page I-51) in the KISS.ini (see page I-51) file. If RPTDIR (see page I-51) was not defined in KISS.ini (see page I-51), you will find the templates in the installation folder under rpt. It is essential that user-defined report templates are stored in the RPT subdirectory in which the directory defined as EXTDIR (see page I-51) is stored. This is the only way to prevent your templates from being overwritten if a patch is installed. When the system generates a report, it uses the user-defined template from the EXTDIR, if present. Otherwise it uses the template from the RPTDIR to create the report.

Descriptors from the report templates have the structure *MMMMlsz.rpt*, which consists of:

MMMM	Module descriptor	e.g. *M040*
l	historical	always = *l*
s	Language of the report	*s = d, e, f, i, s or a*
z	historical	always = *0*
.rpt	File type	

EXAMPLES

Bolt calculation:

M040LD0.RPT	Bolt calculation, German printout
M040USER.RPT	Default printout via the interface, results in the M040USER.OUT file

Cylindrical gear calculation:

Z012LD0.RPT	Cylindrical gear pair, German printout
Z012USER.RPT	Default printout via the interface, results in the Z012USER.OUT file
Z10GEAR1.RPT	Output via interface, contains only data for gear 1, results in file Z10GEAR1.OUT

Z10GEAR2.RPT	Output via interface, contains only data for gear 2, results in file Z10GEAR2.OUT
Z011LD0.RPT	Single gear, German printout
Z013LD0.RPT	Rack, German printout
Z014LD0.RPT	Planetary gear, German printout
Z015LD0.RPT	3 gears, German printout
Z016LD0.RPT	4 gears, German printout
Spring calculation	
F10SPRING.RPT	Default printout for drawing data results in the F10SPRING.OUT file
English printout:	
M040LE0.RPT	Bolt calculation, English printout
American printout:	
M040LA0.RPT	Bolt calculation, American printout

7.5.2 Scope of a report

The scope, or the length of a report can be preset on a scale of 1 to 9 in the `Report > Settings` menu. 9 will produce a complete report, and 1 will produce a short report. In the report template, you see a number between 1 and 9 at the beginning of every row. This number acts together with the setting described above to determine whether or not the row is to be read.

Example: If you entered 5 (medium) as the report length, all the lines in the report template that start with 1, 2, 3, 4 or 5 are read. Rows with 6, 7, 8 and 9 will be not read.

7.5.3 Formatting

Both the report template and the report created from this are text files that are created with the Microsoft Windows font. You should always edit text in MS Windows, otherwise accented characters such as ä, ö, ü, as well as some special characters, may be represented incorrectly.

The following statements and key words are defined in the report format:

- Texts that are to be output
- Comments that are not to be output
- Descriptions and formatting of calculation variables

- Limited branchings (IF ELSE END)
- Repetitions (*FOR* loops)

7.5.3.1 Text formatting features

In general, reports in KISSsoft are created in RTF format. RTF can handle the following text formats:

Description	Start	End
Underline	<UL>	</UL>
Cross out	<STRIKE>	</STRIKE>
Bold	<BF>	</BF>
Italic	<IT>	</IT>
Superscript	<SUPER>	</SUPER>
Subscript	_	
Font size	<FONTSIZE=xx>	
Increase font size	<INCFONTSIZE>	</INCFONTSIZE>
Reduce font size	<DECFONTSIZE>	</DECFONTSIZE>
Page break	<NEWPAGE>	
Line break	 	
Text color red	<RED>	<BLACK>
Text color green	<GREEN>	<BLACK>
Text color blue	<BLUE>	<BLACK>
Blank space	<SPACE>	
Insert figure	<IMAGE=name,WIDTH=xx,HEIGHT=yy>	
Insert image	<INCLUDEGRAPHIC=name,WIDTH=xx,HEIGHT=yy>	
Adding a report template	<EXECUTE=name.rpt>	

7.5.3.2 Comments

Comment lines begin with //. Comments are ignored when a report is created

EXAMPLE

```
// I changed the report template here on 12.13.95, hm
External diameter mm : %10.2f {sheave[0].da}
```

In this case, only the second line will be output.

7.5.3.3 Calculation variables

You cannot define your own variables (apart from the number variables for FOR loops (see section "FOR loop" on page I-111) which the user specifies and which can output a value).

Placeholder

Use placeholders to specify the file type and formatting of a variable:

- *%i* stands for a whole number
- *%f* stands for a floating comma number
- *%* v_1*.* v_2*f* represents a formatted floating comma number with v_1 places in total (including prefix operator and decimal character) and v_2 decimal places
- *%s* represents a left-justified character string (text)
- *%ns* represents a right-justified character string in an *n* character-long field (n is a whole number).

The data types must correspond with the definition in the program. The value is returned in exactly the place where the placeholder is positioned. The syntax of the formatting corresponds to the C/C++ standard.

EXAMPLES

- *%10.2f* outputs a floating comma number to 10 places with 2 decimal places, justified to the right.
- *%i* returns an unformatted whole number exactly in this location.
- *%30s* represents a right-justified character string in a 30 character long field (if the number 30 is omitted, the characters will be output, but left-justified).

COUNTER-EXAMPLES

- *%8.2i* is an invalid formatting because a whole number has no decimal places.
- *%10f2* outputs a floating comma number to 10 places, right-justified. However the 2 decimal places are ignored and output as text 2. The default setting is to output floating comma numbers to 6 decimal places.

Variables

The variable to be shown must be specified after the placeholder in the same row. The variable is marked as such with curly brackets. If these brackets are omitted, the variable name will be shown as normal text.

Important: It is essential that the number of placeholders exactly matches the number of pairs of brackets {}.

EXAMPLE

%f {sheave[0].d} returns the value of the variable *sheave[0].d* in the location *%f* as a floating comma number with 6 decimal places.

Basic calculation types - output of changed variables

In the report you can show changed variables. They can be multiplied or divided with a coefficient. You can also add or subtract a number. This functionality is also available in the arguments of the *IF* or *FOR*statements (see below).

Value of the variable multiplied	*%3.2f*	*{Var*2.0}*
Value of the variable divided	*%3.2f*	*{Var/2.0}*
Value of the variable added	*%3.2f*	*{Var+1.0}*
Value of the variable subtracted	*%3.2f*	*{Var-2}*

The two degree and gear functions can also be used to perform conversions to degrees or to radians:

Angle %3.2f {Grad(angle)}

Variables can also be directly linked with each other, e.g. in the form *{sheave[0].d-sheave[1].d}*. You can also link more than two numbers. Numbers containing prefix operators must stand in brackets, for example *{ZR[0].NL*(1e-6)}*.

The available functions are listed in Table 7.2.

Function	**Meaning**
sin(angle)	sine of angle in the radian measure
cos(angle)	cosine of angle in the radian measure
tan(angle)	tangent of angle in the radian measure
asin(val)	arcsine of val, returns radian measure
acos(val)	arccosine of val, returns radian measure
atan(val)	arctangent of val, returns radian measure
abs(val)	|val|
exp(val)	e^{val}

log(val)	Return value x in e^x = val
log10(val)	Return value x in 10^x = val
sqr(val)	Return value val^2
sqrt(val)	Return value $\sqrt{val}$
int(val)	Whole number of val
pow(x;y)	Return value x^y
sgn(val)	Return value $\begin{cases} 1 & wenn\ val > 0 \\ 0 & wenn\ val = 0 \\ -1 & wenn\ val < 0 \end{cases}$
sgn2(val)	Return value $\begin{cases} 1 & wenn\ val \geq 0 \\ 0 & wenn\ val < 0 \end{cases}$
grad(angle)	Converting from the radian measure to degrees
rad(angle)	Converting from degrees to radian measure
mm_in(val)	Return value val/25.4
celsius_f(val)	Return value $\frac{9}{5}$val + 32
min(v_1; ...; v_5)	The return value is the minimum of $v_1,...,v_5$
max(v_1; ...; v_5)	The return value is the maximum of $v_1,...,v_5$
and(v_1; v_2)	binary and function
or(v_1; v_2)	binary or function
xor(v_1; v_2)	binary exclusive or function
AND(v_1; ...; v_5)	logical and function
OR(v_1; ..., v_5)	logical or function
NOT(val)	Return value $\begin{cases} 0 & wenn\ val \neq 0 \\ 1 & wenn\ val = 0 \end{cases}$
LESS(v_1; v_2)	Return value $\begin{cases} 1 & wenn\ v_1 < v_2 \\ 0 & wenn\ v_1 \geq v_2 \end{cases}$
EQUAL(v_1; v_2)	Return value $\begin{cases} 1 & wenn\ v_1 = v_2 \\ 0 & wenn\ v_1 \neq v_2 \end{cases}$
GREATER(v_1; v_2)	Return value $\begin{cases} 1 & wenn\ v_1 > v_2 \\ 0 & wenn\ v_1 \leq v_2 \end{cases}$
strlen(str)	Length of character string
strcmp(str1;str2)	Compare character string Return value:

	-1 str1 <str2 0 if str1 = str2 1 if str > str2

Table 7.2: Functions available for calculations in the report.

7.5.3.4 Condition query IF ELSE END

The condition query or branching enables you to only output certain values and texts if a particular condition has been fulfilled. The following conditions are supported:

Combination of characters	Meaning
==	equal
>	greater than or equal
<	less than or equal
! =	unequal
<	smaller
>	larger

This condition is entered as follows:

IF (condition) {Var}
  Case 1
ELSE
  Case 2
END;

EXAMPLE

IF (%i==0) {Zst.kXmnFlag}
  Addendum modified no
ELSE
  Addendum modified yes
END;

If the *Zst.kXmnFlag* variable is equal to 0, then output the first text, otherwise output the second text. There can be any number of rows between *IF*, *ELSE* and *END*. For each branching opened with *IF* you must use *END*; to close it again (do not forget the semicolon after *END*). The key word *ELSE* is optional. It reverses the condition. Branchings can be nested within each other up to a depth of 9.

EXAMPLE OF A SIMPLE BRANCHING

IF (%i==1) {ZP[0].Fuss.ZFFmeth}
  Calculation of tooth form coefficients according to method: B
END;

If the variable *ZP[0].Fuss.ZFFmeth* is equal to 1, then output the first text, otherwise it is not output.

EXAMPLE OF ENCAPSULATED BRANCHINGS

```
IF (%f<=2.7) {z092k.vp}
        Lubrication regularly by hand                          (Text1 )
ELSE
        IF (%f<12) {z092k.vp}
                Lubrication with drop dispenser (2 to 6 drops per   (Text 2)
                minute)
        ELSE
                IF (%f<34) {z092k.vp}
                        Lubrication with oil bath lubrication       (Text 3)
                ELSE
                        Lubrication with circulation system         (Text4)
                        lubrication
                END;
        END;
END;
```

If variable *z092k.vp* is smaller than or equal to 2.7, text 1 is output. Otherwise you are prompted to confirm that *z092k.vp* is smaller than 12. If yes, text 2 is output. Otherwise you are prompted to confirm that *z092k.vp* is smaller than 34. If yes, text 3 is output, otherwise text 4 is output.

7.5.3.5 FOR loop

In KISSsoft you can also use *FOR* loops in the report generator. Within a FOR loop a numerical variable will be incremented (or decremented). You can use constructs that are nested down to 10 levels.

This loop is specified as follows:

FOR varname=%i TO %i BY %i DO {startwert}{endwert} {schritt}
// access variable with #varname or $varname

...
END FOR;

- Instead of *%i* or *%f* you can also use fixed figures (static *FOR* loop):

  FOR varname=0 TO 10 BY 1 DO
  ...
  END FOR;

- or a mixture:

  FOR varname=5 TO %i BY -1 DO {endwert}
  ...
  END FOR;

- Each *FOR* loop must end with the statement *END FOR;* (including semicolon). Each defined numerical variable (*varname*) within the loop can be addressed with the statement *#varname*.
- The increment can also be selected as a negative value (for example -1). However, it must never be 0. You must always specify the intervals.
- The *#varname* statement can be used for defining a variable. For example:
- Number of teeth: *%3.2f {ZR[#varname].z}*
- The *$varname* statement can be used for outputting the variable value as a letter. The value 0 corresponds to A, 1 corresponds to B etc. For example:

  FOR quer=0 TO 3 BY 1 DO
  Cross section *$quer-$quer : %8.2f {Qu[#quer].sStatisch}*
  E*ND FOR;*

EXAMPLE OF A SIMPLE LOOP

FOR i=0 TO 10 BY 1 DO
Run number #i $i
END FOR;

Results in the following output:

Run number 0 A
Run number 1 B
Run number 2 C
Run number 3 D
Run number 4 E
Run number 5 F
Run number 6 G

Run number 7 H
Run number 8 I
Run number 9 j
Run number 10 K

The numerical variable can be used anywhere within the loop, even for arrays.

Chapter 8
Database Tool and External Tables

As calculation inputs you may, in addition to the unique data, also encounter recurring data, for example the characteristics of a material. KISSsoft will store these characteristics in databases. You view and change them with the `database tool`, whose use will be explained in the following sections. Tables form the elements of the databases and are contained in your program package as editable ASCII files. The External tables (on page I-122) section deals with the setting up and handling of external tables (also called "look-up tables).

In KISSsoft there are four databases:

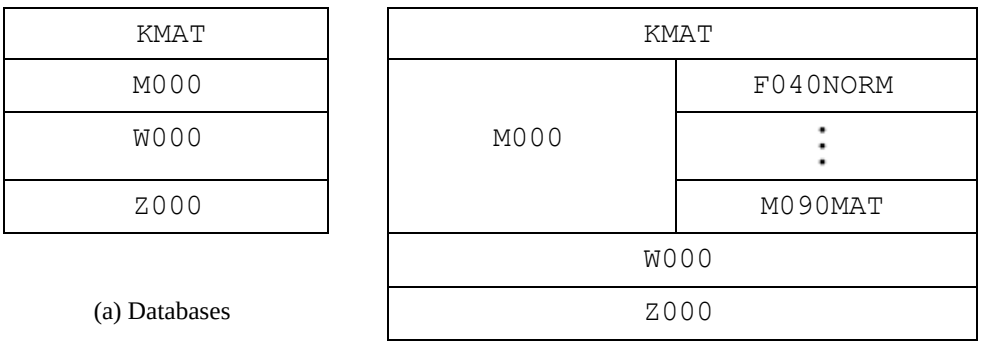
`KMAT` - Materials

`M000` - Shaft-hub Connection and Bolts

`W000` - Shafts and Bearings

`Z000` - Gears

In Figure 8.3 you can see an example of the `M000` database which shows how data is organized in KISSsoft. As shown there, the `F040NORM` and `M090MAT` tables belong to the group of shaft-hub connections.



(a) Databases

(b) tables

Figure 8.4: How data is organized in KISSsoft

Up to now, the following tables have been created in the databases: Center distance tolerances, Reference profiles, Bore standard, Thread type bolt, Production process for hypoid bevel gears, Production process for bevel gears, V-belt standard, Spline standard, Chain type DIN 8154, Chain type DIN 8187, Chain type DIN 8188, Glue

materials, Load spectra, Soldering materials, Key standard, Polygon standard, Woodruff key standard, Lubricants, Bolt Type, Washer standard, Multi-spline standard, Roller bearing, Materials for glued and soldered joint, Material, Tooth thickness tolerances, Toothed belt standard.

8.1 Viewing database entries

Open the database in the `Extras Database tool` menu item, as shown in (Figure 8.5, ①). A dialog window appears with the question whether you want to open the database with write authorization (②). If you press `Yes`, you can edit the database entries, otherwise they are write protected. If you choose `No`, the actual database tool window (③) will start in read-only mode. There, you can select a table from a list that is assigned to a particular database. The row of a table contains the values that set the parameters for the database entry. The columns contain the parameters for the database entries, i.e. values for the yield point of different materials. This section (see section "Managing database entries" on page I-119) describes how to edit database entries. You can also display table entries by selecting a row in the database tool window and then confirming this by clicking `Display` (④). The `Display entry` window opens with a structured display of the value amount from a table row (⑤).

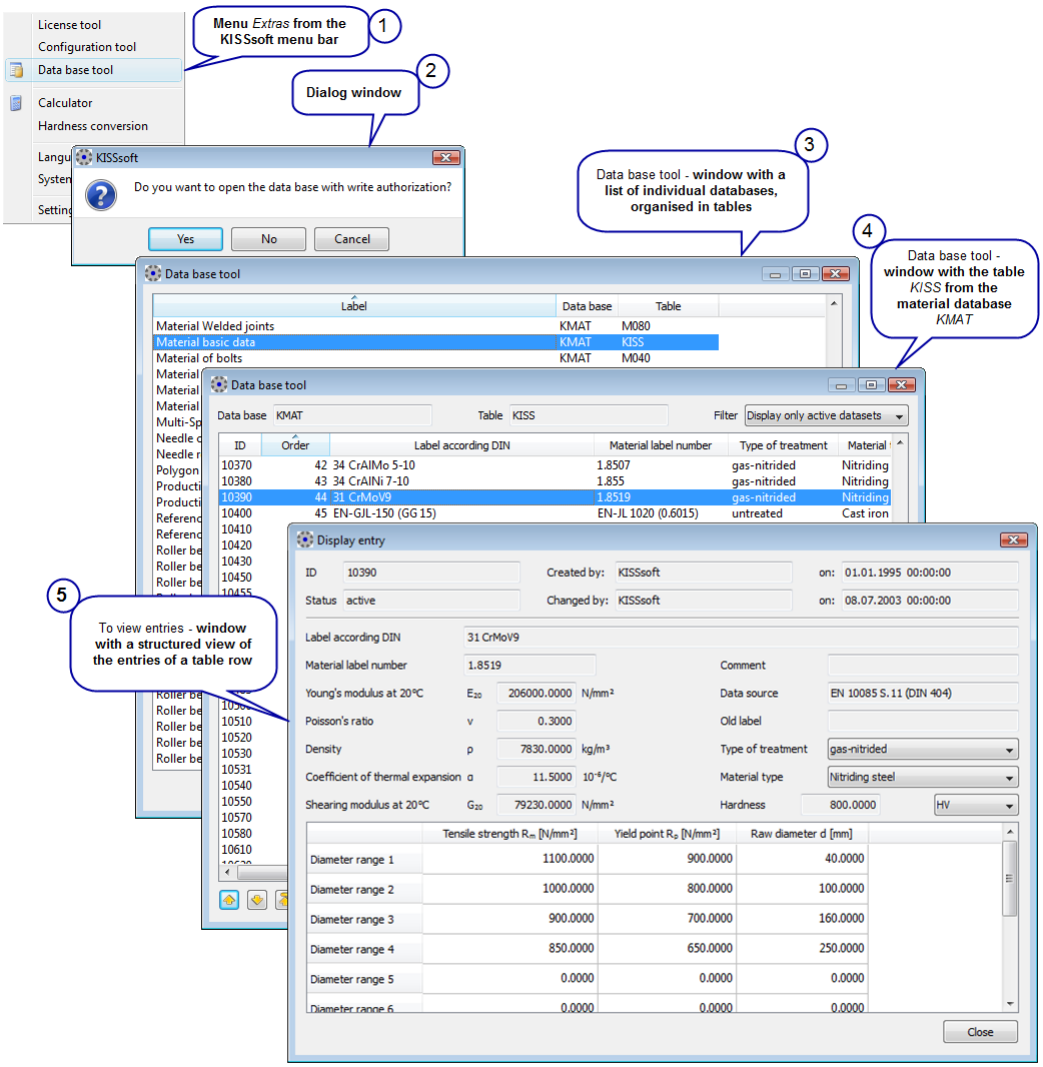


Figure 8.5: Accessing database entries

NOTE:

With the KISSsoft database tool you can change the databases and expand them with your own entries. The data stored in the databases are in a sense "sensitive", so that incorrectly entered values can have consequences that are initially imperceptible noticeable, yet eventually far-reaching and serious. For this reason, when you open the database you are asked whether the access should have write authorization. If you answer this question in the negative, you can view the data in the tables but not change it.

If you want to make absolutely sure that the databases remain unchanged, you can write protect their corresponding files (`*.kdb`). Any attempt to open a table with write authorization results in an error message and the table will normally be opened in write protected mode. To change the write protection attribute of a file, right-click on the file in Windows® Explorer, and then click on `Properties`. Click in the `Properties` dialog field, on the `General` tab, and then click the `Write Protected` selection box. If you want to make changes to a write protected file, you must first deselect the selection box `Write Protected` or else save the file with a different name.

8.2 Managing database entries

If you want to change one of your own entries in a table in the database, you must work in write authorization mode. To do this, go to the (→ sh. Figure 8.5 on page I-116) dialog window and click on `Yes`. In the list that you see next, (③) select the required table by double-clicking on the appropriate row or single-click on the `Edit` button at the bottom right of the window after you have selected the row. The database tool window now shows a list of the table entries (④) and a row of new buttons appears on the bottom left in the window:

Moves the selected item up one row

Moves the selected item down one row

Moves the selected item(s) to the start of the list

Moves the selected item(s) to the end of the list

Adds a new item to the list

Moves the selected item into the list of hidden data records

With the `Filter` drop-down menu on the top right of the window you can select between displaying active data records, hidden data records, or both. Active data records can be used within the calculation modules, hidden ones cannot.

8.2.1 Generating a database entry

If you click on the button without having selected a row, the `Display entry` window (⑤) opens and the input fields in it are empty. Only the `Name` field contains the entry `_NEW`, which normally identifies the new table entry. After you have transferred the necessary data, confirm your entries by clicking on `OK` and then `Save`in the database tool window. The new entry is assigned an identification number (`ID`) ≥ 20000 and is then transferred into the list of active data records. Use the `Edit` button to change entries with an ID of ≥ 20000.

If you click on the button after having selected a row, the `Display entry`window opens and contains predefined values in the input fields according to the table entry. The suffix `_NEW` will automatically be attached to the name, in order to differentiate it from the original data record. In all remaining steps, you then proceed as described above.

Example: Generating a database entry

Let's assume you want to add a new spring material to the `KMAT.F000` table. In accordance with the described procedure you would select the `F000` table from the `KMAT` database, in it press the button to add a new entry/new row to the table, and then transfer the new data into the input fields in the `Display entry`

window. However, as only few parameters can be freely selected there, the next question is, where can the other values such as the yield point and Young's modulus be changed? The reply is: In the base material input fields, i.e. in table `KMAT.KISS`. You must always specify a base material before you can introduce a new spring material. If this is not present, you have to define it in the `KMAT.KISS` table first of all and finally complete the missing entries in `KMAT.F000`.

NOTE

All material-specific tables such as `KMAT.F000` or `KMAT.Z080` - with the exception of `KMAT.KLUB` - have a checkbox beside the `Base material` drop-down menu. If you have marked the checkbox, you have the option to select an alternative base material in the associated drop-down list-menu. If the checkbox is empty, access to the menu of the base materials is blocked. This option helps to protect against unwanted changes during the assignment of the base material.

8.2.2 Deleting a database entry

Data records in KISSsoft will never be deleted. It is only possible to move entries with an ID ≥ 20000 into the table of hidden data records. Select the appropriate entry with a single mouse click in the ④ window and then click the button. The selected row will be copied into the range that contains the hidden data records and removed from the list of active data records. To access the table of inactive data records, select the `Display only hidden data records` option in the `Filter`drop-down list menu in the top right of the database tool window.

8.2.3 Restoring a database entry

In the table of hidden data records you select the appropriate row with a single mouse click and then click on the button. The entry will be copied into the table of active data records and then removed from the range of inactive data records.

8.3 Import and export data with the database tool

The data records from every table in the database tool can be exported to a file or imported from a file. These data records can be imported or exported as a single data record or imported from a list of data records.

To import a list of data records, you will need to save it in a file, preferably an Excel spreadsheet with the extension "*.csv". The inputs in the spreadsheet columns should correspond to the database table columns.

Lists saved with the "*.txt" extension can also be interpreted by the software. The list inputs should be separated by a "comma" or a "semicolon". The settings in your operating system will show which separator should be used.

- Import a data record from a file with the extension "*.kds".
- Export the selected data record to a file with the extension "*.kds".
- Import a list of data records from a file with the extension "*.csv".

Important notes:

1. Only "user data records" (ID >= 20000) >can be exported or imported.
2. An existing "user data record" can be overwritten if you are processing individual data records.
3. The names of the columns in the "*.kds" files is case sensitive and must exactly match the names in the database tool. You could export a data record to verify the column names.
4. A new ID is assigned automatically to every data record when an entire list is imported or exported.

8.4 External tables

KISSsoft uses external tables, also called look-up tables, to handle larger data volumes. It is the task of external tables to assign one or more output values to one or more input values (see Figure 8.6).

Figure 8.6: Principle of functionality of external tables

The output data that is assigned to the input data are contained in the table.

The external tables are stored in the */<KISSsoft installation folder>/*`dat` directory. If a new table name is entered into a database, a file with the same name and the file extension `.dat` must also be created manually.

Because tables are located externally, KISSsoft can only determine how many of them there are during program execution. The user directly benefits from the fact that they can generate their own files with data tables in a similar way to the files supplied by KISSsoft. The tables are readable ASCII files and consequently can be edited and expanded by the user. It would for example be possible to use an internal standard as an alternative to the ISO base tolerances.

Figure 8.7 shows the three table types used by KISSsoft in one diagram:

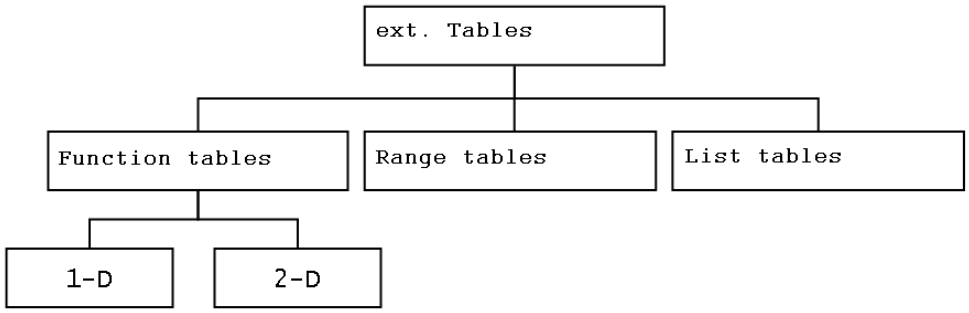


Figure 8.7: Types of external tables

A table always has the following structure, no matter what type it is:

```
:TABLE  <Type> <variable or ID>
  <Table header >
DATA
```

```
  <Data>
END
```

The `:TABLE` command marks the external table as an external table. You must use one of the following designations for the `<Type>` argument:

`FUNCTION`	Functions tables
`RANGE`	Range tables
`LIST`	List tables

NOTE

Blanks in tables can be marked with `*`, `-` or blank spaces. Note here that no space characters may be used if they are followed by more values. KISSsoft interprets blank space as value separators.

The structure of the table header and the body data, which is dependent on the type, is described with example applications in the following sections.

8.4.1 Functions tables

Functions tables are tables that expect one or two input values (1D or 2D table) and which return exactly one corresponding value.

EXAMPLE 1D TABLE

The angle coefficient (`factor`) is determined on the basis of a specified (`angle`). For example: if the input value `angle = 45` supplies an output value of `factor = 0.35`.

```
-- table type: Functions table; output variable: factor
:TABLE FUNCTION factor
-- INPUT X angle defines the input parameter angle;
-- interim values will be interpolated linearly
    INPUT X angle TREAT LINEAR
-- data: 1st line: input values), 2nd line: output values

DATA
    0       30      60      90      . . .
    0.1     0.25    .45     .078    . . .
END
```

INPUT is a key word, i.e. a word that is reserved by the Tables Interpreter, and is followed by an argument X, which assigns a dimension to the angle input parameter. The key word TREAT, with associated LINEAR argument, specifies that interim values are to be interpolated linearly. The output value factor will determined using the value of the angle variable. The first row of data in the 1D table (between DATA and END) corresponds to the input value angle, and the second row corresponds to the output value. The data in a 1D table is therefore always a (2 × n) matrix, i.e. both rows must contain the same number of values.

EXAMPLE OF A 2D TABLE

The nominal power is defined on the basis of the speed and the sheave diameter. For example: if the input values diameter = 60 and speed = 60 supply an output value power = 8.6.

```
-- table type: Functions table; output variable: power
:TABLE FUNCTION power
-- INPUT X diameter defines the input parameter diameter;
-- INPUT Y speed defines the input parameter speed;
-- interim values will be interpolated linearly in both dimensions
    INPUT X angle TREAT LINEAR
    INPUT Y Speed TREAT LINEAR
-- data: (→ sh. Figure (see section "Example: Interference fit calculation" on page I-162))

DATA
        50      100     200     300     ...
50      4       7       12      25      ...
75      12      25      30      35      ...
...     ...     ...     ...     ...     ...
END
```

Here, the variable power is defined with the input variables INPUT X and INPUT Y. Interim values running down the columns (Y) should be interpolated linearly. The same applies across the rows (X). The first row of the table corresponds to the values of the input variables INPUT X and the first column corresponds to the values of the input variables INPUT Y. The values at the

intersections of the input values are the values that correspond to the output variables (see Figure <chapterNo.>.8).

	X_1	X_2	X_3
Y_1			
Y_2	$f(X_1, Y_2)$		
Y_3			

Figure 8.8: Data schema of 2D tables

Please note that in this way it would be possible to define an inverse table. Assuming that, in your `XY` belt catalog, the table that shows the power output contains the speed in the first row, and the diameter in the first column, then there is no need for you to turn your table upside down. Instead, simply change the assignment in the table header (i.e. replace `X` with `Y`).

8.4.2 Range tables

Range tables check whether a given value is moving within a defined range.

EXAMPLE

```
-- table type: range table; Name of the table: 'A'
:TABLE RANGE 'A'r
-- INPUT X drehzahl defines the drehzahl (speed) input parameter;
-- interim values will be interpolated logarithmically.
-- INPUT Y power defines the input parameter power
    INPUT X drehzahl TREAT LOG
    INPUT Y power
-- data: 1st row: INPUT X, 2nd row: INPUT Y upper limit
 -- 3rd row: INPUT Y lower limit

DATA
                200     300     500     1000    4000
    LOWER       1.5     2.0     3.0     10      20
```

```
    UPPER      10          15          20          15          40
END
```

The two input variables are `drehzahl` (speed) and `leistung` (power). The output value represents the decision on whether the power in dependency with the speed is moving within a defined range and does not have to be declared. Interim values of the speed will be interpolated logarithmically. The first row of the body data corresponds to values of the `drehzahl` (speed) variable. The other rows correspond to values of the `leistung` (power) variable with `LOWER` as the lower, and `UPPER` as the upper, limit. The input value of `leistung` (power) will be compared with these limits and a report sent to the program stating whether the `leistung` is located below, within, or above, the given range `A`.

8.4.3 List tables

In list tables containing at least one input value, several output values will be defined. If more than one input value is entered, the sequence of the input values is important. The reading direction goes from left to right and the first input value defines the range of the next input value, which in turn defines that of the next one, etc. up to the last. All input values apart from the last one must correspond with the entries in the body data (`TREAT DIRECT`, list of used key words (see page I-128).

EXAMPLE 1

The following three input values are assumed:
`g.d = 2.0; g.P = 0.8; s.l = 6`
In accordance with the code shown below, the output values would then be:
`s.l = 7; s.k = 3; s.k = 4.5.`

```
-- table type: List table; output variable s.norm
:TABLE LIST s.norm
-- INPUT g.d defines the input parameter g.d;
-- INPUT g.P defines the input parameter g.P;
        INPUT g.d
        INPUT g.P
-- IN_OUT s.l defines s.l as phase variable
-- TREAT NEXT_BIGGER specified how interim values are handled
        in_OUT s.l                    TREAT NEXT_BIGGER
-- OUTPUT s.k, s.dk declares s.k and s.dk as output variables
```

```
        OUTPUT s.k,s.dk
 -- data: A (N × Nin) matrix

DATA
      2.0     0.4     0
      2.0     0.8     5       3       4.5
      2.0     0.8     7       3       4.5 - relevant data row
      2.0     0.8     10      3       4.8
END
```

In contrast to functions tables, `s.norm`in the first row of the code specifies the name of the external table and not the output variable. `IN_OUT s.l` declares a variable `s.l`, which is used both as an input and output variable (phase variable). `TREAT` functions again as a key word for processing the interim values: `NEXT_BIGGER` shows that input values are to be evaluated it they are not present in the corresponding column of the body data. In the example, the input value `s.l = 6` lies between the values 5 and 7 and, in accordance with `NEXT_BIGGER`, will be promoted to the next bigger value. `OUTPUT s.k, s.dk` declares not only `s.l.` but also the output variables `s.k` and `s.dk`. The number of the columns of the body data must at least correspond to the number of input variables and at most correspond to the number of input variables + output variables, in this case: $3 < N_{in} > 5$.

EXAMPLE 2

To determine the diverse measurements of a bolt, two input values will be used: the bolt type, here represented by the `typ` variable and the length of the bolt, given by `l`.

```
:TABLE LIST schraube.geometrie
   INPUT type
   INPUT L                    TREAT NEXT_SMALLER
   OUTPUT M, dw, (s), e, bez, vorrat
```

```
DATA
    . . .
    12x2.5    20    12    14.57    23.78    5.75    ID 1    1
    12x2.5    25    12    15.78    24.88    5.75    ID 2    1
    . . .
END
```

This table is called `bolts.geometry`. The sequence in the table header defines that within the columns, the first column therefore corresponds to the `typ` variable, the second to the `l` variable, etc. The `typ` and `l` variables will be used as inputs, where the value for the `typ` variable must be present in the list. If an interim value is given for the `l` variable, the row with the next smaller value will be interpreted as the result. Blanks are not permitted, i.e. in this type table values must always be present. It may happen that individual variables are shown in brackets in the output definition. This has the effect that the corresponding column is ignored, i.e. this variable will not be specified.

NOTE

Commented-out output definitions cannot be changed by the user.

8.4.4 List of key words used

`--`	Everything in a row coming after this comment character will be ignored by the Interpreter.
`DATA`	Below this is located the data matrix.
`END`	Ends the input area of the external table.
`INPUT [<dim>] <var>`	Input variable, with definition of the dimension if required.
`IN_OUT <var 1>[, <var 2>, ...]`	List tables: Phase variables
`LOWER`	Range tables: Lower limiting value.
`OUTPUT <var 1>[, <var 2>, ...]`	Output value(s)
`:TABLE <Type>`	Defines the type of the external table.
`TREAT DIRECT`	Interim values: none permitted. Input values in the corresponding column/row must match up with those of the body data.
`TREAT NEXT_SMALLER`	Interim values: The next smallest value is

	assigned.
`TREAT NEXT_BIGGER`	Interim values: The next highest value is assigned.
`TREAT LINEAR`	Interim values: Linear interpolation.
`TREAT LOG`	Interim values: Logarithmic interpolation.
`UPPER`	Range tables: Upper limiting value.

8.5 Description of database tables

The individual database tables have very different structures. The next section describes these database tables and their specific fields.

The **Label** field appears in every table and is only described once, here. You must enter a unique name for the data record in this field. This name is then used to select the data records in the program.

Note: Fields in which file names are to be entered have an auto-fill function. To perform this, the software searches in the ..\dat and ..\ext\dat folders, and also in the current project directory.

8.5.1 Center distance tolerances

- **File name:** the database entries refer to external tables (see External tables (on page I-122)). The tables used for center distance tolerances begin with K10-???.dat. The center distance tolerances specified in ISO 286 are imported directly from the program code and not from a file.

8.5.2 Machining allowance for cylindrical gear

- **File name:** the database entries refer to external tables (see External tables (on page I-122)). The tables for the cylindrical gear machining allowance begin with ZADDT-???.dat.

8.5.3 Reference profiles

You input reference profile data directly in the database. However, each individual value depends on the other.

- **Description in accordance with ISO: the standard on which this is based**
- **Comment: Text field for your own use**
- **Data source: Text field for your own use**
- **Definable reference profile data: Dedendum coefficient h^*_{fP}, root radius coefficient ρ^*_{fP}, addendum coefficient h^*_{aP}, tip radius factor ρ^*_{aP}, topping, protuberance height coefficient h^*_{prP}, protuberance angle α_{prP}, tip form height coefficient h^*_{FaP}, ramp angle α_{KP}**

8.5.4 Compression springs standard

You can store data from geometry standards for compression springs.

- **File name:** the database entries refer to external tables (see External tables (on page I-122)). The tables for compression spring standards begin with f010-??.dat.
- **Tolerance:** tolerance data for the geometry standard

8.5.5 Selection of hobbing cutters

- **File name:** the database entries refer to external tables (see External tables (on page I-122)). The table for cutter data in accordance with DIN 3972 is called Z000-BP.dat.

8.5.6 Basic material Glued and Soldered joints

- **Tensile strength R_m:** [N/mm^2] Data about the tensile strength of the material is required to calculate glued and soldered joints.

8.5.7 Manufacturing process for bevel and hypoid gears

These values are only necessary for calculations using the Klingelnberg method. They correspond to tables for machine types that use the Klingelnberg in-house standard.

- Values that must be defined: **Machine type, Cutter radius cutter tip r_0** [mm]**, No. of cutter blade groups z_0, Maximum machining distance MD_{max}** [mm]**, Minimum normal module $m_{n,min}$** [mm]**, Maximum normal module $m_{n,max}$** [mm]

8.5.8 V-belt Standard

- **File name:** the database entries refer to external tables (see External tables (on page I-122)). Tables for the V-belt standard begin with Z090-???.dat.
- **Calculation method:**
  - 1) Narrow V-belts (Fenner)
  - 2) Narrow V-belts/ Force belts
  - 3) Conti belts

- More definitions: **Maximum belt speed v_{max}** [m/s]**, Elasticity E** [N]**, Weight per length q** [kg/m]**, Coefficient of friction** μ_r

8.5.9 Spline Standard

- **File name:** the database entries refer to external tables (see External tables (on page I-122)). Tables for spline standard norms begin with M02C-???.dat.
- **Calculation method:** the appropriate calculation method is selected for each spline.

8.5.10 Chain profiles ISO606

- Values to be defined for this table: **Type, Pitch p** [mm]**, Number of strands n_s, Maximum diameter of roller d_1** [mm]**, Maximum bearing pin body diameter d_2** [mm]**, Minimum width between inner plates b_1** [mm]**, Maximum width over inner link b_2** [mm]**, Total width b_{tot}** [mm]**, Maximum inner plates depth h_2** [mm]**, Ratio t_H/t_S**

8.5.11 Adhesives

- **Comment:** Text field for your own use
- Definable values: **Minimum** and **maximum shear strength $\tau_{B,min}$,$\tau_{B,max}$ [N/mm^2]**.

8.5.12 Modifications

The different modifications applied to gears are defined as database classes. If a data record is not displayed in the database it will no longer appear in the modifications selection list.

Although you can add new data records to the database, these will not be visible in the calculation module.

8.5.13 Load spectra

- All inputs (frequency, power, speed) must be defined in coefficients. The power and speed are given as factors of the nominal power. In the calculations, the coefficient for torque (load factor/speed factor) is used for forces and torques. You can either import load spectra from a file or enter them directly. If

you input this data directly, the number of load cases is defined by the number of rows you enter.

- **Input:** Specify whether the factors are for power or torque. This also applies if the load spectrum is imported from a file.
- **Read load spectrum from file:** If this flag is set, you can select a file with the load spectrum. If the flag is not set, you can input the load spectrum directly.
- **Own input of load spectra:** You can input the load spectrum directly or load it from a file.
- **File name:** Click the button to select a file from the directories. The file containing the load spectrum must be a text file (*.dat). You will find a sample load spectrum file (called "Example_DutyCycle.dat") in the "dat" directory. You should store load spectra you define yourself in the "EXT/dat" directory to ensure they are always available even after a version upgrade.

```
0.1  1    1
0.1  0.9  1
0.1  0.8  1
0.1  0.7  1
0.1  0.6  1
0.1  0.5  1
0.1  0.4  1
0.1  0.3  1
0.1  0.2  1
0.1  0.1  1
```

Example of a file used to input a load spectrum

- **Frequency:** H_0 ... H^{19}, the total of these frequencies must be 1.
- **Load factor (torque factor):** P_0 ... P^{19}, $0 < P_n < \infty$.
- **Speed factor:** N_0 ... N^{19}, $0 < N_n < \infty$.

Load spectrum elements with negative power or speed are interpreted as a change from driving to driven. This does not affect the root and flank strengths, however it does change the safety against scuffing and against micropitting.

8.5.14 Solders

- Definable values: **Minimum** and **Maximum shear strength** $\tau_{B,min}$ **,** $\tau_{B,max}$ [N/mm^2].

8.5.15 Surface roughness of shafts and shaft-hub connections

You can also input your own data for shaft-hub connections in the "Own Input" field. The Rz values for the shafts are taken from the list.

- **Comment:** Text field for your own use
- Definable values: **Average roughness height R_z** [µm] and **Roughness average value R_a** [µm].

8.5.16 Key standard

- **File name:** the database entries refer to external tables (see External tables (on page I-122)). Tables for key standards begin with M02A-???.dat.

8.5.17 Polygon standard

- **File name:** the database entries refer to external tables (see External tables (on page I-122)). Tables for polygon standards begin with M02D-???.dat.

8.5.18 Woodruff Key standard

- **File name:** the database entries refer to external tables (see External tables (on page I-122)). Tables for polygon standards begin with M02E-???.dat.

8.5.19 Bolts/pins

- **File name:** the database entries refer to external tables (see External tables (on page I-122)). Tables for bolts/pin standards begin with M03A-???.dat.

8.5.20 Lubricants

- **Comment, description as specified in ISO, data source:** Text fields for your own use
- **Additive for rolling bearings:**
  - **without additives:** lubricants without additives, or with those additives whose effectiveness in roller bearings has not been tested.
  - **with additives:** lubricants, whose effectiveness has been tested in roller bearings

- **Oil/ Grease:** specify whether the lubricant is an oil or a grease.
- **Kinematic viscosity at 40°C and at 100°C** ν_{40},ν_{100} **:** [mm^2/s]
- **Lubricant base:** Selection options:
  - Mineral oil
  - Synthetic oil based on polyglycol
  - Synthetic oil based on polyether
  - Synthetic oil based on polyalphaolefin
  - Synthetic oil based on esters

  Polyalphaolefin: similar to mineral oil, easily mixable with mineral oil, some approved for use with foodstuffs.

  Esters: some approved for use with foodstuffs, some biodegradable.

- **Test procedure scuffing:** Selection options:
  - No information about scuffing
  - FZG Test A/8.3/90; ISO 14635-1 (normal)
  - FZG Test A/16.6/90
  - FZG Test A/16.6/120
  - FZG Test A/16.6/140
  - FZG Test A10/16.6R/120; ISO 14635-2
  - Input of scuffing temperature
- **Load stage for scuffing FZG test:** Input scoring load level as specified in the FZG test. These values are required for gear calculations.

  1= weakest level; 12=best level

  Good gear lubricants all have scuffing load level 12.

- **Scuffing temperature** θ_s**:** You can also input the scuffing temperature for the scuffing test procedure.
- **Micropitting process:** Selection options
  - No information available for micropitting load stage
  - C-GF/8.3/90 (FZG)
- **Load stage micropitting test:** The best achievable load stage is 10.
- **Density** ρ**:** [kg/dm^3]

- **Cone penetration at 25°C (grease) P_e:** [0.1mm] This value is only required to calculate grease lubricated sliding bearings.
- **Soap proportion (Grease) c_s:** [Vol%] This value is only required to calculate grease lubricated sliding bearings.
- **k-coefficient, s-coefficient (compression viscosity) k, s:** Coefficient used to calculate compression viscosity (AGMA 925):

$$\alpha = k \cdot \eta_M^S$$

$$\eta_P = \eta_{atm} \cdot e^{\alpha_p}$$

If you do not know these values, you can input 0 and then the values are taken from the standard (AGMA 925-A03, Table 2).

- **Lower/ Upper limit working temperature θ_{min}, θ_{max} :** [°C]

8.5.21 Bolts: Tightening factor

- **Calculation method:** Select VDI 2230(1990) and VDI 2230(2003)
- **Minimum tightening factor α_{Amin}:** Minimum value, can be defined per tightening technique.
- **Medium tightening factor α_{Amid}:** Medium value, can be defined per tightening technique.
- **Maximum tightening factor α_{Amax}:** Maximum value, can be defined per tightening technique.

8.5.22 Bolts: Bore

- **File name:** the database entries refer to external tables (see External tables (on page I-122)). Tables for bores begin with M04-???.dat.
- **Unit in use**: select whether the values in the file are to be given in mm or inches.

8.5.23 Bolts: Strength class

- **Comment:** Text field for your own use
- Definable values: **Yield point R_p** [N/mm^2] and **Tensile strength R_m** [N/mm^2]

NOTE:

For strength grades 8.8 and SAE J429 Grade 2 and 5, the yield point and tensile strength for the lower diameter limit are always displayed in the database. If the diameter is greater than the diameter limit, this is corrected in the program.

8.5.24 Bolts: Nuts strength class

- **Comment:** Text field for your own use
- Definable values: **Vickers hardness for standard thread for diameter D <= 16 mm** [HV], **Vickers hardness for standard thread for diameter D > 16 mm** [HV], **Vickers hardness for fine thread for diameter D <= 16 mm** [HV], **Vickers hardness for fine thread for diameter D > 16 mm** [HV]

NOTE:

The minimum value specified in the DIN EN ISO 898-2:2012 standard is assumed in the strength grades. However, as this value depends on the diameter, in some strength grades, both values are defined in the database. In the program, you can then specify which diameter the value is to be used for. The same applies to differentiating between fine or standard threads. The program handles the value that is being used differently, according to the selection in the interface.

8.5.25 Bolts: Coefficients of friction classes

- **Comment:** Text field for your own use
- Definable values: **Minimum coefficient of friction** μ_{min} [-] and **maximum coefficient of friction** μ_{max} [-]

NOTE:

The minimum and maximum coefficients of friction are always displayed in the database for the coefficients of friction classes. The values are used to size the coefficients of friction for bolts.

The predefined values are taken from Table A5 in VDI 2230, Sheet 1 [75].

8.5.26 Bolts: Thread type

- **Name:** Text field for your own use
- **File name:** the database entries refer to external tables (see External tables (on page I-122)). Tables for threads begin with M04-???.dat.
- **Coefficient used to calculate the flank diameter/core diameter**

- **Flank angle** α**:** [°]

8.5.27 Bolts: Nuts

- **File name:** the database entries refer to external tables (see External tables (on page I-122)). Tables for nuts begin with M04-???.dat.
- **Unit in use**: select whether the values in the file are to be given in mm or inches.

8.5.28 Bolts: Type

- **File name:** the database entries refer to external tables (see External tables (on page I-122)). Tables for bolt types begin with M04-???.dat.
- **Name:** Text field for your own use
- **Thread type screw:** selection list to show which thread type this bolt belongs to.
- **Unit in use**: select whether the values in the file are to be given in mm or inches.

8.5.29 Bolts: Washer

- **File name:** the database entries refer to external tables (see External tables (on page I-122)). Tables for washers begin with M04-???.dat.
- **Unit in use**: select whether the values in the file are to be given in mm or inches.

8.5.30 Selection of pinion type cutters

- **File name:** the database entries refer to external tables (see External tables (on page I-122)). Tables for pinion type cutters begin with Z000-Cutter-?.dat.

8.5.31 Disc spring standard

- **File name:** the database entries refer to external tables (see External tables (on page I-122)). Tables for disc springs begin with F040-?.dat.

8.5.32 Tolerances standard

- **File name:** the database entries refer to external tables (see External tables (on page I-122)). Tables for tolerances begin with K10-???.dat.

The tolerances in accordance with DIN EN ISO 286 have been programmed directly into KISSsoft. For tolerance classes H, h, JS and js, the tolerance has been extended up to the nominal length 10000 mm (according to the standard up to 3150 mm). The values were determined by extrapolation.

8.5.33 Beam profiles

- **Drawing file:** image displayed on screen when a shaft is calculated.
- Values for profiles: **Height h** [mm], **Width b** [mm], **Cross section A** [cm^2], **Moments of inertia of plane area on x/z axis I_x/ I_z** [cm^4], **Moment of inertia of torsion I_t** [cm^4], **Section modulus relating to x /z axis W_x/ W_z** [cm^3], **Moment of resistance in torsion W_t** [cm^3]

8.5.34 Multi-Spline standard

- **File name:** the database entries refer to external tables (see External tables (on page I-122)). Tables for multi-spline profiles begin with M02b-???.dat.

8.5.35 Materials

The materials consist of a database table **Basic data Materials** and the particular table for the modules. The Basic data table lists the general material data. As the materials can then be transferred to the individual module tables, you therefore only need to define the basis data once. Module-specific data is then defined in the module tables.

In module-specific tables, you must always select one **Base material**.

8.5.35.1 Basic data: Materials

- **Label in accordance with DIN, BS, Old label, Material number, Origin of data, Comment:** Text fields for your own use
- **Young's modulus at 20°C E_{20}:** [N/mm^2]
- **Poisson's ratio ν:** [-]
- **Density ρ:** [kg/dm^3]

- **Coefficient of thermal expansion** α**:** [10^{-6}/K]
- **Shearing modulus at 20°C** G_{20}**:** [N/mm^2]
- **Type of treatment:** In this list you can select the Type of treatment.
- **Material type:** In this list you can select the material type.
- **Hardness value:** This value is purely for information purposes and has a negligible effect on the calculation
- **Unit of the hardness:** can be selected in the list.
- **Tensile strength** R_m**:** [N/mm^2] a max. of 10 different diameter ranges can be defined
- **Yield point** R_p**:** [N/mm^2] a max. of 10 different diameter ranges can be defined
- **Raw diameter d:** [mm] a max. of 10 different diameter ranges can be defined

8.5.35.2 Material Spring calculation

This table applies to Compression (F010), Tension (F020) and Leg springs (F030)

- **Permissible shear stress:** the database entries refer to external tables (see External tables (on page I-122)). Tables for springs begin with F01-???.dat. In this file you can view or define the permissible shear stress, the values for the Goodman diagram and the values for the relaxation diagram. If the curves of the relaxation diagram are only defined with 2 points, you must set the values for tau3 and rel3 to 0 so KISSsoft can recognize them.
- **Comment:** Text field for your own use
- **Minimum** and **Maximum wire diameter** d_{min}**,** d_{max} [mm]
- **Shearing modulus depending on temperature** α_G**:** [1/K]
- **Use:** selection list with the cold and thermo-formed variants

8.5.35.3 Material of plain bearings

- **Comment:** Text field for your own use

8.5.35.4 Material of enveloping worm wheels

The table applies to worm wheels (Z080)

- **Comment:** Text field for your own use

- **Material characteristics:** Selection list (such as CuSn bronze/ CuAl bronze/ GGG40/ GG25/ PA-12)
- **Mineral oil coefficient W_{MLOel}:** Material/lubrication coefficient for mineral oil
- **Coefficient for polyglycol (DIN)/ (ISO) W_{MLGDIN}/ W_{MLGISO}:** Material/lubrication coefficient for polyglycol
- **Coefficient for polyalphaolefin W_{MLA}:** Material/lubrication coefficient for polyalphaolefin
- **Material coefficient Y_W:** (see DIN 3996, Table 5)
- **Pitting strength σ_{HlimT}:** [N/mm^2] (We recommend you use reduced values as specified in ISO 14521)
- **Shear fatigue strength τ_{FlimT}:** [N/mm^2]
- **Reduced shear fatigue strength $\tau_{FlimTred}$:** [N/mm^2] (If no slight deformation is permitted, you must include reduced strength values in the calculation.)

8.5.35.5 Material Interference fit

- **Comment:** Text field for your own use

8.5.35.6 Material of bolts

The table applies to the bolt module (M040)

- **Comment:** Text field for your own use
- **Permissible pressure p_G:** [N/mm^2] (Data should be entered as specified in VDI 2230)
- **Shear strength τ_B:** [N/mm^2]

8.5.35.7 Material Welded joints

- **Comment:** Text field for your own use

8.5.35.8 Material Disc spring calculation

The table applies to disc springs (F040):

- **Comment:** Text field for your own use

- **Goodman diagram:** the database entries refer to external tables (see External tables (on page I-122)). Tables for the Goodman diagram begin with F04-???.dat.
- **Young's modulus depending on temperature α_E:** [1/K]

8.5.35.9 Material of shaft-hub connection

- **Comment:** Text field for your own use

8.5.35.10 Material Shaft calculation

The table applies to shafts (w010):

- Values for calculating strength according to Hänchen:
  - **Fatigue limit σ_{bW}:** [N/mm^2]
- Values for strength calculation according to DIN 743:
  - **Reference diameter d_b** [mm], **Tensile strength R_m** [N/mm^2], **Yield point R_p** [N/mm^2], **Fatigue limit for bending σ_{bW}** [N/mm^2], **Tension/Compression fatigue limit σ_{zdW}** [N/mm^2], **Torsion fatigue limit τ_{tW}** [N/mm^2]
  - **Experimental data:** the database entries refer to external tables (see External tables (on page I-122)). Tables for the experimental Haigh diagram begin with W01-???.dat.
  - **CrNiMo case hardening steel:** When determining the size coefficient for case hardening steel, you need to specify whether the steel is CrNiMo case hardening steel.
- Values for strength calculation according to FKM:
  - **Tensile strength for reference diameter $R_{m,N}$** [N/mm^2], **Yield point for reference diameter $R_{e,N}$** [N/mm^2], **Effective reference diameter for deff,N,p $d_{eff,N,p}$** [mm], **Effective reference diameter for Rm,N $d_{eff,N,m}$** [mm], **Constant used to calculate Kd (flow) $a_{d,p}$, Constant used to calculate Kd (fracture) $a_{d,m}$, Tension/compression fatigue limit for reference diameter $\sigma_{W,zd,N}$** [N/mm^2], **Fatigue limit for bending for reference diameter $\sigma_{W,b,N}$** [N/mm^2], **Shear stress fatigue limit for reference diameter $\tau_{W,s,N}$** [N/mm^2], **Torsion fatigue limit for reference diameter $\tau_{W,t,N}$** [N/mm^2]
  - **Elongation at break A:** [%]

– **FKM Group:** Selection list showing the material group to which the entry belongs.

8.5.35.11 Material of gears

- **Comment:** Text field for your own use
- **File for hardness curve:** the database entries refer to external tables (see External tables (on page I-122)). Tables for the hardness curve begin with Z22-???.dat. Measured hardness value of the material shown as a graphic in module Z22. Does not influence the calculation. Here is an example of how to create this type of hardness curve in an external table.

```
-- ---------------------------------------------------------------------------
-- File    = ~modname: z22-100.dat~
-- Verlauf der Haerte des Werkstoffs/Curve of the material hardness
-- Werkstoff/Material: GGG50 180HB
--
-- (c) KISSsoft AG, CH-8634 Hombrechtikon
--
-- 1.0     14.10.1995  em        Inital Version
--
-- INPUT  nummer                Punktnummer/ number of the point
-- OUTPUT Tiefe, HaerteHV       Tiefe(mm), HV-Wert/Depth(mm), HV-Value
-- ---------------------------------------------------------------------------

:TABLE LIST haerteHV
  INPUT nummer
  OUTPUT Tiefe, HaerteHV
DATA
1     0.0    890
2     0.1    523
3     0.2    470
4     0.3    295
5     0.4    290
6     0.5    207
7     0.6    207
8     0.7    207
9     1.0    207
10    1.5    207
END
```

Figure 8.08: Example of a hardness curve definition (Z22-100.dat)

- **Woehler line file:** the database entries refer to external tables (see External tables (on page I-122)). Tables for Woehler lines begin with Z014-10?.dat. You **must input a file name for plastics** here. The file contains material data (Woehler lines, Young's modulus, etc.) used in the calculation.
  For metallic materials you can also input a file name here. The file contains the Woehler lines for resistance to bending and for Hertzian pressure that are used in the calculation, if the `Calculate with own Woehler line` flag is set.

```
-- Data for significant No of cycles (edge points for interpolation of Woehler line)
:TABLE FUNCTION EdgeCycle
      INPUT X number  TREAT NEXT_BIGGER
DATA
        1        2        3        4        5        6        7        8        9
      1E4      1E5      1E6      2E6      4E6      1E7      1E8      4E8     1E10
END

-- Data for Hertzian pressure sigH
:TABLE FUNCTION FlankSigH
      INPUT X Cycles    TREAT LOG
DATA
        0      1E4      1E5      1E6      4E6      1E7      1E8      4E8      1E9
     1200     1200     1050      900      850      848      846      844      844
END

-- Data for fatigue strength tooth root sigF
:TABLE FUNCTION FootSigF
      INPUT X Cycles    TREAT LOG
DATA
        0      1E4      1E5      1E6      2E6      1E7      1E8      2E8      1E9
      500      500      450      400      380      370      360      350      340
END
```

Figure 8.09: Example of a file with Woehler lines for a metallic material

- **Endurance limit root (ISO, DIN/ AGMA 2101) σ_{Flim}/s_{at}, Endurance limit flank (ISO, DIN AGMA 2101) σ_{Hlim}/s_{ac}:** [N/mm^2] Endurance limit values specified in DIN 3990 or ISO 6336 Part 5.
- **Endurance limit root (AGMA 2001) s_{at},Endurance limit flank s_{ac} (AGMA 2001):** [lbf/in^2] Strength calculation based on AGMA 2001.
- **Mean peak-to-valley roughness root/flank R_{zF}/ R_{zH}:** [µm]
- **Thermal contact coefficient B_M:** [N/mm/s$^{0.5}$/K] This coefficient is needed to calculate the flash factor. You will find more information about this in DIN 3990, Part 4, equations 3.11, 4.17, 4.18, 4.19. For the most commonly used materials it is 13.795.

8.5.36 Roller bearing

Roller bearing tables are sub-divided into 2 different tabs:

- Basic data tab
- Internal geometry tab

8.5.36.1 Roller bearing basic data

- **Bearing label:** The codes for the bearing series are as specified in DIN 623 Part 1.
- Main dimensions of the bearing: **Inside diameter d** [mm], **External diameter D** [mm], **Bearing width b** [mm], **Corner radius r_{smin}** [mm]
- **Dynamic load number C:** [kN]

- **Static load number C_0:** [kN]
- Factors **X1, Y1, X2, Y2, e, e0, X01, Y01, X02, Y02**

	Defining individual factors:
X1,Y1,e:	Coefficients in formula *P = X1*Fr + Y1*Fa* for *Fa/Fr <= e*
X2,Y2:	Coefficients in formula *P = X2*Fr + Y2*Fa* for *Fa/Fr > e*
X01,Y01,e0:	Coefficients in formula *P0 = X01*Fr + Y01*Fa* for *Fa/Fr <= e0*
X02,Y02:	Coefficients in formula *P0 = X02*Fr + Y02*Fa* for *Fa/Fr > e0*
X1,Y1,X2,Y2,e:	For some bearings, these values are not imported from the database. Instead they are imported from the files, depending on the axial force.

Ball bearing: dependent on *f0*Fa/C0*

- at normal bearing clearance: Data is imported from file W05-100.dat
- at bearing clearance C3. Data is imported from file W05-101.dat
- at bearing clearance C4: Data is imported from file W05-102.dat

Shaft bearing: dependent on *f0*Fa/C0/i*

for bearings with contact angle 15°

- Single bearing: Data is imported from file W05-103.dat
- Bearing in back-to-back arrangement or face-to-face arrangement from file W05-104.dat

- **Speed limit using grease lubrication n_{Gmax}:** [1/min]
- **Speed limit using oil lubrication n_{Omax}:** [1/min]
- **Weight m:** [kg]
- **Contact angle α_0:** [°] Input the contact angle for high precision angular contact ball bearings, ball bearings etc.
  for four-point contact bearings: If you input 0° this is set to 35°
  for axial spherical roller bearings: If you input 0° this is set to 50°.
- **Permitted axial force F^*_{azul}:** [-] Input the permitted axial force in % of Fr. The permitted axial force is not checked if you input 0.

- **Maximum set angle α:** [min] If you input 0, the angle adjustability (i.e. a comparison of the permitted angular deviation of the shaft with the effective angular deviation in the bearing) is not checked.
- **Thermal reference speed $n_{\theta r}$**: [1/min]
- currently not evaluated in KISSsoft: **Availability** (0=in stock; 1=not in stock), **price** [in local currency]
- **Addition A to E:** You can input additional data for specific types in these fields. (see table: Use of additions A-E.)
- **radial** and **axial spring stiffness c_r ,c_a**: [N/µm]
- **Spring stiffness for bending c_{rot}:** [Nm/°] Input spring stiffness for bending.
- **Coefficient f_0:** Required to determine x and y (for example, for deep groove ball bearings) as these values depend on the coefficient *f0*Fa/C0*.
- **Minimum load P/C:** Minimum load P/C (P: average dynamic equivalent load: C: dynamic load rating) is usually:
  - 0.01 for ball bearings with a cage
  - 0.02 for roller bearings with a cage, 0.04 for pure roller bearings with a cage

If you input 0 in the database, these values are used automatically in the calculation.

This entries only apply to radial load. The minimum axial force is calculated directly in the software.

- **Fatigue load limit C_u:** Coefficient for calculating the extended service life

Type	Addition A	Addition B	Addition C	Addition D	Addition E
Angular contact ball bearing (single row)		Displacement a (mm) (*2)			
Angular contact ball bearing (paired)		Displacement a (mm) (*2)			
Axial cylindrical roller bearing		Coefficient A (*1)		Max. axial force (kN)	
Taper roller bearing (single row)		Width B (mm)	Displacement a (mm) (*2)	Dimension C (mm)	
Taper roller bearing (paired, O)		Dimension T (mm) (*1)		Dimension C (mm) (*1)	
Taper roller bearing		Dimension 2B		Dimension 2T	

(paired, X)		(mm)		(mm)	
Barrel-shaped and spherical roller bearings	Calculation coefficient k1 (SKF: CARB, Internal geometry)	Calculation coefficient k2 (SKF: CARB, Internal geometry)			
Axial spherical roller bearings	Mass d1 (mm)	Mass T2 (mm)	Mass D1 (mm)	Mass T1 (mm)	Coefficient A (*1)

Table 8.4: Use of additions A-E

Descriptions given in additional data conform with those in the INA/FAG catalog 2008.

(*1) Values are only used for SKF bearings, as specified in SKF catalog 2005.

(*2) Values for the mass a for FAG bearings have been provided by the manufacturer up to the center point. In KISSsoft, half the bearing width was then added to this value for the database (this may result in values that vary slightly from those in the bearing catalog). The value at the bearing center was used in this calculation. This corresponds to the data we received from the manufacturer.

8.5.36.2 Roller bearing Internal geometry

Description of the **Inner geometry** tab:

Inner geometry data is not yet available for every bearing type.

The **Material ID** is present in every table in which you must select a material for the balls. However, this is not yet taken into account.

List of bearings whose inner geometry is taken into account.

You need the details documented below in order to calculate inner geometry.

You can specify a user-defined roller profile definition file (".dat") for roller bearings. The third column is the custom defined roller profile, given relative to the roller diameters in non-dimensional form. (Acceptable range of values from 0 to 0.5). The second column is the position on the rollers of the profile defined above, and is given relative to the roller length in a non-dimensional form. The origin is the roller center, so the acceptable range here is from -0.5 to +0.5. The first column is the data index, and is not used in the calculation.

- Deep groove ball bearing (single row), four-point contact bearing: **Number of balls Z** [-], **Ball diameter D_W** [mm], **Reference diameter D_{PW}** [mm], **Inside diameter of the rim, pressure side D_{BI}** [mm], **Outside diameter of the rim,**

pressure side D_{BA} [mm], **Radius of curvature, inside ri** [mm], **Radius of curvature, outside ro** [mm]

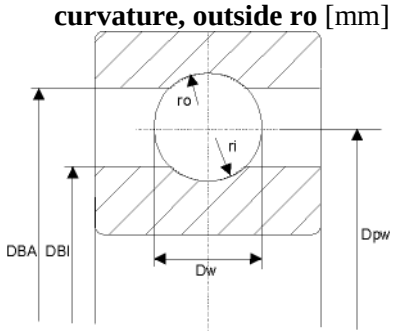


Figure 8.10: Mass of the deep groove ball bearing

- Angular contact ball bearing (single row): **Number of balls Z** [-], **Ball diameter D_W** [mm], **Reference diameter D_{PW}** [mm], **Inside diameter of the rim, pressure side D_{BI}** [mm], **Outside diameter of the rim, pressure side D_{BA}** [mm], **Radius of curvature, inside ri** [mm], **Radius of curvature, outside ro** [mm], **Minimum initial tension v_{min}** [µm], **Maximum initial tension v_{max}** [µm], **Minimum pretension force F_{vmin}** [N], **Maximum pretension force F_{vmax}** [N]

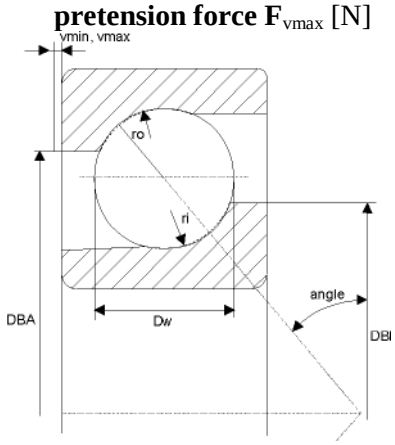


Figure 8.11 Mass of the angular contact ball bearing

- Cylindrical roller bearing (single row): **Number of rollers Z** [-], **Diameter of roller D_W** [mm], **Reference diameter D_{PW}** [mm], **Inside diameter of the rim, pressure side D_{BI}** [mm], **Outside diameter of the rim, pressure side D_{BA}** [mm], **Roller length L_{WE}** [mm], **Axial displacement possibility non-locating bearing v_l** [mm], **Axial displacement possibility fixed bearing v_f** [mm]

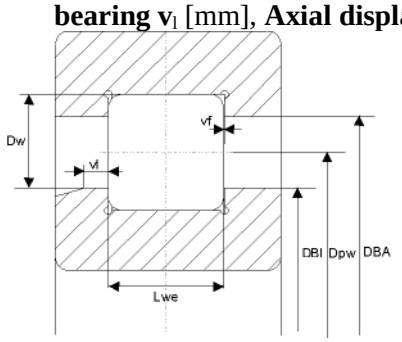


Figure 8.12: Mass of the cylindrical roller bearing

- Cylindrical roller bearing (double row): **Number of rollers Z** [-], **Diameter of roller D_W** [mm], **Reference diameter D_{PW}** [mm], **Inside diameter of the rim, pressure side D_{BI}** [mm], **Outside diameter of the rim, pressure side D_{BA}** [mm], **Roller length L_{WE}** [mm], **Row distance a** [mm]

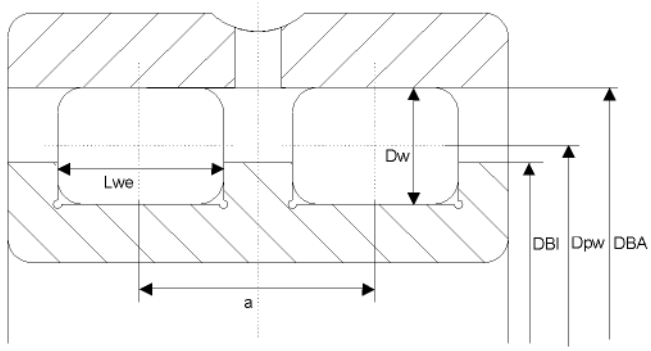


Figure 8.13: Mass of the double row cylindrical roller bearing

- Taper roller bearing (single row): **Number of rollers Z** [-], **Diameter of roller D_W** [mm], **Reference diameter D_{PW}** [mm], **Roller length L_{WE}** [mm]

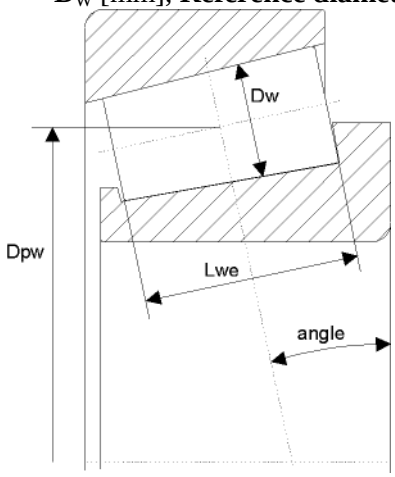


Figure 8.14: Mass of the taper roller bearing

- Double row self-aligning roller bearings **Number of rollers Z** [-], **Diameter of roller D_W** [mm], **Reference diameter D_{PW}** [mm], **Inside diameter of the rim, pressure side D_{BI}** [mm], **Outside diameter of the rim, pressure side D_{BA}** [mm], **Radius of curvature, inside ri** [mm], **Radius of curvature, outside ro** [mm]

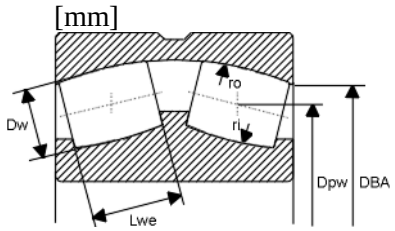


Figure 8.15: Mass of the double row self-aligning roller bearing

- Needle roller bearing, Needle cage: **Number of rollers Z** [-], **Diameter of roller D_W** [mm], **Reference diameter D_{PW}** [mm], **Roller length L_{WE}** [mm], **Axial displacement possibility non-locating bearing v_l** [mm]

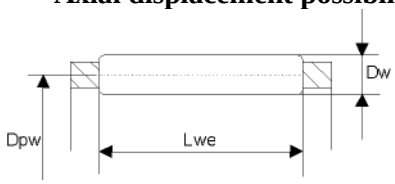


Figure 8.16: Mass of the needle roller bearing/needle cage

- Deep groove thrust ball bearing **Number of balls Z** [-], **Ball diameter D_W** [mm], **Reference diameter D_{PW}** [mm], **Radius of curvature, inside ri** [mm], **Radius of curvature, outside ro** [mm]

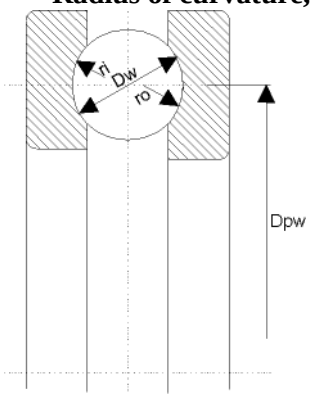


Figure 8.17: Mass of the deep groove thrust ball bearing

- Cylindrical thrust roller bearing **Number of rollers Z** [-], **Diameter of roller D_W** [mm], **Reference diameter D_{PW}** [mm], **Roller length L_{WE}** [mm]

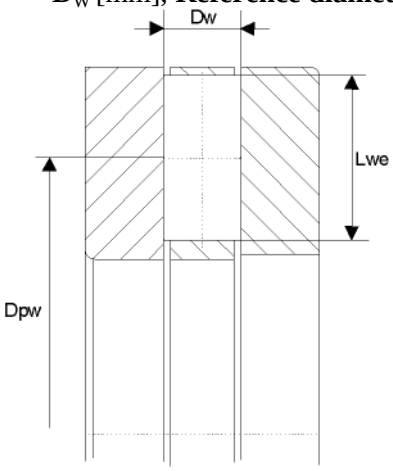


Figure 8.18: Mass of the cylindrical thrust roller bearing

- Double row self-aligning thrust ball bearing **Number of rollers Z** [-], **Diameter of roller D_W** [mm], **Reference diameter D_{PW}** [mm], **Roller length L_{WE}** [mm], **Distance L_{WC}** [mm], **Radius of curvature, inside r_i** [mm], **Radius of curvature, roller R_p** [mm], **Radius of curvature, outside r_o** [mm]

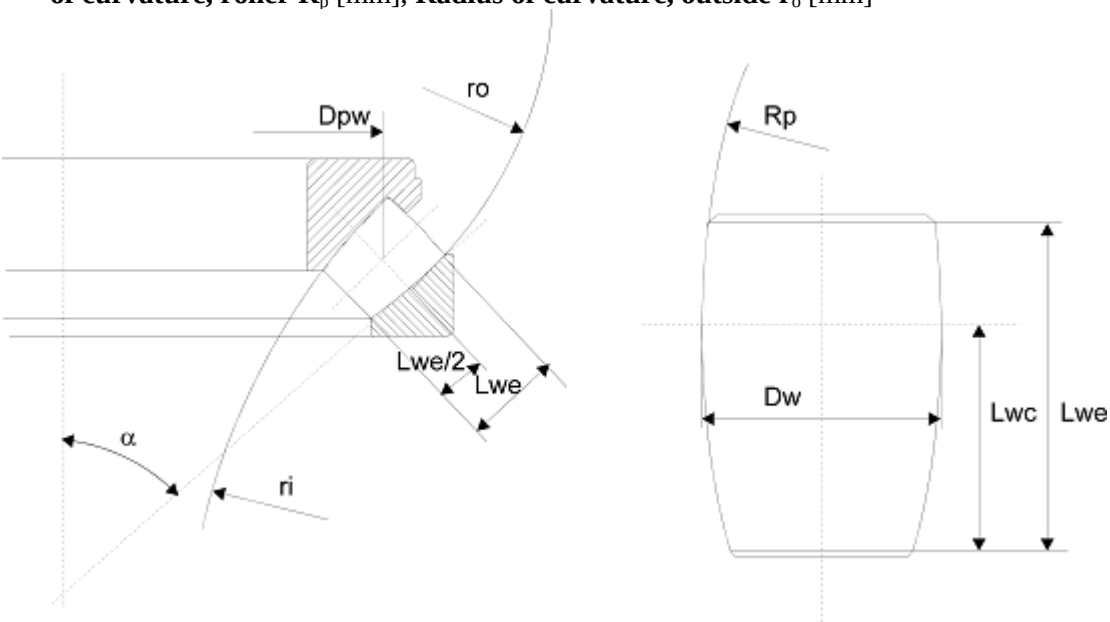


Figure 8.19: Mass of the Double row self-aligning thrust ball bearing

8.5.37 Roller bearing tolerance

- **File name:** the database entries refer to external tables (see External tables (on page I-122)). Tables for roller bearings begin with W05-??-??.dat.

8.5.38 Roller bearing Tolerance classes

- **File name:** the database entries refer to external tables (see External tables (on page I-122)). Tables for roller bearing tolerance classes begin with W05-???.dat.

8.5.39 Tooth thickness tolerances

- **File name:** the database entries refer to external tables (see External tables (on page I-122)). Tables for tooth thickness tolerances begin with Z01-???.dat or Z9-???.dat.

- **Interpret as:**
  - Tooth thickness allowance: the data is interpreted as a tooth thickness allowance.
  - Base tangent length allowances: the data is interpreted as the base tangent length allowances (or normal play).

8.5.40 Toothed belt standard

- **File name:** the database entries refer to external tables (see External tables (on page I-122)). Tables for the toothed belt standard begin with Z091-???.dat.
- **Calculation method:**
  - 1)"normal" toothed belts (RPP)
  - 2) GT types (PolyChain)
  - 3) AT types (Brecoflex)
  - 4) PG types (PowerGrip)

  Differences:

  - Special calculation for toothed belts with integrated steel rope (Method 3)
  - Calculating the operating factor: the special factor for the speed increasing ratio is added (Method 1, 2, 4) or multiplied (Method 3)
  - Additional performance table for higher performance at greater conversions (Method 2)
- **Calculation method for belt pretension:**
  - 1) in % of the circumferential force; indentation depth = 1/50 of the tension length
  - 2) in % of the maximum permitted circumferential force; indentation depth = 1/50 of the tension length
  - 3) in % of (operating factor*performance(W)/circumferential speed reference circle (m/s)) (according to DAYCO RPP Panther = 1/64 of the tension length
  - 4) in % of the circumferential force ; indentation depth = 1/64 of the tension length
- **Nominal range for power table b:** [mm] belt width, which corresponds to the performance data stored in the file (see file name).

- **Coefficient for belt pre-tension f:** 0 ... 1.0 (%-factor for calculating the coefficient for belt pre-tension)
- **Maximum belt speed v_{max}:** [m/s]
- **Summand for operation F_s:** No influence
- **Pitch p:** [mm] pitch of the toothed belt
- **Elasticity E:** [N] Elasticity = force that doubles the length of a belt (with nominal width). If you do not know this value, enter 0 as the threshold value (in this case the elasticity is ignored when the bending test is performed).
- **Strain ε:** [%] Strain along the total length of the belt
- **Weight per length q:** [kg/m/mm] per meter length and millimeter width

File contents:	
List of suggested standard tooth numbers for toothed lock washers	*:TABLE LIST z.RadZahne*
List of suggested belt standard tooth numbers	*:TABLE LIST z.NormZahne*
Minimum number of teeth, depending on the speed (small disc)	*:TABLE FUNCTION z091k.factorINCR*
Correction factor for powering up, depending on the ratio (this is added to the operating factor)	*:TABLE FUNCTION z091k.factorINCR*
Transmissible power depending on the number of teeth (small disc) and speed (small disc)	*:TABLE FUNCTION z091k.powerNr*
Correction factor for the number of contacting teeth (small disc)	*:TABLE FUNCTION z091k.factorCorrEZ*
Correction factor for belt length	*:TABLE FUNCTION z091k.factorLength*
Correction factor for belt width	*:TABLE FUNCTION belt.bth*
Correction factor for belt width (same values as shown in the table above)	*:TABLE FUNCTION belt.beff*
Disc width depending on belt width	*:TABLE FUNCTION z091k.ScheibenBreite*
Belt type-layout: minimum transmissible power (lower limit) depending on the speed (small disc)	*:TABLE FUNCTION z091k.kWlower*
Belt type-layout: maximum transmissible power (upper limit) depending on the speed (small disc)	*:TABLE FUNCTION z091k.kWupper*

Chapter 9
Description of the public interface

9.1 Interfaces between calculation programs and CAD - Overview

The closest contact point of calculation programs within a CIM concept is the one with the drawing program (CAD). KISSsoft's public data interface can be freely formatted, allowing for very powerful communication with third party programs.

All input and output data can be exported in ASCII format. The scope and format of this data is freely definable. Each calculation module contains a special, editable report file for this purpose. The MMMMUSER.RPT[1] files are used as a template for this data transfer. The default setting is that these files are empty. If you want to output data over the interface, you first have to expand the templates. External programs can, in addition, transfer input data (also in ASCII format) to calculation modules. This data is read automatically during startup and the data is displayed on the screen.

9.1.1 Efficient interfaces

An automated data transfer between calculation and CAD should only be established if the benefits are considerably larger than the effort required. For example, an interface between a bolt calculation program and CAD is only of secondary importance since the information to be transferred (for example that, due to the calculation, an M10 bolt has to be selected) is too limited and could be transferred much faster "by hand". If, however, a standard parts library with bolts is available, the bidirectional link between the three components (calculation program, standard parts library and CAD) can prove very efficient.

The following efficient interfaces are available (but this list can be extended):

- General

  It should be possible to start the calculation programs from the CAD environment (for example by pressing a function key). This enables you to carry out a short calculation while you are drawing, transfer the results and then continue drawing.

- Shafts and bearing calculation

  – Output of a contour from the CAD (i.e. a shaft from detailed or drawing with combined elements) and reading it into the calculation program. (Problem: in many CAD programs it is unfortunately rather difficult to define the contour to be exported.)

[1] *The character string MMMM in a file name is a placeholder for the module to which the file refers. Example: M040USER.RPT*

- Output of a shaft that has been optimized in the calculation program (including roller bearings etc.) and reading it into CAD as drawing information.
- Transfer of bending and similar data into the CAD.
- Roller bearings and plain bearings are calculated and then the contour is transferred to the CAD system. (Frequently, the CAD already contains information on roller bearings, so that only the bearing label is of interest.)

Gear calculation

- Calculation of fabrication data in the program and transfer of the required values to the CAD as text. This is a very important function, since the recording of the data is rather error-prone, and the consequences of errors can be correspondingly serious.

  Calculation of the exact tooth form in Print Preview and transfer to the CAD. (Although this results in very pretty drawings, it usually does not supply any necessary information, except if the data undergoes further processing, i.e. via transfer onto a wire electro-discharge machine.)

- Transfer of the schematic axial section or the Print Preview of the gears to the CAD system (can, however, be done just as fast "by hand" in CAD).

Machine elements

Transfer the contour of calculated machine elements to the CAD such as bolts, V-belt sheaves etc. (Frequently, the CAD station already contains appropriate, preprogrammed information, so that only the parts definition is of interest).

Shaft-hub connection

The sizing or proofing of connections should be implemented directly in a CAD system, so that known data from the CAD can be transferred into the calculation and the results of the calculation can in turn be returned to the CAD system.

9.1.2 Open interfaces concept in KISSsoft

The KISSsoft interfaces concept has a simple, yet very flexible structure.

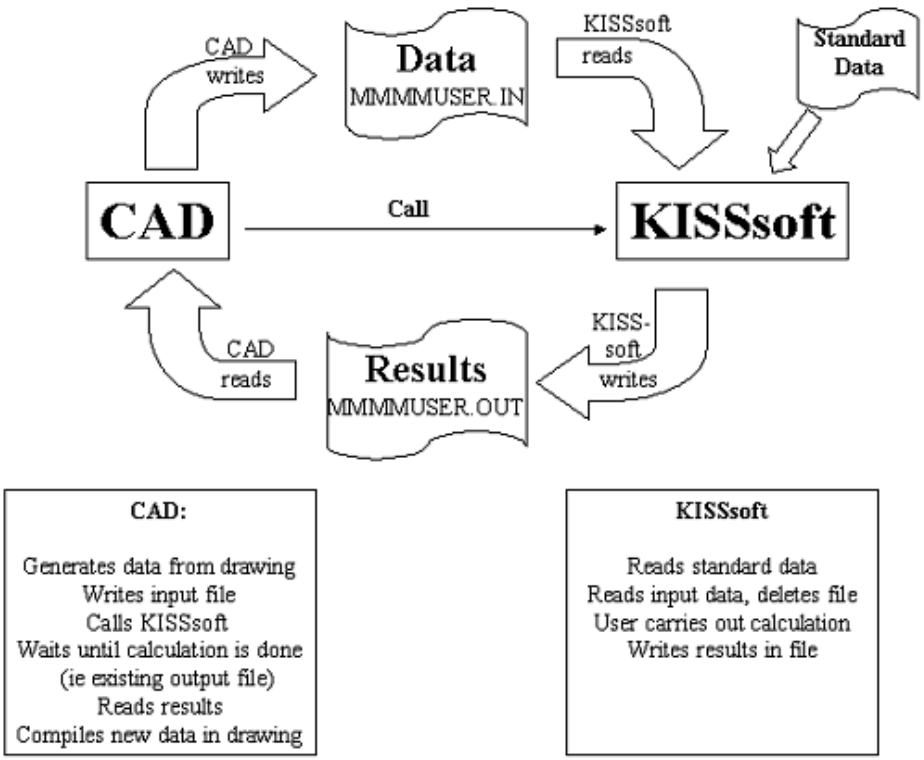
It should be possible to integrate calculation programs into all kinds of CAD systems as simply as possible, and use them in different environments (operating systems such as MS Windows or UNIX).

The interface mechanism between CAD and KISSsoft is based on a text data record (ASCII file), and an ID is transferred together with the numerical value for

all transfer data (see Figure in example (see section "Example: Interference fit calculation" on page I-162)). This data record can be of variable length, while only the values that are known in the CAD will be transferred. This depends on the CAD system and the currently active drawing.

The data record transferred by the third party program will be tested for completeness and consistency by KISSsoft and if it should prove necessary, additional data will be requested in the KISSsoft input system. Subsequently, the calculation will be carried out and the output data important for the CAD will be written into a second text data record and returned to the CAD. By using the report generator you can select any format for the output file, i.e. KISSsoft adapts itself to the third party program. The CAD can now read the data required by the situation and selectively process them.

This concept results in simple interface forms, consequently enabling even non-specialists to write applications quickly.



9.2 Defining input and output

9.2.1 Preamble

- In this description the KISSsoft program is always taken as a reference, i.e. an input file for KISSsoft becomes an output file for the third party program and vice versa.
- For automatic data exchange with other programs you will require files with the name *MMMMUSER.RPT*. You can adapt these files to your own requirements. However, if you have purchased KISSsoft interfaces, you should act with caution, since these files are also required for these interfaces.

File name	Storage location	Description
MMMMUSER.IN	<CADDIR> *)	Input file for KISSsoft (is written by the third party program) User's temporary input file (= will be deleted when imported into KISSsoft)
MMMMUSER.OUT	<CADDIR>	KISSsoft output file (will be written by KISSsoft and read by the third party program). Temporary (= should be deleted by the third party program)
MMMMUSER.RPT	<KISSDIR>	Defines the output format (similar to report), can be permanent/optional (= will usually be created once and is retained)
Z10Gear1.RPT *Z10Gear2.RPT* *Z10Gear3.RPT* *Z10Gear4.RPT*	<KISSDIR>	Defines the output format for the manufacturing data in the case of cylindrical gears (see below). Corresponds to MMMMUSER.rpt for this special case.
Z10Gear1.OUT ⋮	<KISSDIR>	Output file of the toothing stamp for cylindrical gears.
Z70Gear1.RPT *Z70Gear2.RPT*	<KISSDIR>	Defines the output format for bevel gears.
Z17Gear1.RPT *Z17Gear2.RPT*	<KISSDIR>	Defines the output format for crossed helical gears.
Z80Gear1.RPT *Z80Gear2.RPT*	<KISSDIR>	Defines the output format for worm gears.
Z9aGear1.RPT *Z9aGear2.RPT*	<KISSDIR>	Defines the output format for spline connections.
Z??Gear1.OUT *Z??Gear2.OUT*	<CADDIR>	Toothing stamp, similar to definition files.

*) If you specify the complete file name including the directory, it can also be read from any location.

9.2.2 Requirements placed on the third party program

To successfully operate KISSsoft within a third party program, the following minimum requirements must be met. The third party program must

1. have a query mechanism (i.e. macro language) in order to provide information, e.g. input data,
2. be able to write and read ASCII files,
3. be able to start a program.

9.2.3 Used files

9.2.3.1 Input file

An input file with the name *MMMMUSER.IN* will be used. It has the same structure and the same function as the saved calculations, except for its temporary status. The values will be assigned to the KISSsoft variable names with =. A separate row will be used for each variable.

EXAMPLE

VERSION=2.5;
m02Aw.dWa=30;
m02Aw.lW=20;
m02An.lN=25;

The input file will be read after the default values are predefined (see page I-49), i.e. the values of the temporary input file will overwrite the values set by the default.

Note: Temporary input files are used for frequently changing variables such as geometry and/or performance data: data which typically changes from calculation to calculation. It would also be possible to write this data into the template files, since they represent normal input variables. This would however mean that the program generating these files had to interpret the data that has already been written, i.e. has to accept permanent constraints, in order to be able to completely define the standard and to reset it again at the end.

9.2.3.2 Output file

To return the data that is relevant for the KISSsoft calling program, the specified output file *MMMMUSER.OUT* will be generated immediately after a calculation.

The scope and the format of the output file will be defined in a report template called *MMMMUSER.RPT*.

This means that KISSsoft can fully adapt itself to the syntax of a third party program. The range of commands and the syntax of the report generator are described in the Reports (see section "Report templates" on page I-104) section. To help you, example report files are supplied.

9.2.4 Service life of files

The input file *MMMMUSER.IN* is generated by the third party program and, after having been read, will be deleted by KISSsoft. The output file *MMMMUSER.OUT* will be deleted when KISSsoft starts, and be written again after a calculation.

9.2.5 Explicitly reading (importing) and generating data

In addition to the automatic definition described above, you can click the `File > Interface > Import data` menu to read (import) data specifically via or click `File > Interface > Export data`. You are therefore completely free to choose the point in time and thus use it for many varied tasks, i.e. the generation of an order form etc.

9.3 Example: Interference fit calculation

The following example of the Interference fit assembly calculation is used to illustrate the way that the KISSsoft interfaces concept works, in more detail.

For the Interference fit assembly between the gear rim and the cylindrical gear hub, the user needs to find the one tolerance pairing that meets the following constraints:

Permanent torque MD = 88000 NM

The tolerance pairing involves a system of the standard drill hole *(H)* .

Safety	against sliding > 1.4
	against fracture of the wheel or hub > 1.5
	against fracture of the gear rim > 1.5
	against the yield point of the hub > 1.1
	against the yield point of the gear rim > 1.1

Procedure:

The necessary information for the geometry will be extracted direct from the drawing, with a suitable CAD routine, and converted into the interfaces format defined by KISSsoft:

```
m01allg.df=640
m01n.da=800
m01w.di=242
m01allg.l=200
```

Content of the *M010USER.INI* file

Then, the user starts the KISSsoft module. It accepts the geometry data and displays it in the main screen.

In the main screen, the user enters any parameters that are still missing, the torque and the materials, and then starts the calculation. KISSsoft also allows the user to size the tolerance pairing. Here, the user is asked to select the suitable tolerance combinations from a list and the system then carries out the calculation with the user's final selection.

After the user has concluded the calculation, the results file is automatically converted to a format that can be read via the CAD macro. The format of this result file is defined via the templates file *M010USER.RPT*:

```
[SHAFT]
```

```
ntol_max = %f{m01w.tol.max}
ntol_min = %f{m01w.tol.max}
ntol_bez = %s{m01w.tol.bez}

[HUB]
ntol_max = %f{m01n.tol.max}
ntol_min = %f{m01n.tol.max}
ntol_bez = %s{m01n.tol.bez}
```

Content of the *M010USER.RPT* file

The result has then following appearance:

```
[SHAFT]
wtol_max = 390.000000
wtol_min = 340.000000
wtol_bez = s6

[HUB]

ntol_max = 50.000000

ntol_min = 0.000000

ntol_bez = H6
```

Content of the *M010USER.OUT* file

Via the macro, this data will now be attached directly to the appropriate dimension in CAD.

Summary:

The individual tasks are therefore split up: Each side of the interface will perform only the tasks it is best suited to. The CAD administers the geometry and passes this information on to the calculation program, which knows how to process the data, and which, in turn, will return the result to the CAD.

By using the defined interface an efficient combination of CAD and calculation program can be achieved.

9.4 Geometry data

KISSsoft contains different interfaces for transmitting geometry data (contours, drawings):

- DXF format (recommended for communication with most CAD systems)
- IGES format (with which tooth forms can be exported as splines)
- BMP format (Windows bitmap)
- JPG/JPEG format (pixel image)
- PNG (Portable Network Graphic) format

9.5 COM Interface

KISSsoft offers the possibility of remote control via a COM interface. It can easily be accessed from Visual Basic or Excel.

9.5.1 Registering the server

Now register the KISSsoft COM server on your local computer. To do this, enter these two command lines in a Windows input prompt in the KISSsoft installation BIN directory:

```
KISSsoftCOM.exe /regserver
regsvr32 KISSsoftCOMPS.dll
```

You will need administrator rights to register the program.

9.5.2 Server functionality

The server provides a number of functions that you can use to start a calculation module, read or set values, and perform a calculation.

- `GetModule([in] BSTR module, [in] VARIANT_BOOL interactive)` starts a calculation module from the module descriptor (e.g. Z012 or W010). "interactive" defines whether the calculation module is to be generated with a graphical user interface.
- `Calculate()` performs the main calculation for the active module.
- `CalculateRetVal([out, retval] VARIANT_BOOL* isOk)` runs the main calculation for the active module and returns a value to tell you whether the calculation is OK.
- `SetVar([in] BSTR name, [in] BSTR value)` is a function with which you can set variables to a required value. This data is transferred as text. You will find the variable names in the report templates, but there is no guarantee that all these variables will remain the same in the future.
- `GetVar([in] BSTR name, [out, retval] BSTR* value)` returns a variable from KISSsoft as text.
- `ShowInterface([in] VARIANT_BOOL wait)` displays the graphical user interface. Use the "wait" parameter to specify whether the function is to wait until the dialog is closed.
- `IsActiveInterface([out, retval] VARIANT_BOOL* isActive)` shows whether a KISSsoft dialog is active.

- `IsActive([out, retval] VARIANT_BOOL* isActive)` shows whether a module has been loaded.
- `ReleaseModule()` releases the loaded module again. You must always release a module again, to reduce the load on the server.
- `LoadFile([in] BSTR filename)` loads the specified file.
- `SaveFile([in] BSTR filename)` saves the calculation in the specified file.
- `GetININame([OUT, retval] BSTR* name)` supplies the name of the loaded INI file.
- `GetVersionFromFile([in] BSTR filename, [out, retval] BSTR* version)` supplies the version number (e.g. 2.6) of the KISSsoft module in the calculation file. (The version number depends on which module is being used)
- `GetModulFromFile([in] BSTR filename, [out, retval] BSTR* name)` supplies the KISSsoft module name (e.g. M040), in the calculation file.
- `GetKsoftVersionFromFile([in] BSTR filename, [out, retval] BSTR* kSoftVersion)` supplies the KISSsoft version number (e.g. 03-2011), given in the calculation file.
- `GetKsoftVersion([out, retval] BSTR* kSoftVersion)` supplies the KISSsoft version (e.g. 03-2011), that is registered and was started via the COM interface.
- `GetKsoftVersionSettings([out, retval] BSTR* kSoftVersionSettings)` reports the KISSsoft version (e.g. 03-2014) of the temporary settings folder in which the personal settings are stored.
- `SetSilentMode([in] VARIANT_BOOL silent)` defines whether messages are to be hidden or not, so that calculations can be performed without the user having to confirm system prompts.
- `Report([in] LONG show)` writes the report. You can specify whether or not this report is to be displayed. The report is created in the Temp directory in the "KISS_?" sub folder.
- `ReportWithParameters([in] BSTR infile, [in] BSTR outfile, [in] LONG show, [in] LONG type)` creates the report using the specified report template (infile) in the predefined place with the predefined name (outfile) and supplies the file type. You can enter file names either with or without the path. When you enter the report template ("infile") you should also input the file extension (e.g."Z012ld0.rpt"). If you do not enter a path for this file the program will search the default directory (see also

Reports) for the file. You must also enter the file extension for an output file. If you do not specify the path, the file is saved to the Temp directory with the name "KISS_?". The file extension should match the specified type. Use the "show" parameter to define whether the report is to be displayed (or not). Use the "art" (meaning "type") parameter to define the output format. (art=0 → rtf format with *1 ; art=1 → rtf format without *1; art=2 → html format with *1 ; art=10 → txt format without *1 ; art=20 → txt format in Unicode without *1)

– *1 = takes into account the data level

– Examples of possible combinations: With default report templates → RTF format: ReportWithParameters("C:\Program Files (x86)\KISSsoft 03-2014\rpt\Z070ld0.rpt","C:\Temp\Z070ld0.rtf" ,1,0), HTM format: Call ksoft.ReportWithParameters("Z070ld0.rpt", "C:\Temp\Z070ld0.htm", 1, 2) ; with drawing stamp report template → TXT format: ReportWithParameters("Z10GEAR1d.rpt","C:\Temp\Z010GEAR1d.txt" ,1,10)

- `Message([out] VARIANT *strings, [out] VARIANT *types:, [out] LONG *numElem)` returns the messages from the last calculation in the first parameter, as an array containing strings. The second parameter contains the particular message type (error, warning, info). The number of existing messages is shown in numElem.
- `CallFunc([in] BSTR name)` Use these functions to run special calculations. A more detailed list of the available calculations is available on request.
- `CallFuncNParam([in] Variant paramArray)` Use these functions to run special calculations. A more detailed list of the available calculations is available on request.

9.5.3 Example of a call from Excel

The best way to describe the functionality is to use an example. To use KISSsoft from Excel, you must first activate the KISSsoftCOM type library in the Visual Basic Editor in `Tools>References`.

The first example shows how to use a single gear calculation to define the tip and root circles of a gear:

```
Public Sub ExampleKISSsoftCOM()

Dim ksoft As CKISSsoft

Dim da As String

Dim df As String
```

```
' get KISSsoft instance
set ksoft = New CKISSsoft

' get KISSsoft module for single gear
Call ksoft.GetModule("Z011", False)

' set values
Call ksoft.SetVar("ZR[0].z", "20")
Call ksoft.SetVar("ZS.Geo.mn", "5.0")
Call ksoft.SetVar("ZR[0].x.nul", "0.5")

' Calculate
Call ksoft.Calculate

' get values
da = ksoft.GetVar("ZR[0].da.nul")
df = ksoft.GetVar("ZR[0].df.nul")

' release module
Call ksoft.ReleaseModule

' release server
Set ksoft = Nothing

End Sub
```

The second example shows how to display the KISSsoft user interface:

```
Public Sub ExampleKISSsoftCOM()

Dim ksoft As CKISSsoft

Dim da As String

Dim df As String


' get KISSsoft instance

Set ksoft = New CKISSsoft


' get KISSsoft module for single gear

Call ksoft.GetModule("Z011", True)


' show interface

Call ksoft.ShowInterface(True)


' get values

da = ksoft.GetVar("ZR[0].da.nul")

df = ksoft.GetVar("ZR[0].df.nul")


Call ksoft.ReleaseModule


Set ksoft = Nothing


End Sub
```

The same example with "later binding" (the exact property or method is not determined until runtime, which enables you to compile the Visual Basic client without having to know the exact function of the call):

```
Public Sub ExampleKISSsoftCOM()

Dim ksoft As Object

Dim da As String
```

```
Dim df As String

' get KISSsoft Object
Set ksoft = CreateObject("KISSsoftCOM.KISSsoft")

' get KISSsoft module for single gear
Call ksoft.GetModule("Z011", True)

' show interface
Call ksoft.ShowInterface(True)

' get values
da = ksoft.GetVar("ZR[0].da.nul")
df = ksoft.GetVar("ZR[0].df.nul")

Call ksoft.ReleaseModule

Set ksoft = Nothing

End Sub
```

The fourth example shows a contact analysis that was run using the caControll.dat control file (you will find an example file in the dat folder) and they way messages were processed after the calculation:

```
Public Sub ExampleKISSsoftCOM()
On Error GoTo ExitOnErr
Dim ksoft As CKISSsoft

' get KISSsoft instance
Set ksoft = New CKISSsoft
```

```
' get KISSsoft module for gear pair
Call ksoft.GetModule("Z012", True)

' load File - change this to fit to a real file on your machine
Call ksoft.LoadFile("C:\yourPathHere\ExCOM3.z12")

' calculate
Call ksoft.Calculate

Dim ioData(3) as String
'Which calculation to start
ioData (0) = "CalculatePathOfContactForPairKS"
'controling file
ioData (1) = "C:\ yourPathHere\caControl.dat"
'Path for results
ioData (2) = "C:\ yourPathHere\prot"

'calculate contact analysis
Call ksoft.CallFuncNParam(ioData)

' Check for messages
Dim mess As Variant
Dim types As Variant
Dim numElem As Long
Dim typesElem As Long
Dim typesElemStr As String

Call ksoft.Message(mess, types, numElem)
```

```
If (numElem > 0) Then
    Dim msg As String
    For i = 0 To numElem - 1
        msg = CStr(mess(i))
        typesElemStr = CStr(types(i))
        typesElem = CLng(types(i))
        If (typesElem = 0) Then
            Call MsgBox(msg, vbInformation)
        ElseIf (typesElem = 1) Then
            Call MsgBox(msg, vbExclamation)
        Else
            Call MsgBox(msg, vbCritical)
        End If
    Next
End If

' close ksoft
Call ksoft.ReleaseModule

' no problems, so exit
Exit Sub
ExitOnErr:
    MsgBox ("error occurred when calling KISSsoft.")

End Sub
```

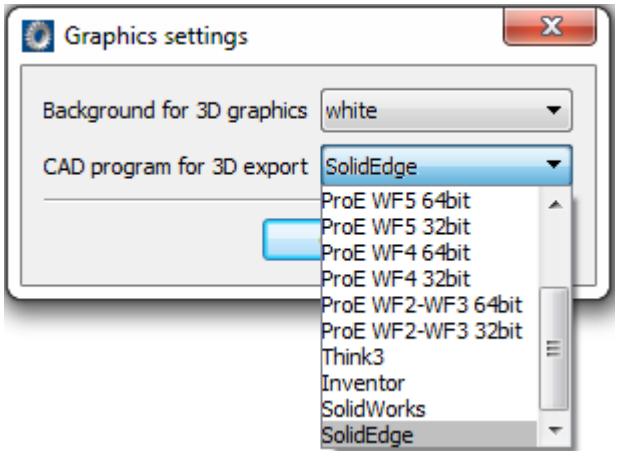
Chapter 10
3D interfaces

10.1 Overview of the available CAD interfaces and their functionality

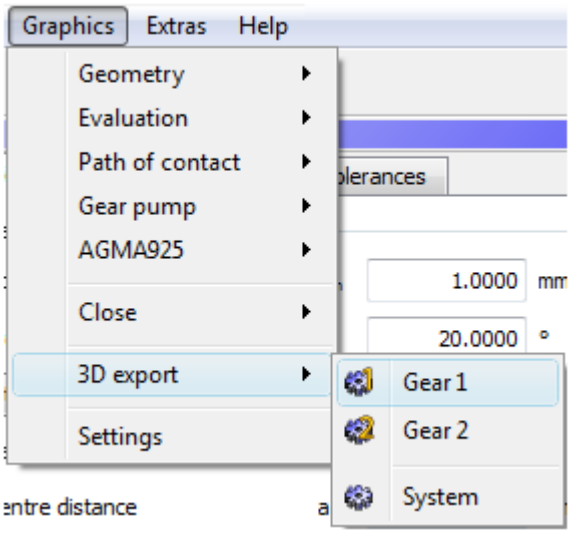
Feature/CAD	SolidWorks	Autodesk Authorized Developer Autodesk Inventor	SIEMENS Solution Partner PLM SolidEdge: Voyager Partner	SIEMENS Solution Partner PLM UGS NX: Level-Gold	PTC PartnerAdvantage Silver ProEngineer, Creo Parametric	CATIA Catia	think3 Shape a new world. Think3 thinkdesign	PTC CoCreate CoCreate, Creo Elements/ Direct Modelling	ASCON KOMPAS 3D	HiCAD	Parasolid / Neutral-formatschnittstelle (step)
Version	2011 - 2014	2011 - 2015	ST3, ST4, ST5, ST6	NX6 – NX9	Wildfire 3, 4 and 5, Creo Parametric	V5 (R14-R19)	2008.1	2006 – 2008, Creo Elements	V13, V14	2010 - 2013	
Cylindrical gears, spur/ helical gears	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Cylindrical gears, internal/ external teeth	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Rack	✓	✓	✓	✓	✗	✗	✗	✗	✓	✓	✓
Worm/helical gears	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Bevel gears, straight	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Bevel gears, helical	✗	✗	✗	✗	✗	✗	✗	✗	✗	✗	✓
Bevel gears, spiral	✗	✗	✗	✗	✗	✗	✗	✗	✗	✗	✓
Face gears	✗	✗	✗	✗	✗	✗	✗	✗	✗	✗	✓
Splines (Shaft-hub)	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Toothing on existing shaft	✓	✓	✓	✓	✓	✓	✗	✗	✗	✗	✗
Shafts	✓	✓	✓	✓	✗	✗	✗	✗	✓	✗	✓
CAD add-in menu	✓	✓	✓	menu-driven only	✗	Selection menu	✗	✗	✗	✗	✗
Manufacturing data	✓	✓	✓	✓	✓	✓	✓	✗	✗	✗	✗
64 bit version	✓	✓	✓	✓	✓	✗	✗	✓	✗	✓	✗

10.2 Generation of 3D gears

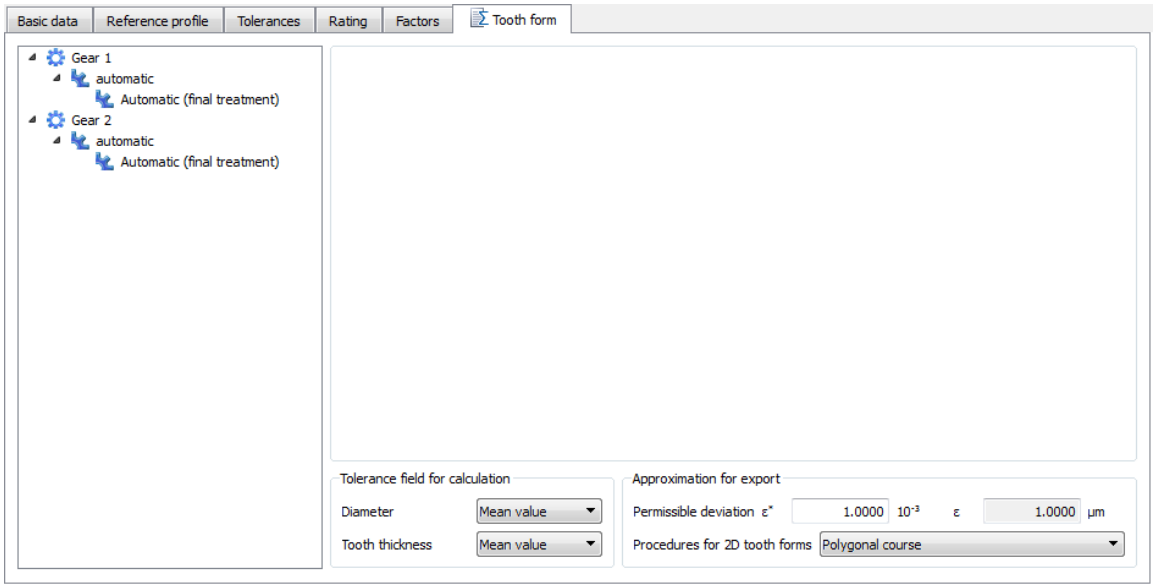
You must first perform a gear calculation to ensure that the results are consistent. Select `Graphics > Settings` to choose the CAD system to which you want to export the selected element.



Then, in the `Graphics > 3D export` menu option, you can select which individual gear and the configuration (only possible as individual gears) you want to generate.



In the case of Unigraphics NX, generation is only possible if you have started KISSsoft from the NX add-in menu, then run the gear calculation and pressed the required Generation button. In the case of Creo Parametric (ProEngineer), CATIA and Think3, the CAD interface must be opened so that the gear generation process can be started from KISSsoft. If you are working with the SolidWorks, Solid Edge, Inventor, Kompas or Creo Elements/Direct Modeling (CoCreate) CAD systems, click on the Generate button to start the CAD program if it has not already been opened.



The default setting runs the gear generation process with a permitted tooth form deviation (tolerance band) 1 μm. If this tolerance is too large, you can open the `Tooth form` tab to change the tolerance. Once this is changed, you must click `Calculate` again (Tooth form tab active), to transfer the inputs and recalculate the tooth form.

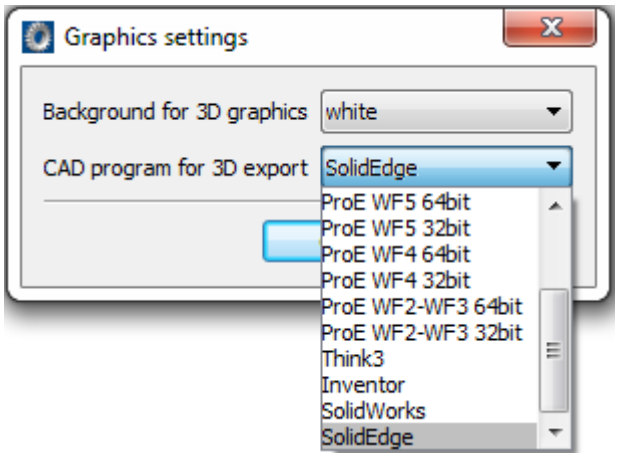
Changing the generation type in the Tooth form tab (polylines, circular pitch approximation, arc estimation, splines) only affects the 2D display. In NX, SolidWorks, Solid Edge and Kompas, the part is generated with splines. In Inventor, Think3, Creo Parametric (ProEngineer), CATIA and Creo Elements/Direct Modeling (CoCreate) it is created with arcs of circles. SolidWorks and Solid Edge also support other generation types, which you can change by entering the additional parameter `APPROXIMATION=1` in the `kiss.ini` (see page I-56) file under the relevant CAD.

In the case of the gears, the transverse section of the tooth space is usually cut out from a cylinder and then duplicated as a pattern. For worms with a helix angle of > 50° and number of teeth < 4, the tooth space is cut out in the axial section and then duplicated.

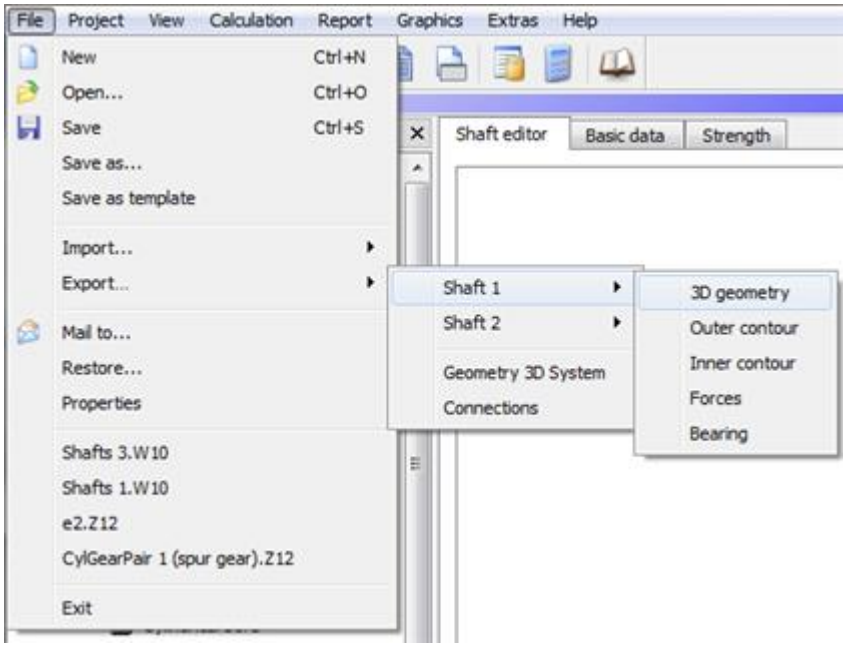
10.3 Generating 3D shafts

Until now it has only been possible to generate shafts in 3D in the SolidWorks, Solid Edge, Autodesk Inventor, Kompas and NX CAD systems.

First a shaft analysis must be performed to ensure the results are consistent. Select `Graphics > Settings` to choose the CAD system to which you want to export the selected element.



Then click `Graphics> 3D Export` to select the shaft and configuration (if you want to generate more than one shaft) that you require. In a configuration, each shaft is created individually, in sequence, in its own part.

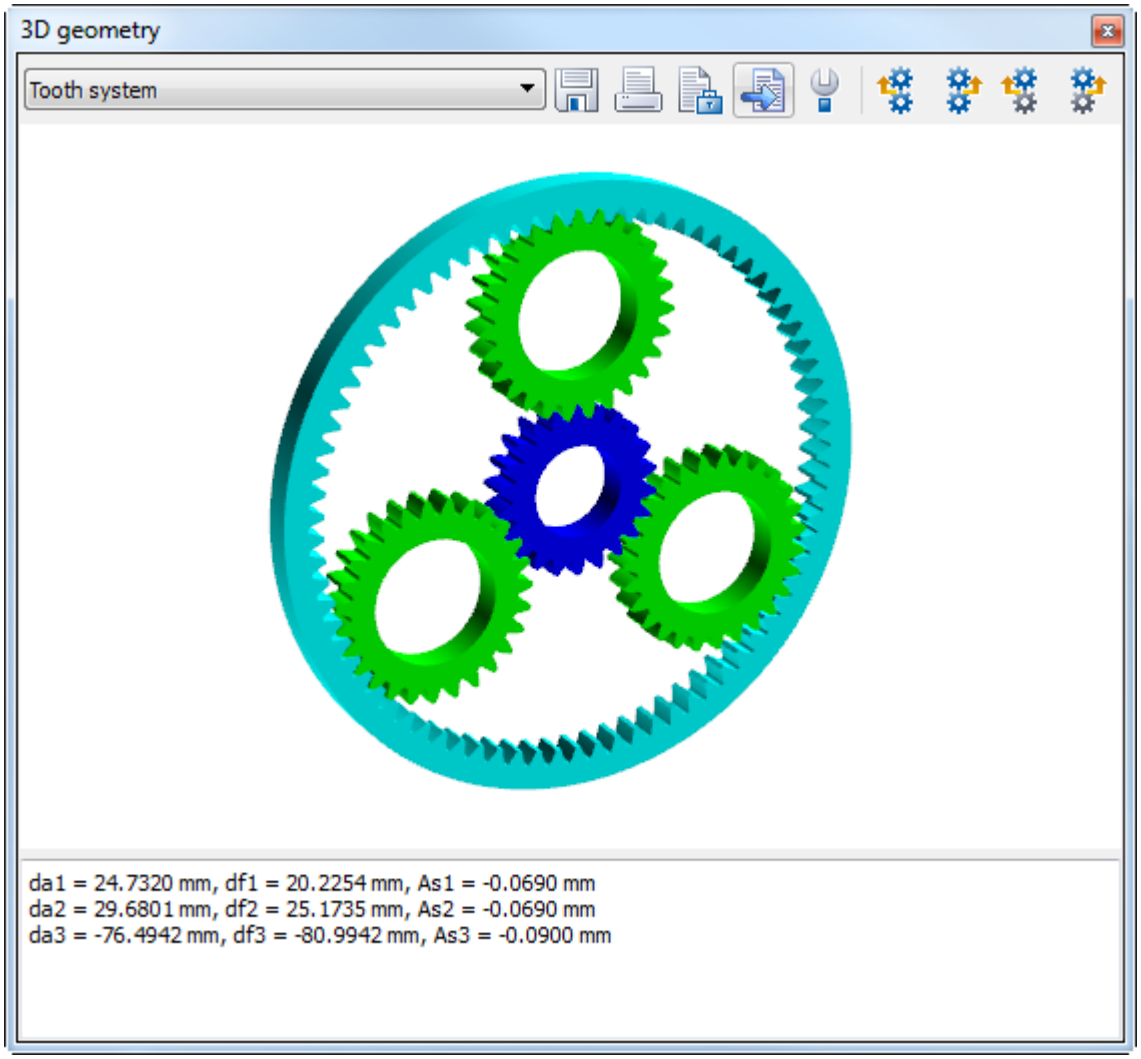


You can therefore create a 3D shaft in the CAD system at the click of a button, using the data from a KISSsoft shaft analysis.

10.4 Viewer with neutral format interface

KISSsoft provides a 3D viewer for displaying individual gears or a gear system. The viewer is activated from the Graphics > 3D Geometry menu.

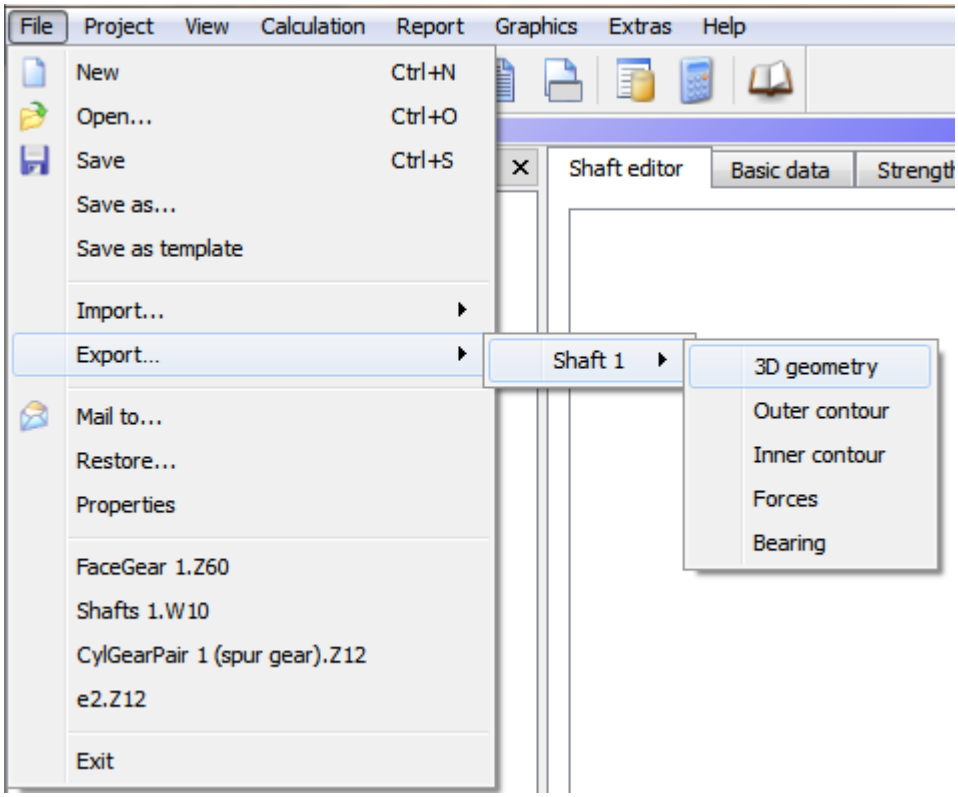
In the 3D viewer, you can export the solid model in STEP and Parasolid formats (text and binary). Supported gears (see page I-174) and for operating the viewer (see page II-584). You can change the settings by selecting Calculation > > Settings > > Parasolid.



10.4.1 Parasolid Export of 3D Shafts

The solid model of the shaft can be generated by using Parasolid. Data can be exported in STEP, Parasolid text (X_T) and binary (X_B) format.

Select `File> Export > Shaft>3D Geometry` to generate the model. If the calculation model contains a number of shafts, you can export these by selecting `File> Export > Geometry 3D System`.



10.4.2 Face gear - 3D geometry

The 3D model of a face gear is generated by simulating the cutting process, and has no limitations affecting the helix angle, shaft angle and offset. The reference coordinates of the model are defined in accordance with Roth [79], and the corresponding positions of pinion and gear are defined by equations (1) and (2).

$$L_S = \overline{O_S Q_S} = \frac{r_{t2} + \left(r_{tS} + x_S \cdot m_n\right) \cdot \cos\theta}{\sin\theta} \tag{1}$$

$$L_2 = \overline{O_0Q_2} = \frac{r_{tS} + x_S \cdot m_n + r_{t2}\cos\theta}{\sin\theta} \qquad (2)$$

Where r_{tS} is the pinion reference radius and x_S is the pinion profile shift coefficient. r_{tS} in the cutting operation is calculated from the pinion cutter.

The shaft angle and radial offset (θ and a) are defined in `Geometry>Details…..`

The face gear model is generated by simulating the cutting process, and the tooth flank is approximated as a spline surface.

The manufacturing process is based on the Parasolid core, where the quality of the model depends on the settings made in Parasolid modeling (see Calculation >Settings > Generation of 3D).

NOTE:

The strength calculation is performed with the assumption that the shaft angle is 90° and the radial offset is 0. The shaft angle and radial offset are only used for 3D model generation, so the strength calculation result may not be valid.

10.4.3 Bevel gear - generating a 3D model

The 3D geometry model for straight-, helical and spiral bevel gears is defined in accordance with ISO 23509 and the tooth form is calculated for several sections along the facewidth. The tooth form is the planar involutes of the virtual cylindrical gear. Then the tooth flank surface is generated by sweeping the tooth forms of the sections (Figure 10.1). The tooth forms in the individual sections are transformed by the angle φ_β into the relevant position. The angle of each section φ_β is calculated both for the generating and face milling processes by using the auxiliary angles φ and η. Therefore, the final tooth form along the facewidth is an extended epicycloid (face hobbing) or circular (generating) form, as shown in Figure 10.2.

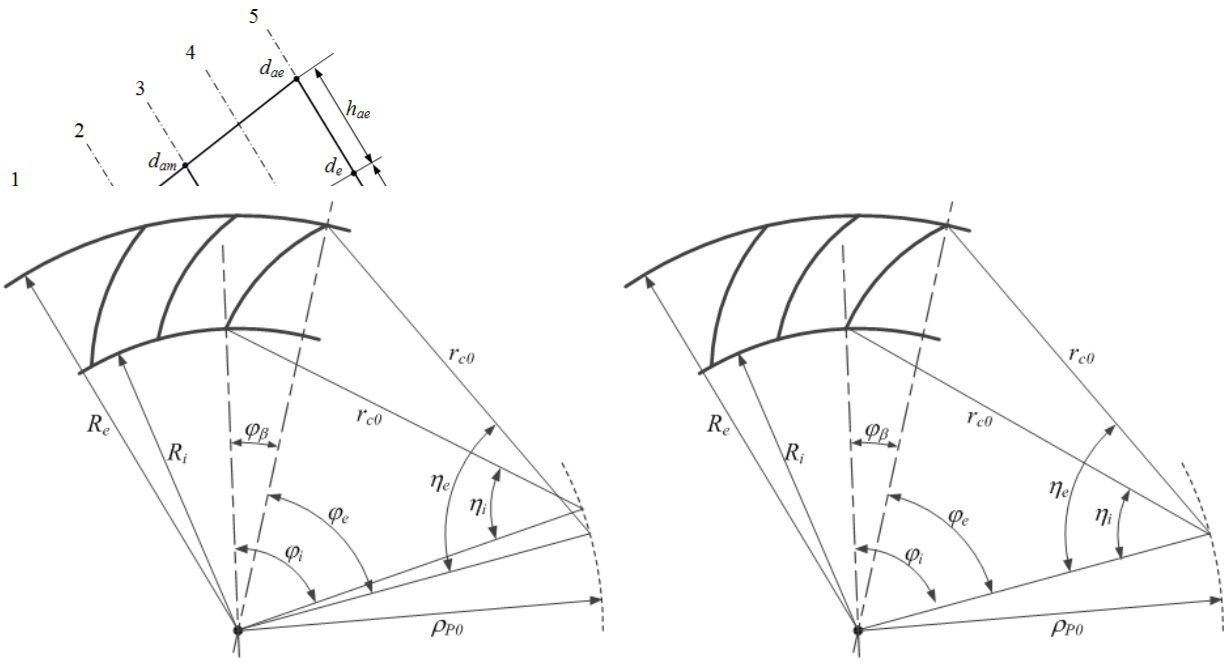


Figure 10.2 Transformation angle of generating (left) and face milling (right) processes

Machine tool manufacturers (such as Klingelnberg and Gleason) also have their own methods for generating tooth forms that differ slightly from the procedures mentioned above. The tooth form is called an octoid and may differ slightly from our tooth form. However, we have ascertained that the difference between the tooth forms is much less than the tolerance range and will not give rise to any problems in practical use.

10.4.4 Worm wheel - generating a 3D model

The 3D model of the enveloping worm wheel is generated by simulating the actual cutting process. The tooth forms at several sections along the facewidth are calculated and the tooth flank is approximated as a spline surface. Theoretically, the hobbing tool is the worm, in regard to arc of circle, pressure angle, and tooth form. However, if the tool itself was manufactured to these specifications, it would no longer be usable after resharpening because it would be smaller than the worm. The hobs used to manufacture worm wheels are therefore slightly larger than the worm they are to create so that they can be resharpened several times, as required [91]. To generate the model using the larger hob, you can set the oversize factor in the module-specific Settings window.

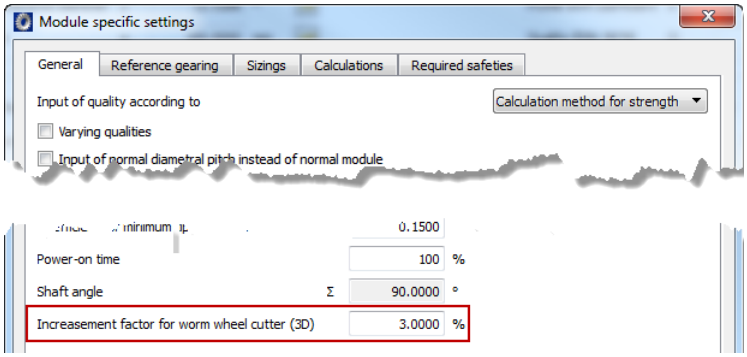


Figure 10.3 Oversize factor for worm wheel cutter

In this case, the hob will have a larger tooth thickness and therefore generate a smaller tooth thickness on the gear. The cutting distance between the hob and the gear will then be changed accordingly, to ensure a consistent result for the root and tip diameter on the gear.

10.5 3D interface to Solid Works

Manufacturer: KISSsoft AG

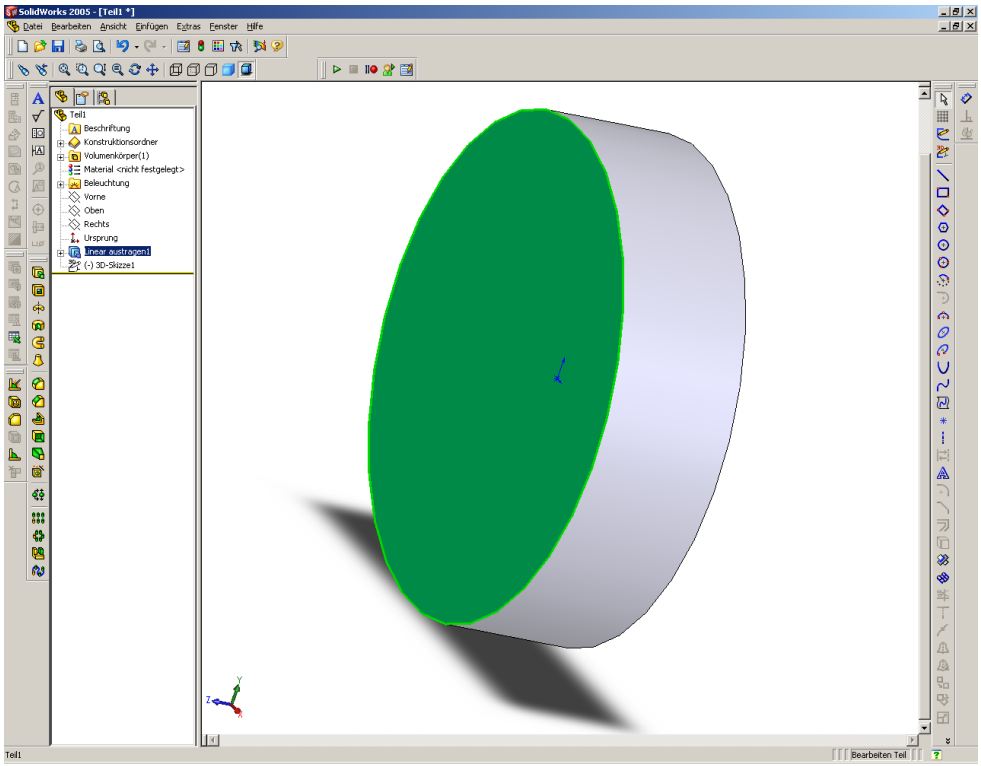
The interface between Solid Edge and KISSsoft creates the direct integration into the 3D CAD system. Use this to start all KISSsoft calculation modules directly from within Solid Works. Face or bevel gears calculated in KISSsoft can be generated directly in Solid Works as a 3D part (see page I-175) with a real tooth form. Shafts calculated with KISSsoft can be generated as a 3D part comprising cylinder and cone elements (see page I-177) directly in Solid Works. From within KISSsoft you can start Solid Works with one click on a button. The system opens a new part and the appropriate part will be generated. You can create cylindrical gears with straight or helical teeth, which are outside or inside, racks with straight or helical teeth, or straight-toothed bevel gears, as defined in DIN 3971, Figure 1, and shafts.
Furthermore, you have the option of adding toothing to existing shafts (see page I-184) at a later point in time. In addition, with the interface in the 2D range, you can automatically insert gear manufacturing data (see page I-189) as a text field. The gear manufacturing data is attached to the relevant cutout (tooth space).

10.5.1 Gear teeth in the case of an existing basic solid

10.5.1.1 Procedure for toothing creation

1. Select the required area in CAD

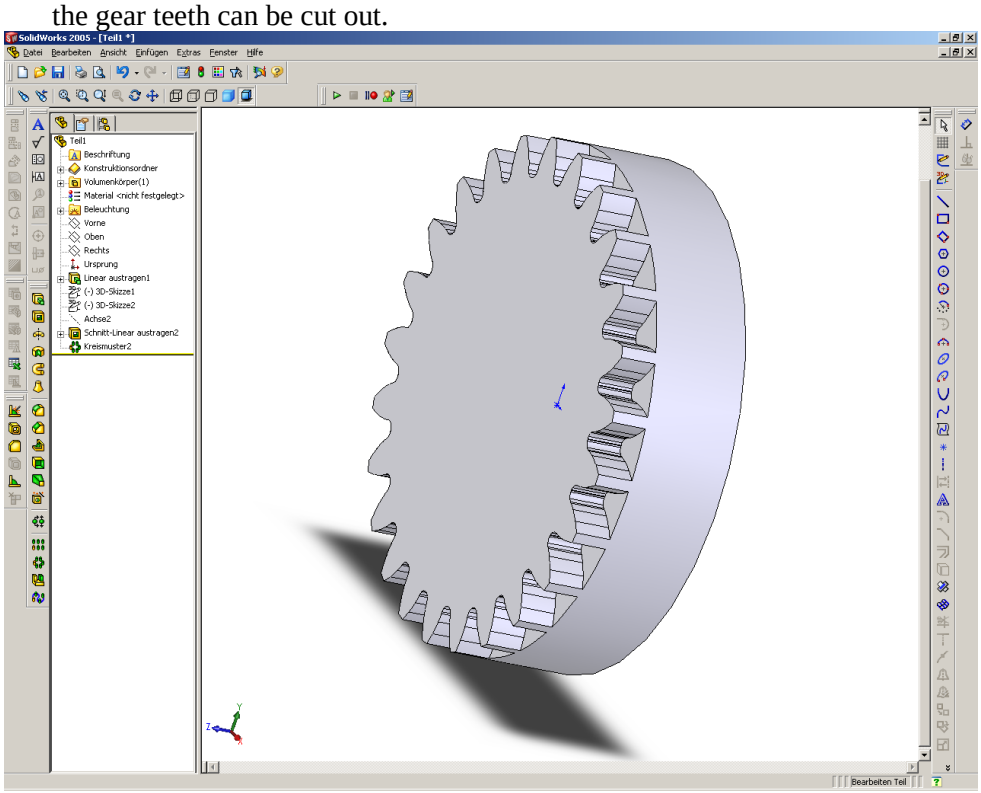
2. In KISSsoft, select which gear (e.g. `Gear 1`) you want to generate on the cylinder.



Prerequisites:

- The diameter of the cylinder must already have the correct external diameter of the toothing before the generation starts.

In the case of internal teeth, a hollow cylinder must already be modeled before the gear teeth can be cut out.



This generation of toothing will be performed for inside and outside cylindrical gears with spur and helical teeth.

10.5.2 Integrating the KISSsoft Add-in (menu items in CAD)

The add-in should be registered as part of the installation process. However, if this did not happen, and you do not see the KISSsoft menu in SolidWorks, you must register the add-in manually.

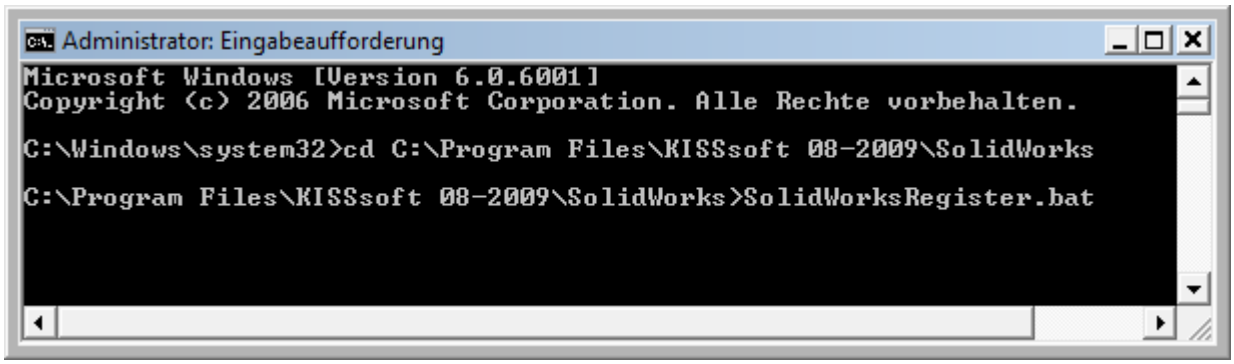
This is how to perform manual registration.

Windows Vista/7/8:

As you must have administrator rights in order to perform the registration, this is only possible here with the command prompt.

1. Start the command prompt as the administrator.
2. Go to the location (`SolidWorks` folder) where the registration file is to be executed. Confirm by pressing `Enter`.

3. Run the registration file. Confirm by pressing `Enter`.

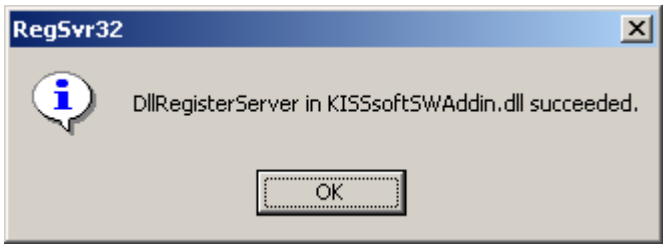


If KISSsoft is running on a server drive, you can enter the following command at the prompt to ensure you can access this drive as an administrator.

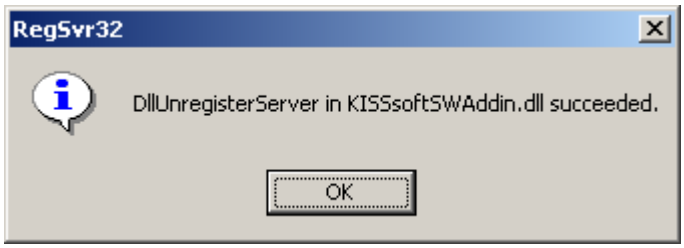
```
Pushd \\SERVER\directory
```

This command assigns a temporary drive letter to the directory. You can then go to where the *.bat file is stored and register the interface.

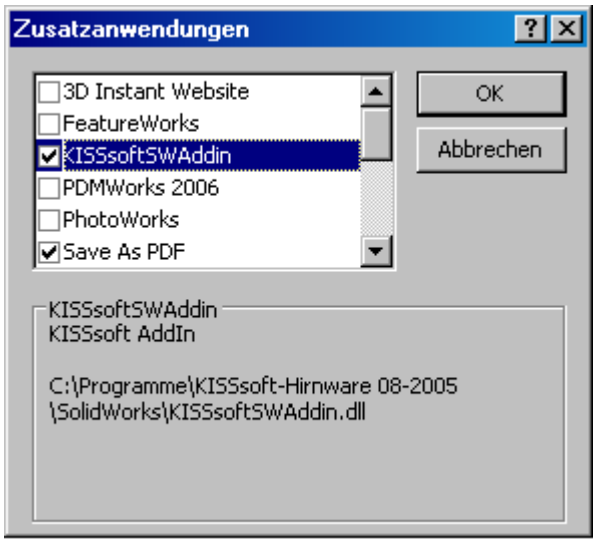
The following message is displayed if the KISSsoft add-in has been successfully registered.



To remove the registration, double-click on the `SolidWorksUnRegister.bat` file in the KISSsoft installation folder. This message is displayed if the process has been performed successfully.



If the add-in is not displayed directly in SolidWorks, select the `Tools > Add-ins` menu to open this window.



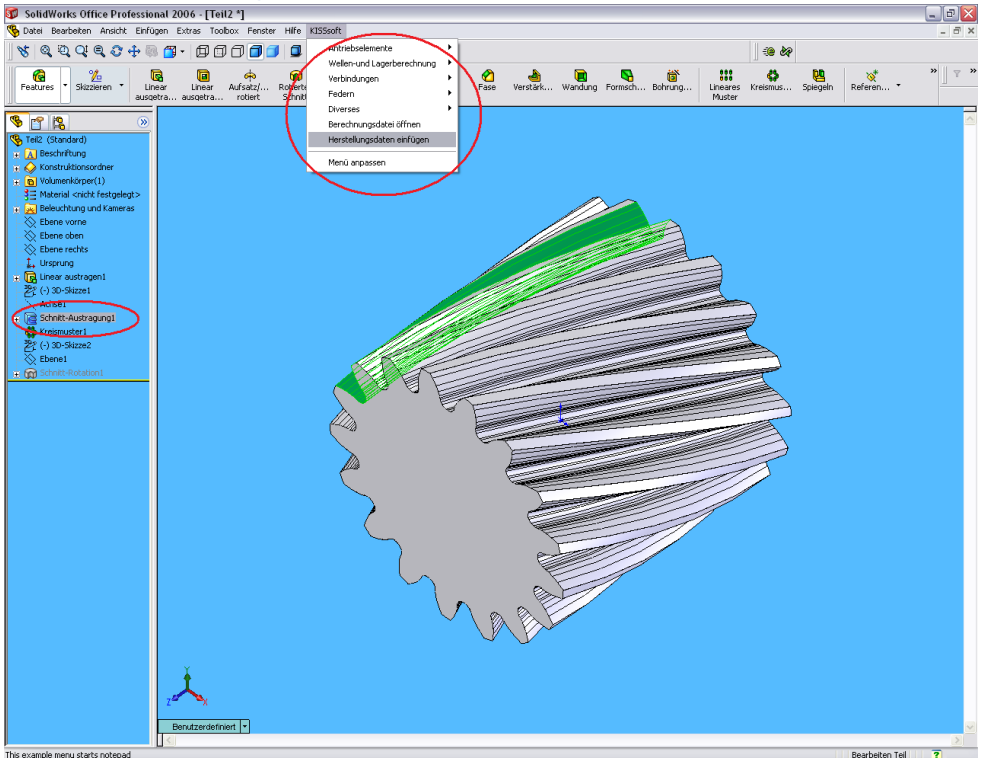
Here, select `KISSsoftSWAddin` and then click `OK` to confirm.

This integrates the KISSsoft menu items in SolidWorks. The menu still remains, even after a restart, and only needs to be linked once.

The menu items in the KISSsoft add-in are available in seven languages (English, French, German, Italian, Spanish, Russian and Portuguese). They use the same language as was selected when KISSsoft was being installed. To set the language, go to the `kiss.ini` file in the KISSsoft installation folder, click on `DISPLAYLANGUAGE`, and set the language you require (0 = german; 1 = english; 2 = french; 3= italian; 4= spanish; 5= russian; 6= portuguese). This language setting now also applies to your KISSsoft system.

10.5.3 Add-in functions (calls)

10.5.3.1 Running KISSsoft via an add-in



Select the `KISSsoft` menu item to open all KISSsoft calculation modules directly. The generation of a new/additional gear will then continue in accordance with the information given about gear generation earlier (see page I-175).

10.5.3.2 Adding manufacturing data

The `Add manufacturing data` menu item only works in the Part view. Procedure for adding a gear stamp on a drawing:

1. Open the part and select the `cutout` of a tooth.
2. Select the `Add manufacturing data` menu item.

This creates a new draft document into which the gear stamp of the selected cutout for the gear teeth will be inserted.

10.5.3.3 Opening the calculation file for the created gear

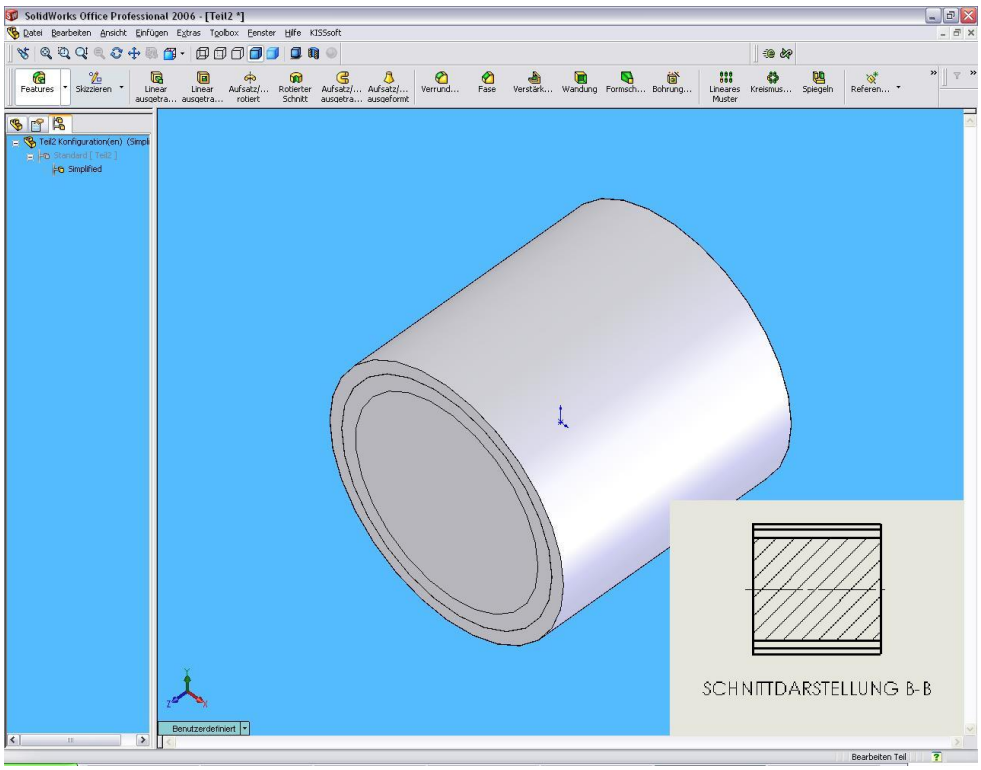
The `Open calculation` file menu item only works in the Part view. Procedure for opening a calculation file:

1. Open the part and select the `cutout` of a tooth.
2. Select the `Open calculation file` menu item.

This starts KISSsoft in each particular calculation module and opens the calculation file.

10.5.3.4 Simplified view of the gears

You have the option to draw the gear in two different views. With the simplified view you can create a section display view of the gear in the drawing extraction in which only the edge contours and the reference circle of the gear are shown. At the moment, the simplified view is only available for external teeth. In the default setting, the simplified view will not be carried out.



To obtain a simplified display, open the `KISS.ini` file in the KISSsoft installation folder and change this entry:

SIMPLIFIEDPRESENTATIONNAME=`Name`

The name given in the `kiss.ini` file is also the name of the view.

10.6 3D interface to Solid Edge

Manufacturer: KISSsoft AG

The interface between Solid Edge and KISSsoft creates the direct integration into the 3D CAD system. This enables you to start all KISSsoft calculation modules directly from Solid Edge. Face or bevel gears calculated in KISSsoft can be generated directly in Solid Edge as a 3D part (see page I-175) with a real tooth form. Shafts calculated with KISSsoft can be generated as a 3D part comprising cylinder and cone elements (see page I-177) directly in Solid Edge. From within KISSsoft, you can start Solid Edge with one click on a button. The system opens a new part and the appropriate part will be generated. You can create cylindrical gears with straight or helical teeth, which are outside or inside, racks with straight or helical teeth, or straight-toothed bevel gears, as defined in DIN 3971, Figure 1, and shafts.
Furthermore, you have the option of adding toothing to existing shafts (see page I-192) at a later point in time. In addition, with the interface in the 2D range, you can automatically insert gear manufacturing data (see section "Adding manufacturing data" on page I-198) in the drawing as a text field. The gear manufacturing data is attached to the relevant cutout (tooth space).

10.6.1 Changes of the parameters for generation

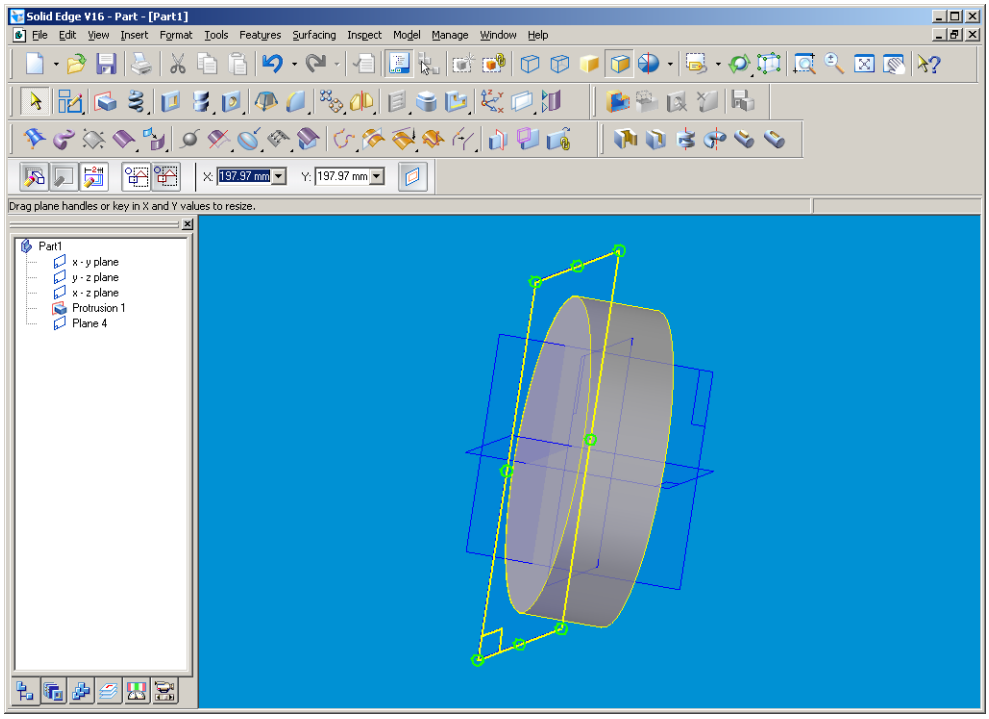
When copying the tooth space (pattern) in Solid Edge, you can switch between two settings. The possible modes are: SmartPattern and FastPattern. In the case of SmartPattern, a more precise generation of the tooth form is carried out, but it takes a long time and the file containing the gear data will be very large. FastPattern uses a less precise method, but this ensures quick construction and a smaller generation file. Until now, SmartPattern has always been used for gear generation, since otherwise the gears cannot be created or represented correctly. In the `kiss.ini` `(see page I-56)` file in the KISSsoft installation folder you can set `SMARTPATTERN=0`, which executes the copying of the tooth space in FastPattern mode.

10.6.2 Gear teeth in the case of an existing basic solid

10.6.2.1 Procedure for toothing creation

1. In Solid Edge, draw a surface in the required area where the gear teeth should be cut out.
2. Select the level

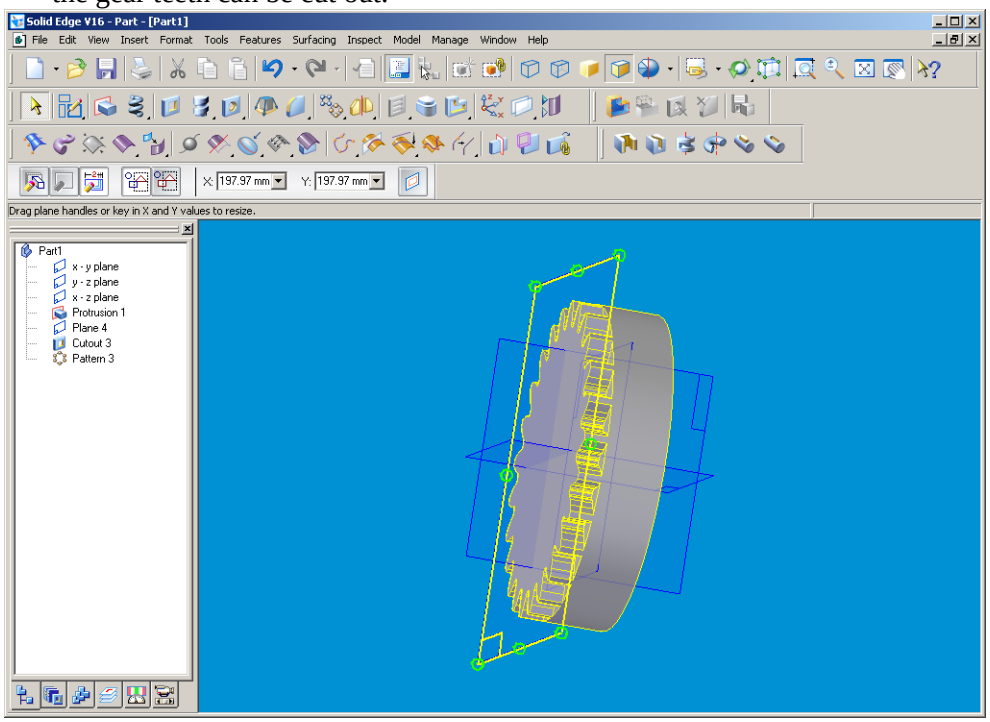
3. In KISSsoft, select which gear (e.g. `Gear 1`) you want to generate on the cylinder.



Prerequisites:

- The diameter of the cylinder must already have the correct external diameter of the gear toothing before the generation process starts.

In the case of internal teeth, a hollow cylinder must already be modeled before the gear teeth can be cut out.



This generation of toothing will be carried out in the case of inside and outside cylindrical gears with spur and with helical teeth.

10.6.3 Integrating the KISSsoft Add-in (menu items in CAD)

The add-in should be registered as part of the installation process. However, if this did not happen, and you do not see the KISSsoft menu in Solid Edge, you must register the add-in manually.

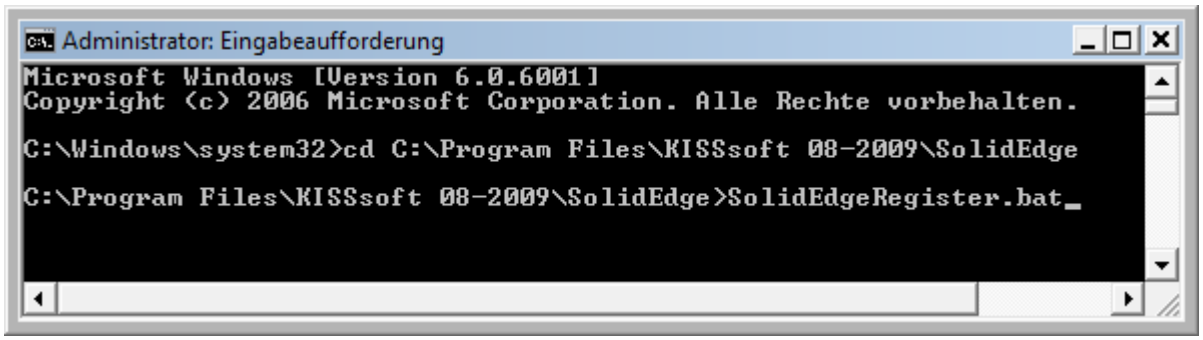
This is how to perform manual registration.

Windows Vista/7/8:

As you must have administrator rights in order to perform the registration, this is only possible here with the command prompt.

1. Start the command prompt as the administrator.
2. Go to the location (`Solid Edge` folder) where the registration file is to be executed. Confirm by pressing `Enter`.

3. Run the registration file. Confirm by pressing `Enter`.



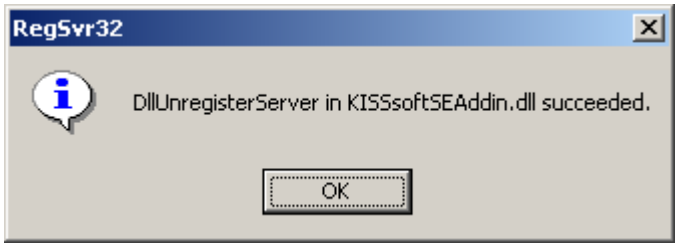
If KISSsoft is running on a server drive, you can enter the following command at the prompt to ensure you can access this drive as an administrator.

```
Pushd \\SERVER\directory
```

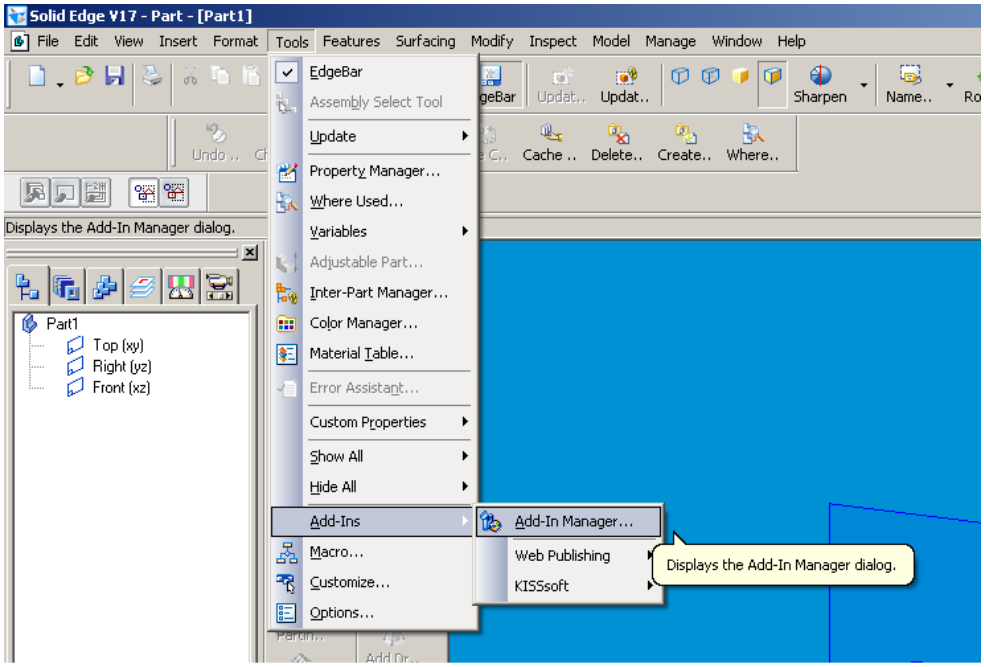
This command assigns a temporary drive letter to the directory. You can then go to where the *.bat file is stored and register the interface.

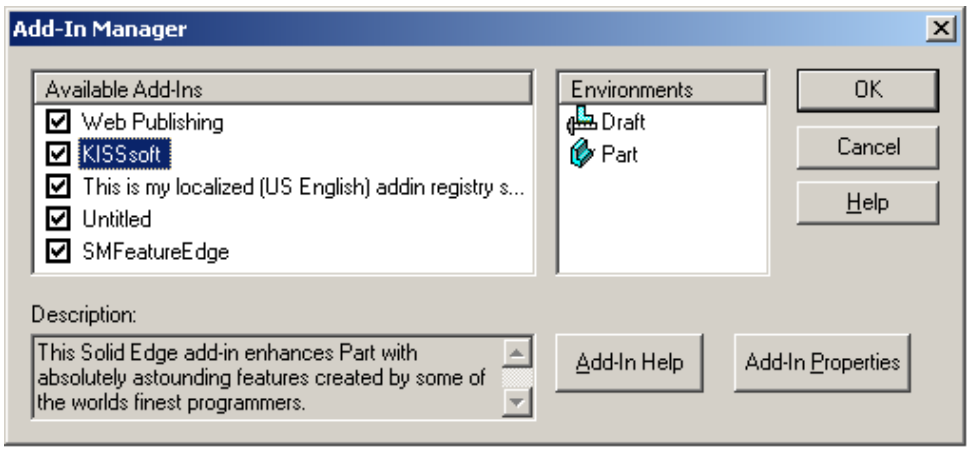
If this message is displayed, the add-in has been successfully registered.

To remove the add-in registration, double-click on the `SolidEdgeUnRegister.bat` file in the KISSsoft installation folder. This message is displayed if the process completes successfully.

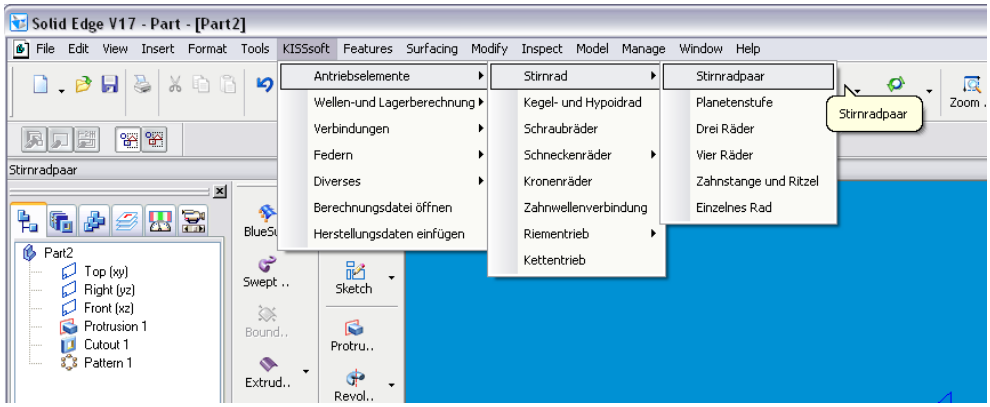


Under `Tools/Add-Ins` you can select `Add-In-Manager` where you can activate/deactivate the add-in.





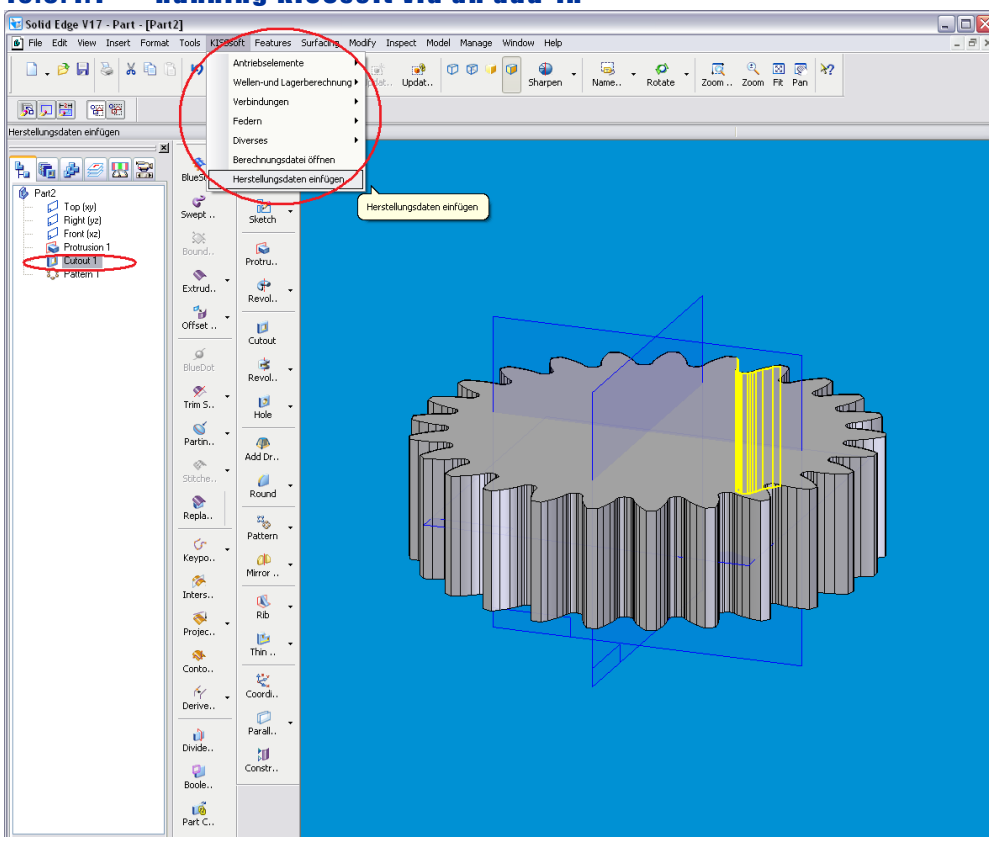
You will see the KISSsoft add-in in the main menu. This integrates the KISSsoft menu items in Solid Edge where they remain even after a restart.



The menu items in the KISSsoft add-in are available in seven languages (English, French, German, Italian, Spanish, Russian and Portuguese). They use the same language as was selected when KISSsoft was being installed. To set the language, go to the `kiss.ini` file in the KISSsoft installation folder, click on `DISPLAYLANGUAGE`, and set the language you require (0 = german; 1 = english; 2 = french; 3= italian; 4= spanish; 5= russian; 6= portuguese). This language setting now also applies to your KISSsoft system.

10.6.4 Add-in functions (calls)

10.6.4.1 Running KISSsoft via an add-in



Select the `KISSsoft` menu item to open all KISSsoft calculation modules directly. The generation of a new/additional gear will then continue in accordance with the information previously given about gear generation (see page I-175).

10.6.4.2 Adding manufacturing data

The `Add manufacturing data` menu item only works in the Part view. Procedure for adding a gear stamp on a drawing:

1. Open the part and select the `Cutout` of a tooth.
2. Select the `Add manufacturing data` menu item.

This creates a new draft document into which the gear stamp of the selected cutout for the gear teeth will be inserted.

10.6.5 Opening the calculation file for the created gear

The `Open calculation` file menu item only works in the Part view. Procedure for opening a calculation file:

1. Open the part and select the `Cutout` of a tooth.
2. Select the `Open calculation file` menu item.

This starts KISSsoft in each particular calculation module and opens the calculation file.

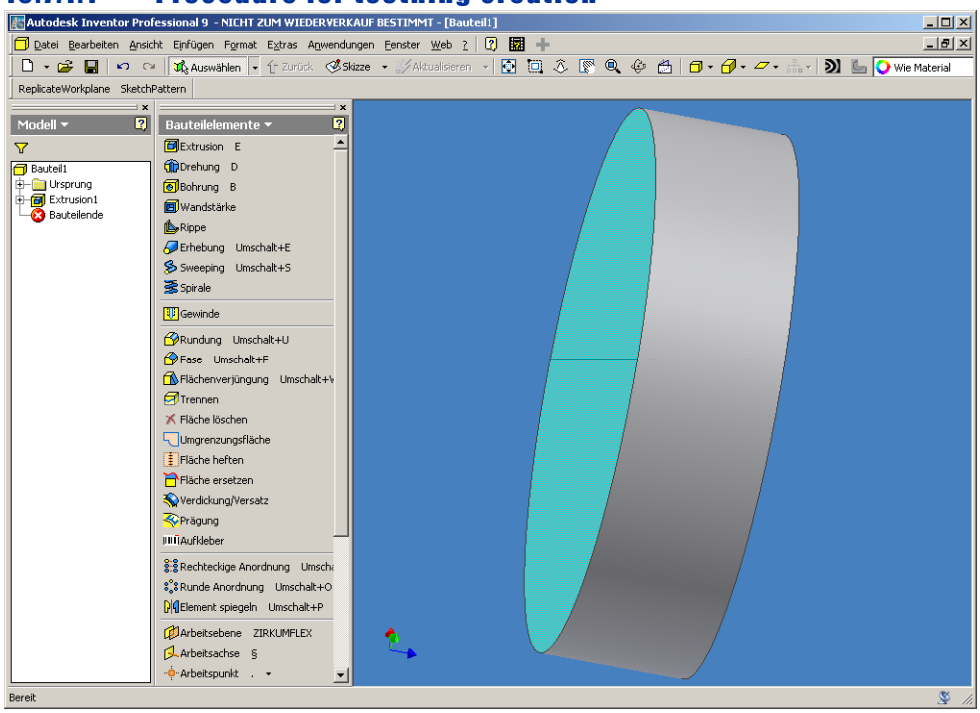
10.7 3D interface to Autodesk Inventor

Manufacturer: KISSsoft AG

The interface between Inventor and KISSsoft is achieved by direct integration in the 3D CAD system. Use this to start all KISSsoft calculation modules directly from within Inventor. Face or bevel gears calculated in KISSsoft can be generated directly in Inventor as a 3D part (see page I-175) with a real tooth form. Shafts calculated with KISSsoft can be generated as a 3D part comprising cylinder and cone elements (see page I-177) directly in Inventor. From within KISSsoft, you can start Inventor with one click on a button. The system opens a new part and the appropriate part will be generated. You can create cylindrical gears with straight or helical teeth, which are outside or inside, racks with straight or helical teeth, or straight-toothed bevel gears, as defined in DIN 3971, Figure 1, and shafts. Furthermore, you have the option of adding toothing to existing shafts (see page I-200) at a later point in time. In addition, with the interface in the 2D range, you can automatically insert gear manufacturing data (see section "Adding manufacturing data" on page I-204) as a table on the drawing. The gear manufacturing data is attached to the relevant cutout (tooth space).

10.7.1 Gear teeth in the case of existing shaft data

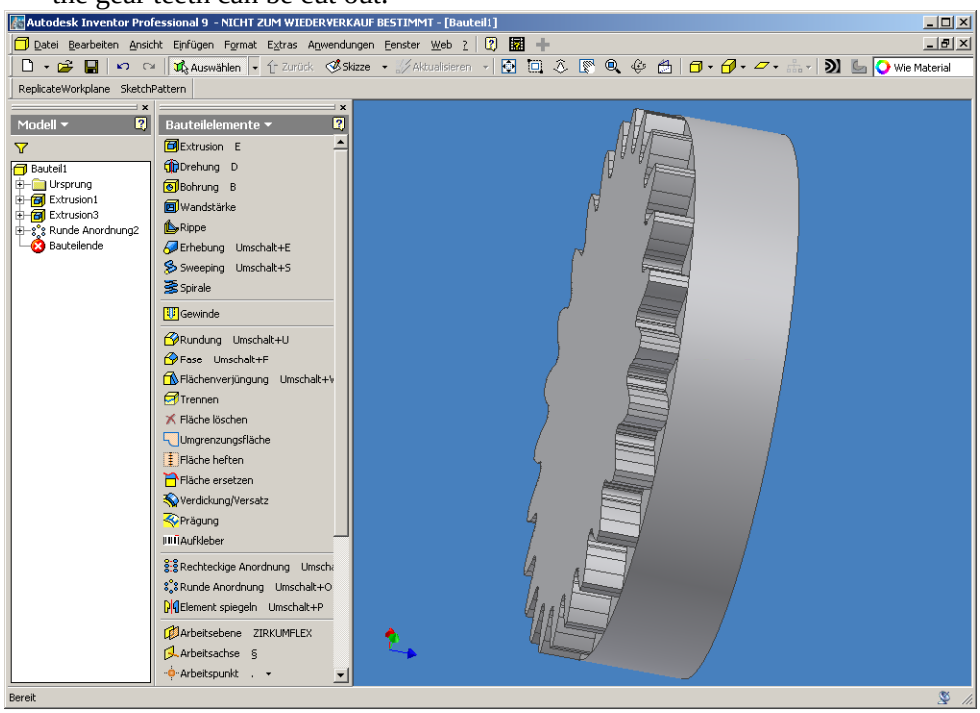
10.7.1.1 Procedure for toothing creation



1. Select the required area
2. In KISSsoft, select which gear (e.g. `Gear 1`) you want to generate on the cylinder.

Prerequisites:

- The diameter of the cylinder must already have the correct external diameter of the gear toothing before the generation process starts.
- In the case of internal teeth, a hollow cylinder must already be modeled before the gear teeth can be cut out.



This generation of toothing will be performed for inside and outside cylindrical gears with spur and helical teeth.

10.7.2 Add-in (menu items in CAD)

10.7.2.1 Integrating the KISSsoft Add-in

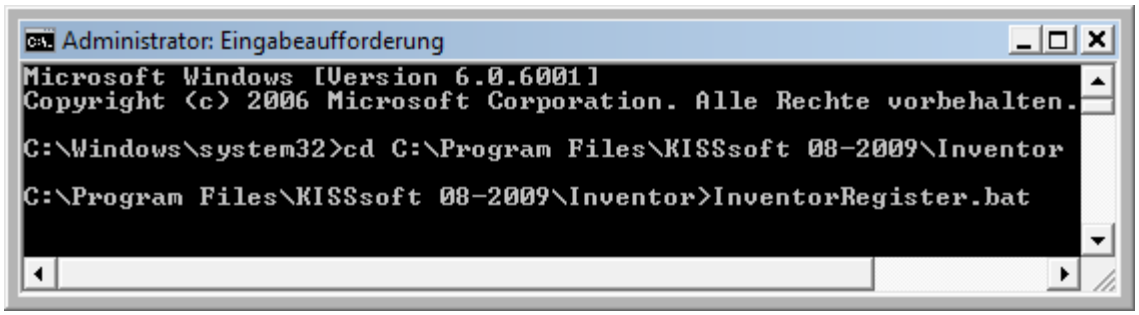
The add-in should be registered as part of the installation process. However, if this did not happen, and you do not see the KISSsoft menu in Inventor, you must register the add-in manually.

This is how to perform manual registration.

Windows Vista/7/8:

As you must have administrator rights in order to perform the registration, this is only possible here with the command prompt.

1. Start the command prompt as the administrator.
2. Go to the location (`Inventor` folder) where the registration file is to be executed. Confirm by pressing `Enter`.
3. Run the registration file. Confirm by pressing `Enter`.

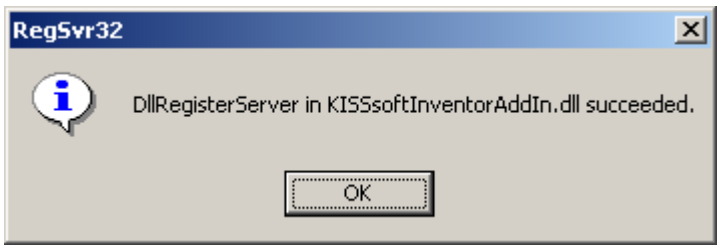


If KISSsoft is running on a server drive, you can enter the following command at the prompt to ensure you can access this drive as an administrator.

```
Pushd \\SERVER\directory
```

This command assigns a temporary drive letter to the directory. You can then go to where the *.bat file is stored and register the interface.

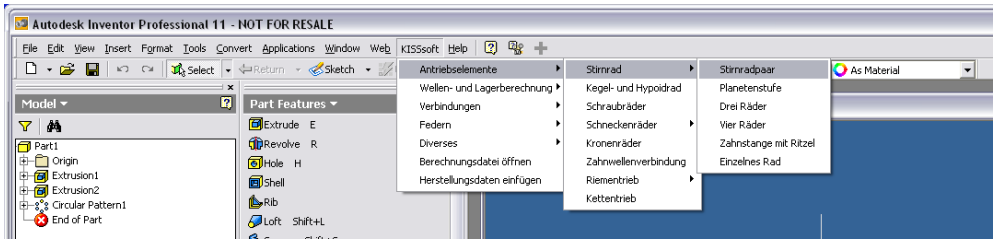
If this message is displayed, the add-in has been successfully registered.



If you no longer want the Inventor add-in to be registered, double-click on the `InventorUnRegister.bat` file in the KISSsoft installation folder. If this message is displayed, the add-in has been successfully registered.

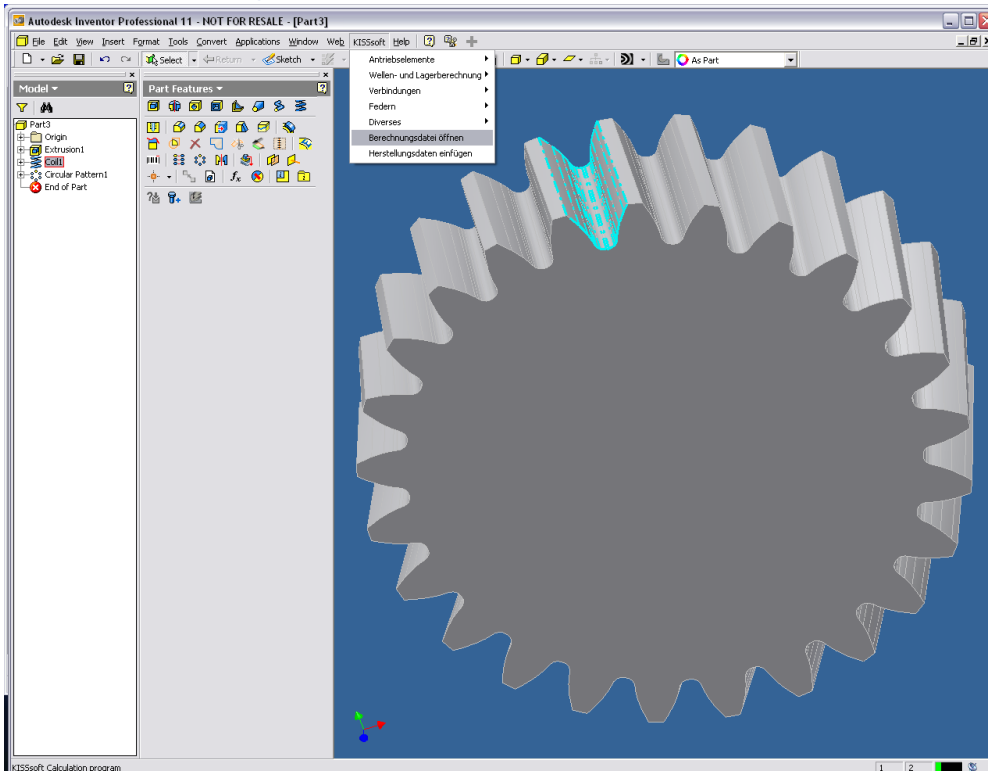
The menu items in the KISSsoft add-in are available in seven languages (English, French, German, Italian, Spanish, Russian and Portuguese). They use the same language as was selected when KISSsoft was being installed. To set the language, go to the `kiss.ini` file in the KISSsoft installation folder, click on `DISPLAYLANGUAGE`, and set the language you require (0 = german; 1 = english; 2 = french; 3= italian; 4= spanish; 5= russian; 6= portuguese). This language setting now also applies to your KISSsoft system.

This integrates the KISSsoft menu items in Inventor. The menu still remains, even after a restart, and does not need to be linked.



10.7.3 Add-in functions (calls)

10.7.3.1 Running KISSsoft via an add-in



Select the `KISSsoft` menu item to open all KISSsoft calculation modules directly. The generation of a new/additional gear will then continue in accordance with the information previously given about gear generation (see page I-175).

10.7.3.2 Adding manufacturing data

The `Add manufacturing data` menu item only works in the Part view. Procedure for adding a gear stamp on a drawing:

1. Open the part and select the `Cutout` of a tooth.
2. Select the `Add manufacturing data` menu item.

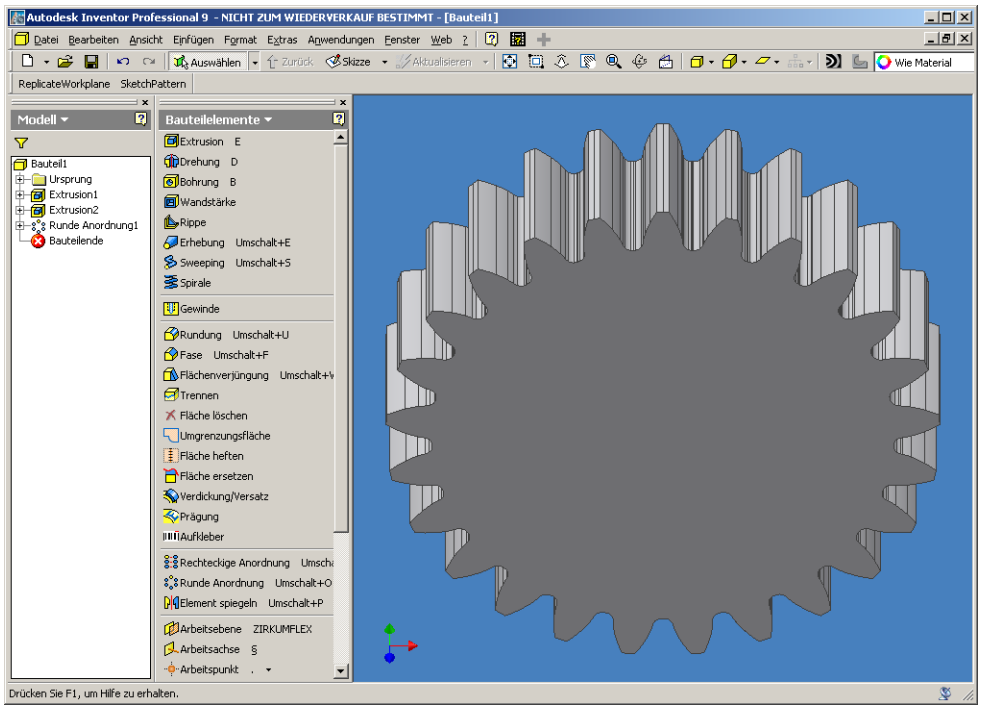
This creates a new draft document into which the gear stamp of the selected cutout for the gear teeth will be inserted.

10.7.4 Opening the calculation file for the created gear

The `Open calculation` file menu item only works in the Part view. Procedure for opening a calculation file:

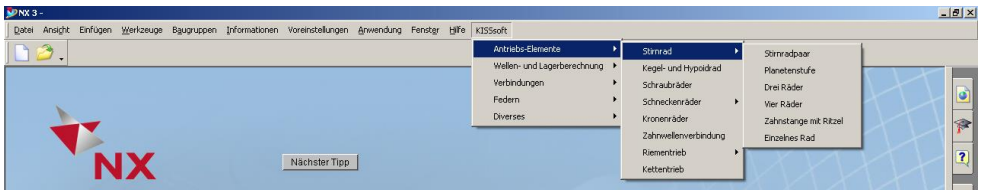
1. Open the part and select the `Cutout` of a tooth.
2. Select the `Open calculation file` menu item.

This starts KISSsoft in each particular calculation module and opens the calculation file.

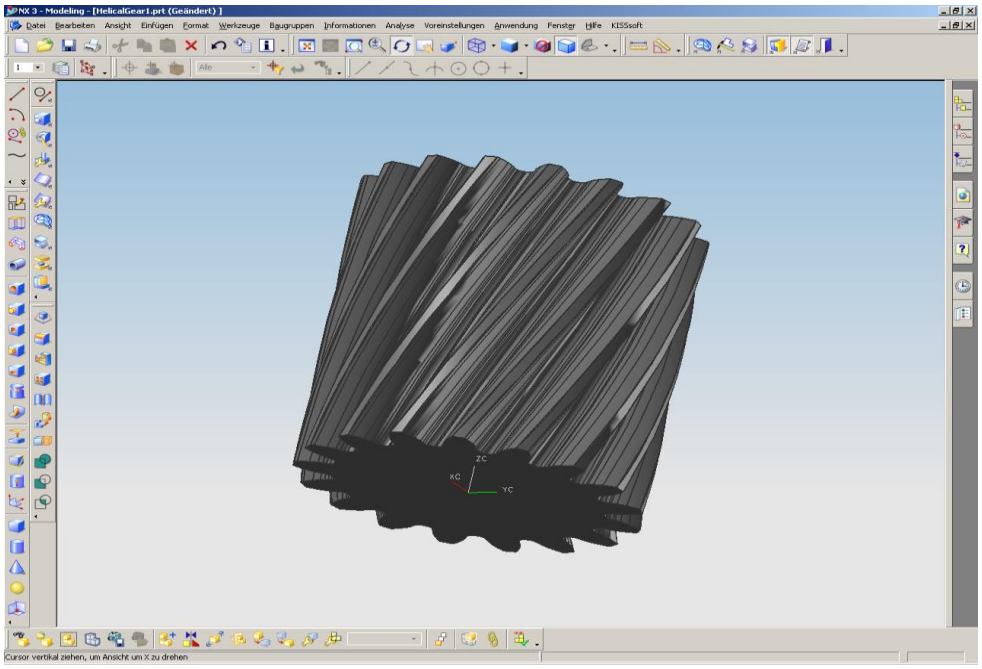


10.8 3D interface to Unigraphics NX

Manufacturer: KISSsoft AG



The interface between NX and KISSsoft is achieved by direct integration in the 3D CAD system. Use this to start all KISSsoft calculation modules directly from within NX. Face or bevel gears calculated in KISSsoft can then be generated directly in NX as a 3D part (see page I-175) with a real tooth form. Shafts calculated with KISSsoft can be generated as a 3D part comprising cylinder and cone elements (see page I-177) directly in NX. You can create cylindrical gears with straight or helical teeth, which are outside or inside, worms, or straight-toothed bevel gears, as defined in DIN 3971, Figure 1, and shafts.
If you create a new part, the `New` dialog opens first. In it you can enter the name of the file in which the part should be generated. When you use Teamcenter, its dialog is displayed automatically so you can also generate or save the part in the Teamcenter environment. Furthermore, you have the option of adding toothing to existing shafts (see section "Gear teeth when existing shaft data is present" on page I-210) at a later point in time. In addition, with the interface in the 2D range, you can automatically insert gear manufacturing data as a table on the drawing. The gear manufacturing data is attached to the relevant cutout (tooth space).



10.8.1 Add-in (menu items in CAD)

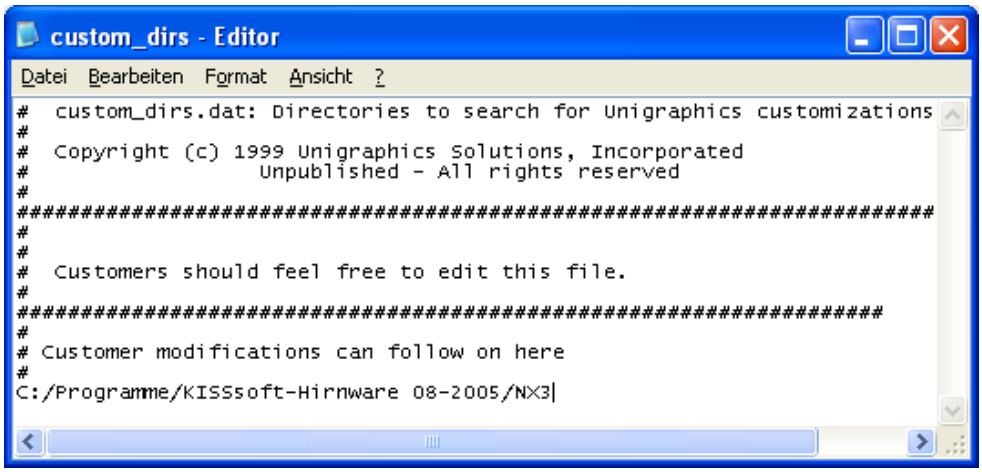
10.8.1.1 Integrating the KISSsoft Add-in

Firstly, copy the supplied folder, e.g. `NX8`, with its `startup` sub-folder, to a location that can be accessed by the user at any time.

The definitions of the KISSsoft AddIns menu items are located in the "kSoftNX_d.men" file. This file has different name endings to reflect which language has been selected. For example, the `_e` in this file name represents `English`. Also used: _d: for `German`; _f: for `French`; _i: for `Italian`; _s: for `Spanish`; _r (in development): for `Russian`; _p: for `Portuguese;` The file in the required language can be copied to the `startup` folder to ensure the KISSsoft menu is displayed in the selected language.

The `kSoftNX8.dll` file (for example), which contains the links and commands for the menu items, can also be stored in this folder.

You must enter the path of the previously copied folder, for example, `NX8`, in the "UGII\menus\custom_dirs.dat" file in the UGS directory so that the UGS system can tell where the files it is to use are stored.



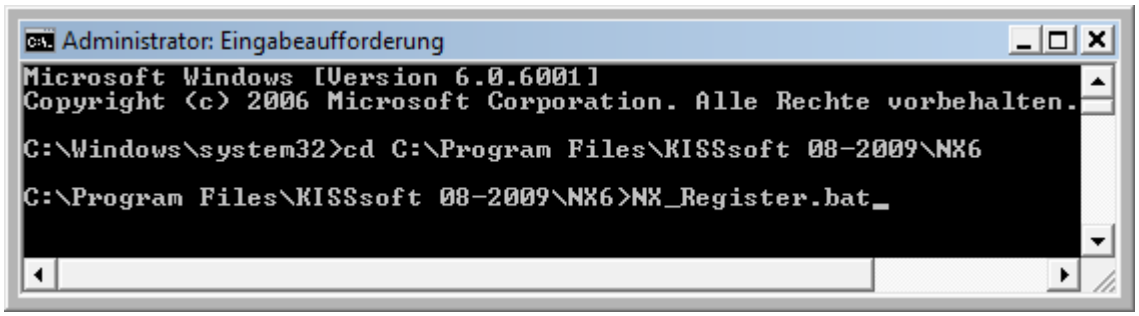
The KISSsoftCOM server should be registered as part of the installation process. However, if this did not succeed, and the KISSsoft interface does not work, you must register KISSsoftCOM manually.

This is how to perform manual registration.

Windows Vista/7/8:

As you must have administrator rights in order to perform the registration, this is only possible here with the command prompt.

1. Start the command prompt as the administrator.
2. Go to the location (e.g. `NX6` folder) where the registration file is to be executed. Confirm by pressing `Enter`.
3. Run the registration file. Confirm by pressing `Enter`.

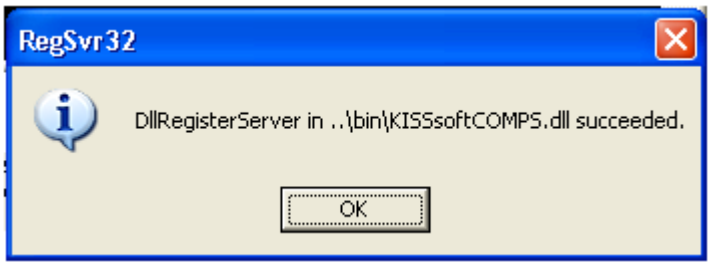


If KISSsoft is running on a server drive, you can enter the following command at the prompt to ensure you can access this drive as an administrator.

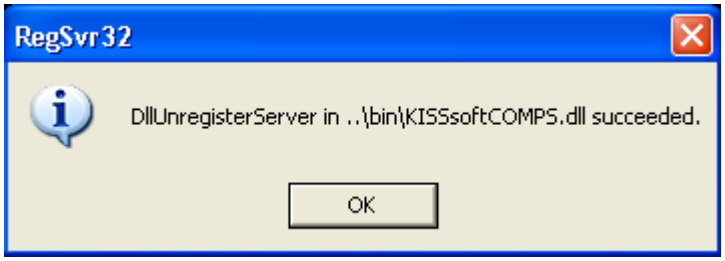
```
Pushd \\SERVER\directory
```

This command assigns a temporary drive letter to the directory. You can then go to where the *.bat file is stored and register the interface.

The following message is displayed if the KISSsoft add-in has been successfully registered.



To remove the registration, double-click on the `NXUnRegister.bat` file in the KISSsoft installation folder. This message is displayed if the process has been performed successfully.



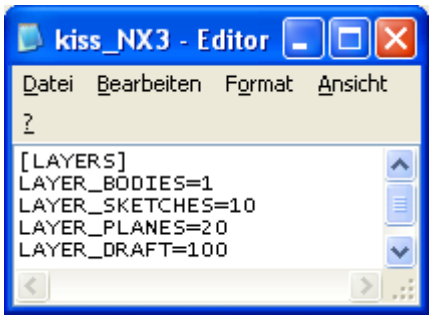
To ensure the KISSsoft icons are displayed next to the menu items, you must also set a system variable with the path, to tell the program where the KISSsoft icons can be found.

Set system variable and use it as a value for the path, for example:

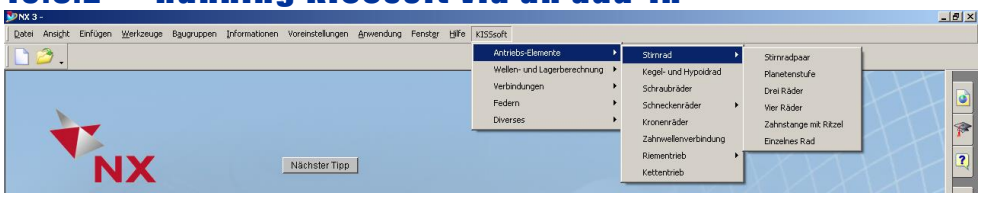
KSOFT_ICONS

C:\Program Files(x86)\KISSsoft 03-2014\bin32\icons

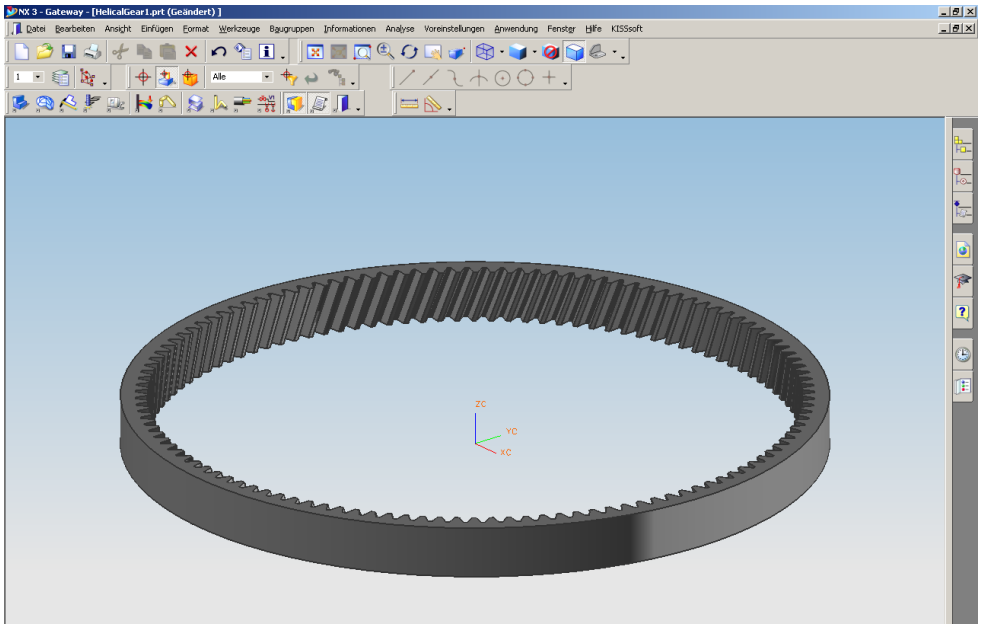
The `startup` folder also contains a `kSoftNX.ini` file in which a part's layers, sketches, planes, and drafts, can be changed.



10.8.2 Running KISSsoft via an add-in



Select the `KISSsoft` menu item to open all KISSsoft calculation modules directly. By doing this you can perform NX8 calculations (for example) in KISSsoft quickly and easily during the design process. The NX8 (etc.) menu items are inactive whilst KISSsoft is open. In order to reactivate the CAD program, you must close KISSsoft.



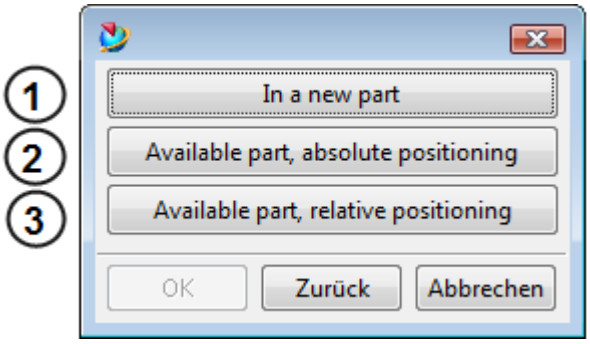
10.8.2.1 Gear teeth when existing shaft data is present

Prerequisites:

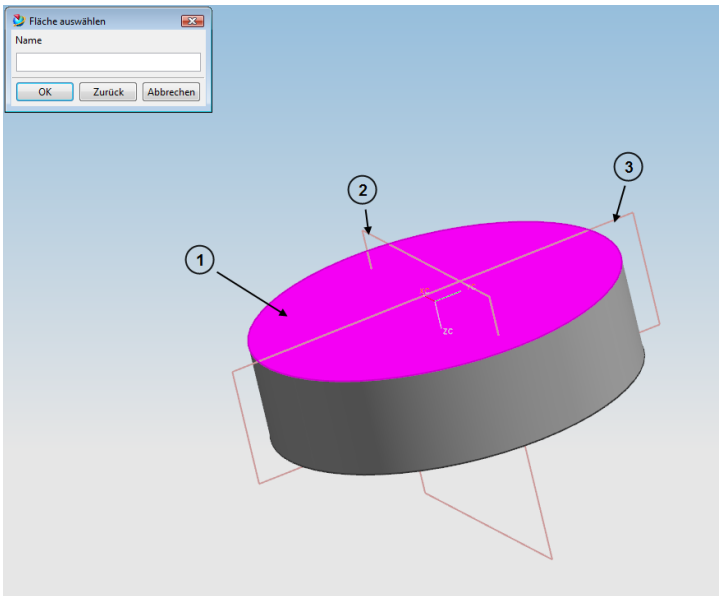
- The cylinder diameter must already have the same external diameter as the gear toothing before the generation process starts.
- In the case of internal teeth, a hollow cylinder must already be modeled before the gear teeth can be cut out.

For example, go to the KISSsoft menu in NX8 and select the cylindrical gear pair calculation. The procedure for generating the gear (see page I-175) is identical to the procedure for creating a new one.

If a part is already opened in NX (whatever version) when you return, the following list is displayed:



1. A new part opens, and the complete gear is generated.
2. If you select `Available part, absolute positioning`, only one side surface needs to be selected, on which the gear teeth are to be cut. For the generation, fixed levels will be generated, on which the gear teeth will be positioned.
3. If you select `Available part, relative positioning`, you can select a side surface and two levels (which will intersect the side surface). Consequently, the toothing can be positioned at relative planes (DATUM PLANE) and is not dependent on the absolute zero point. This positioning is mainly required in the case of the methodical operational behavior defined in the Master Model concept (Teamcenter).



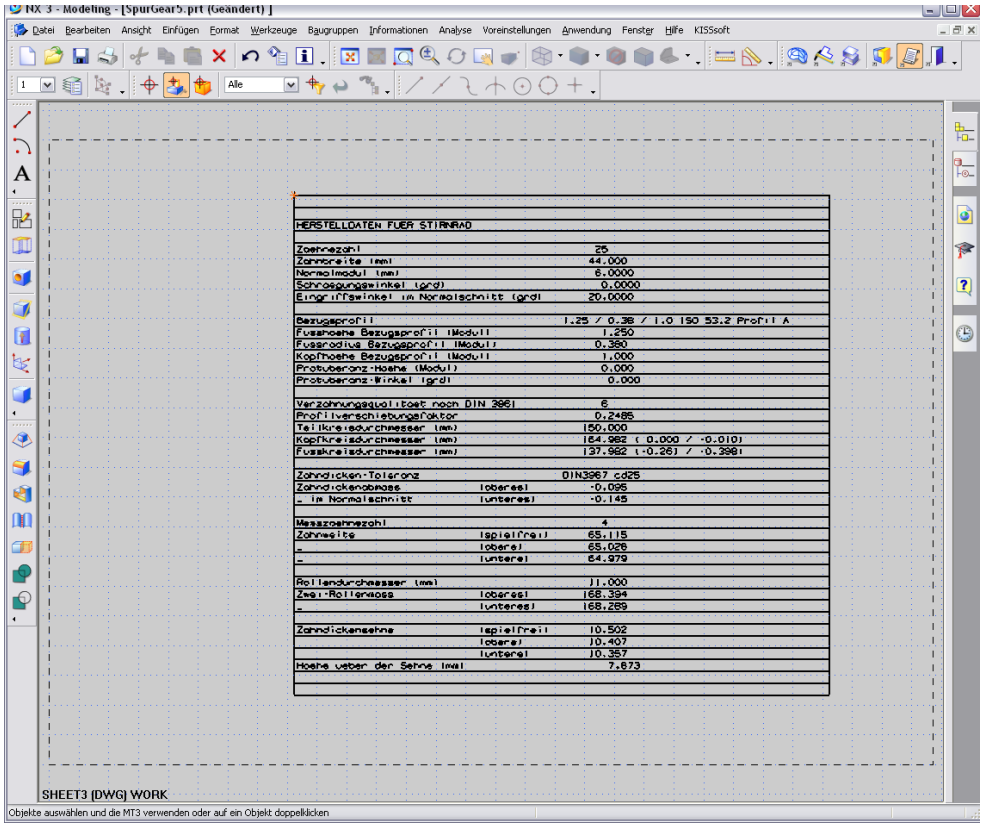
The generation of toothing on existing cylinders is performed on both inside and outside cylindrical gears with straight and sloping teeth.

10.8.2.2 Adding manufacturing data on the drawing

You can use the `Add manufacturing data` menu item to insert a gear stamp of the current gear in a drawing.

- Teamcenter: If you are working in accordance with the `Master Model concept`, the features of the master part are displayed automatically in the non-master drawing when you call up `Add manufacturing data`.

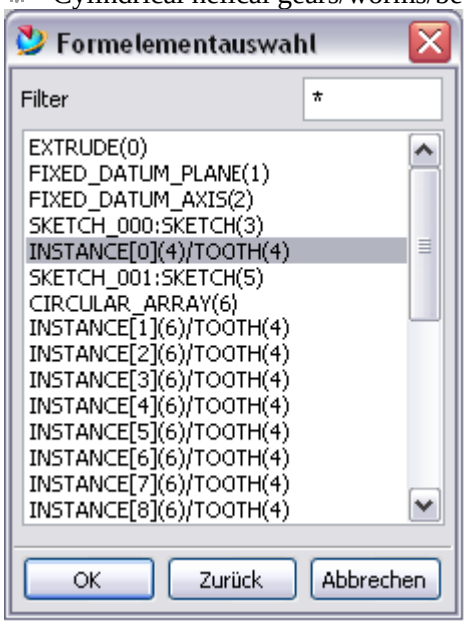
After you select this menu item, the following screen appears:



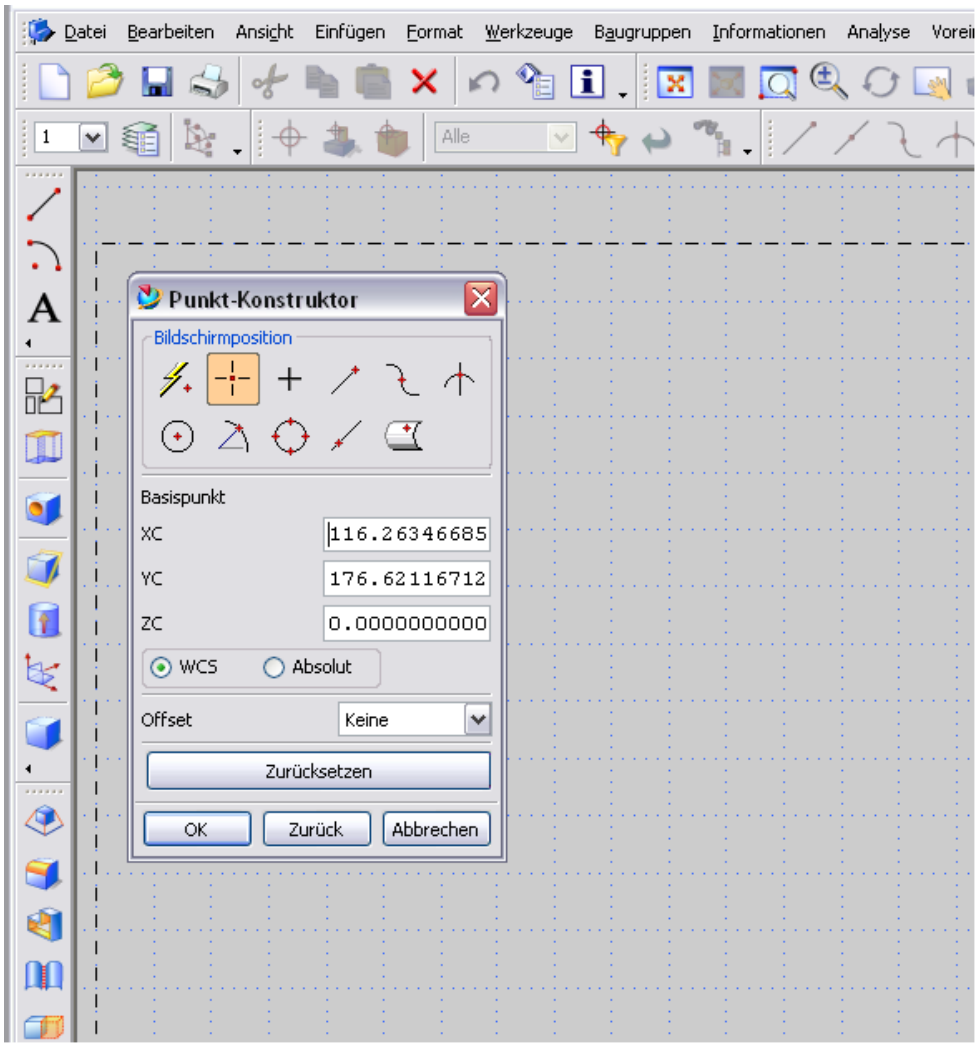
In this screen, select the following:

- Straight-toothed cylindrical gears: INSTANCE[0](4)TOOTH(4)

Cylindrical helical gears/worms/bevel spur gears: TOOTH

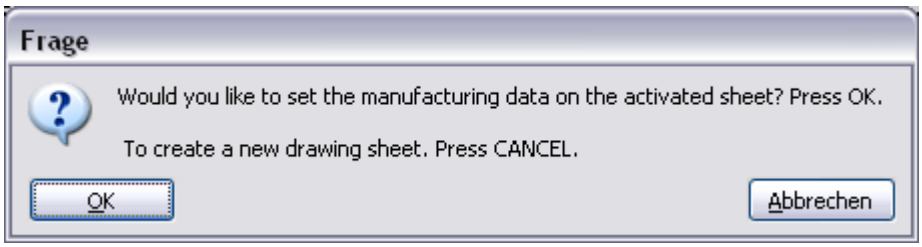


If you press OK, a new drawing will open and the following window appears:

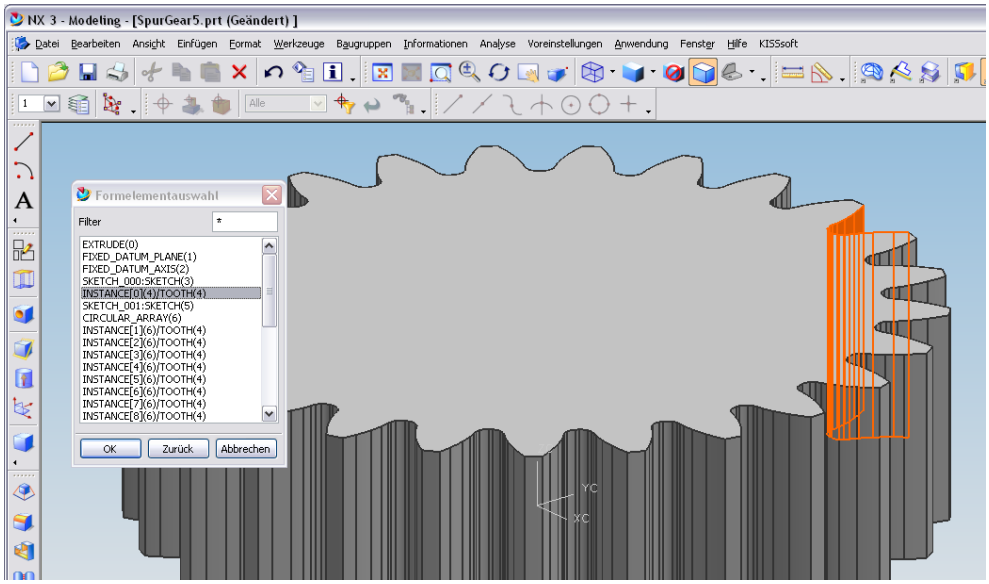


With one mouse click you can position the manufacturing data on the drawing. The mouse click will position the upper left corner of the table.

If you want to insert the data into an already existing drawing sheet, you have to select the tooth space in the Drawing view if the required drawing sheet is opened. You will then see the screen in which you can select the tooth space, and are then prompted to decide if it should be inserted into the current drawing sheet.

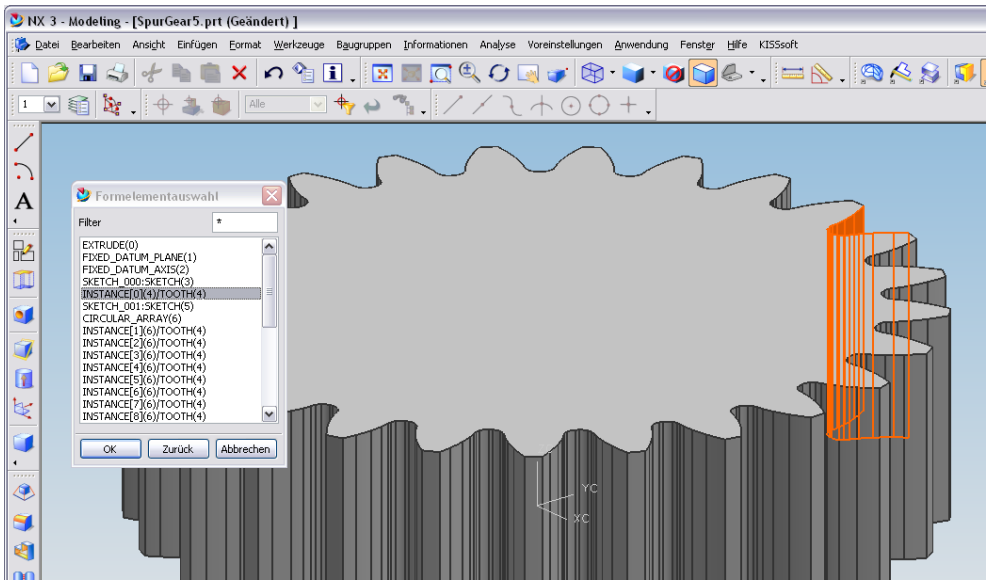


If you click `OK` you can use the mouse to position the current manufacturing data on the drawing. Click `CANCEL` to open a new drawing sheet in which you then can insert the manufacturing data.



10.8.2.3 Opening the calculation file

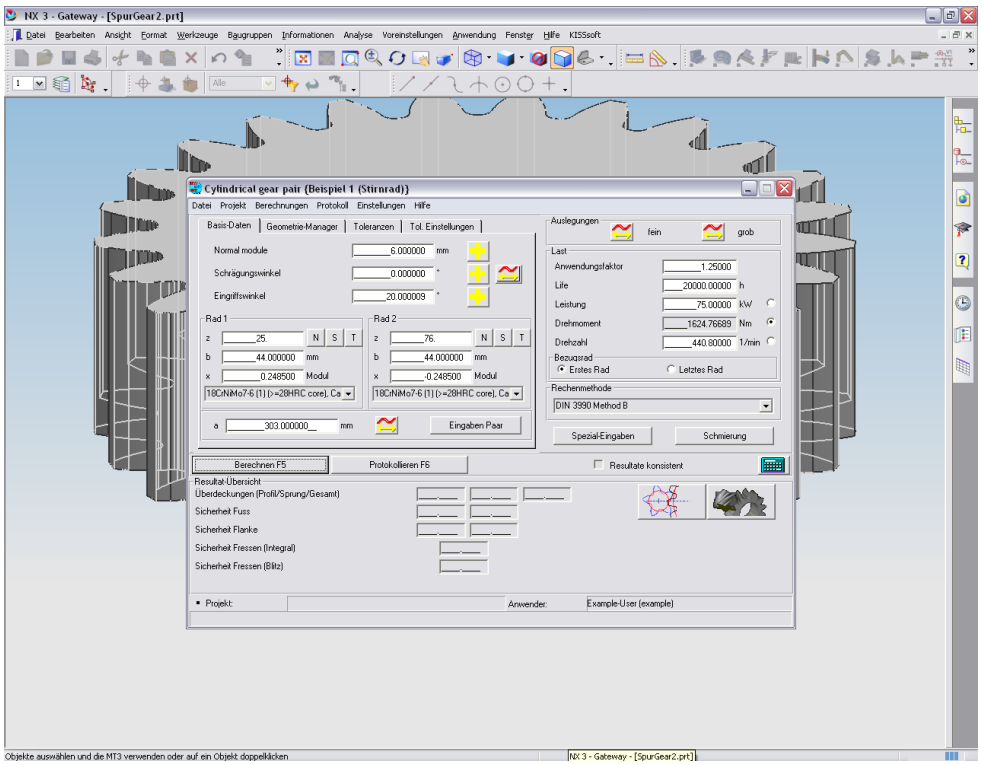
Select the `Calculation file` menu item to start KISSsoft and load calculation file for the toothing whose information is saved directly on the toothing Feature (tooth space). After you select this menu item, the following screen appears:



In this screen, select the following:

- Straight-toothed cylindrical gears: INSTANCE[0](4)TOOTH(4)
- Cylindrical helical gears/worms/bevel spur gears: TOOTH

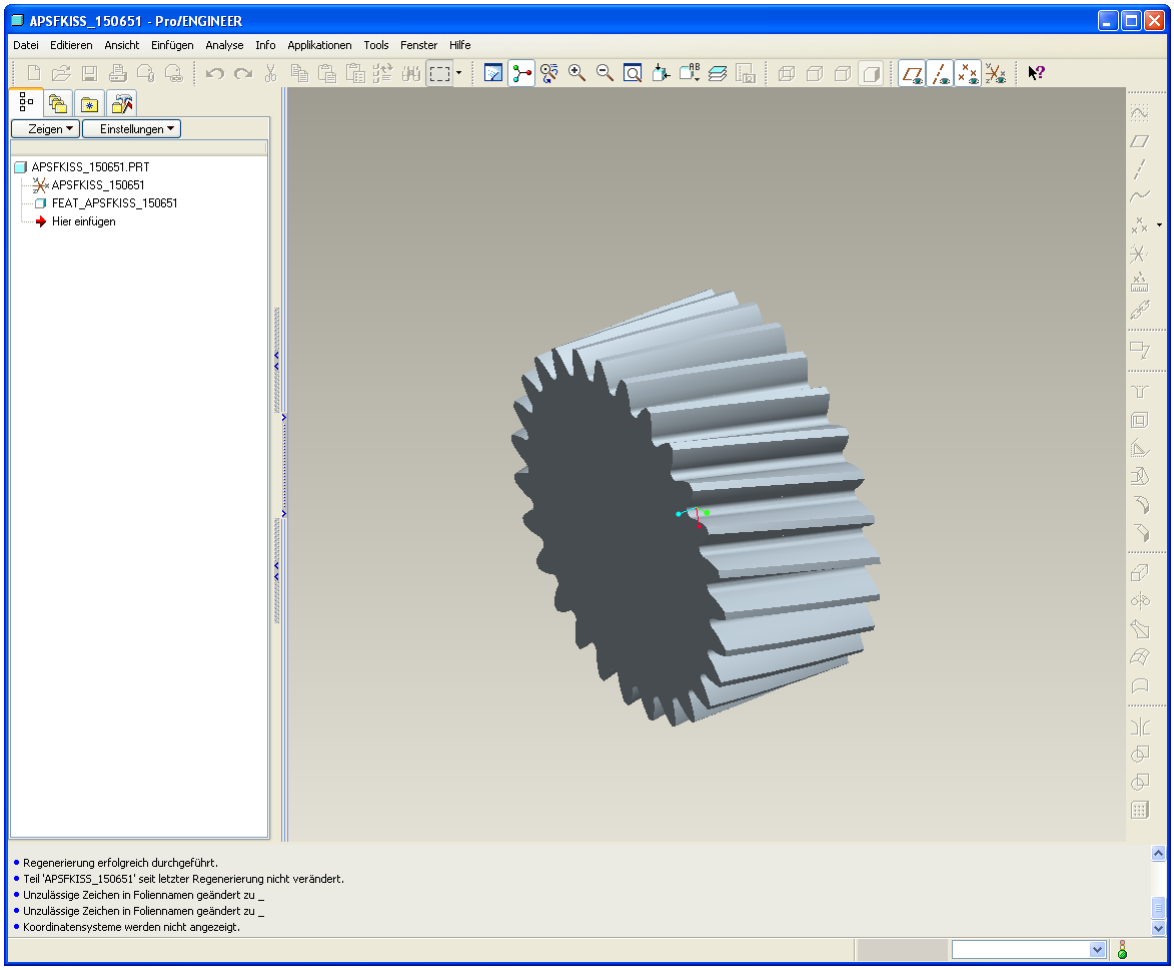
If you then click OK, KISSsoft opens in the corresponding module with the toothing's calculation file loaded.



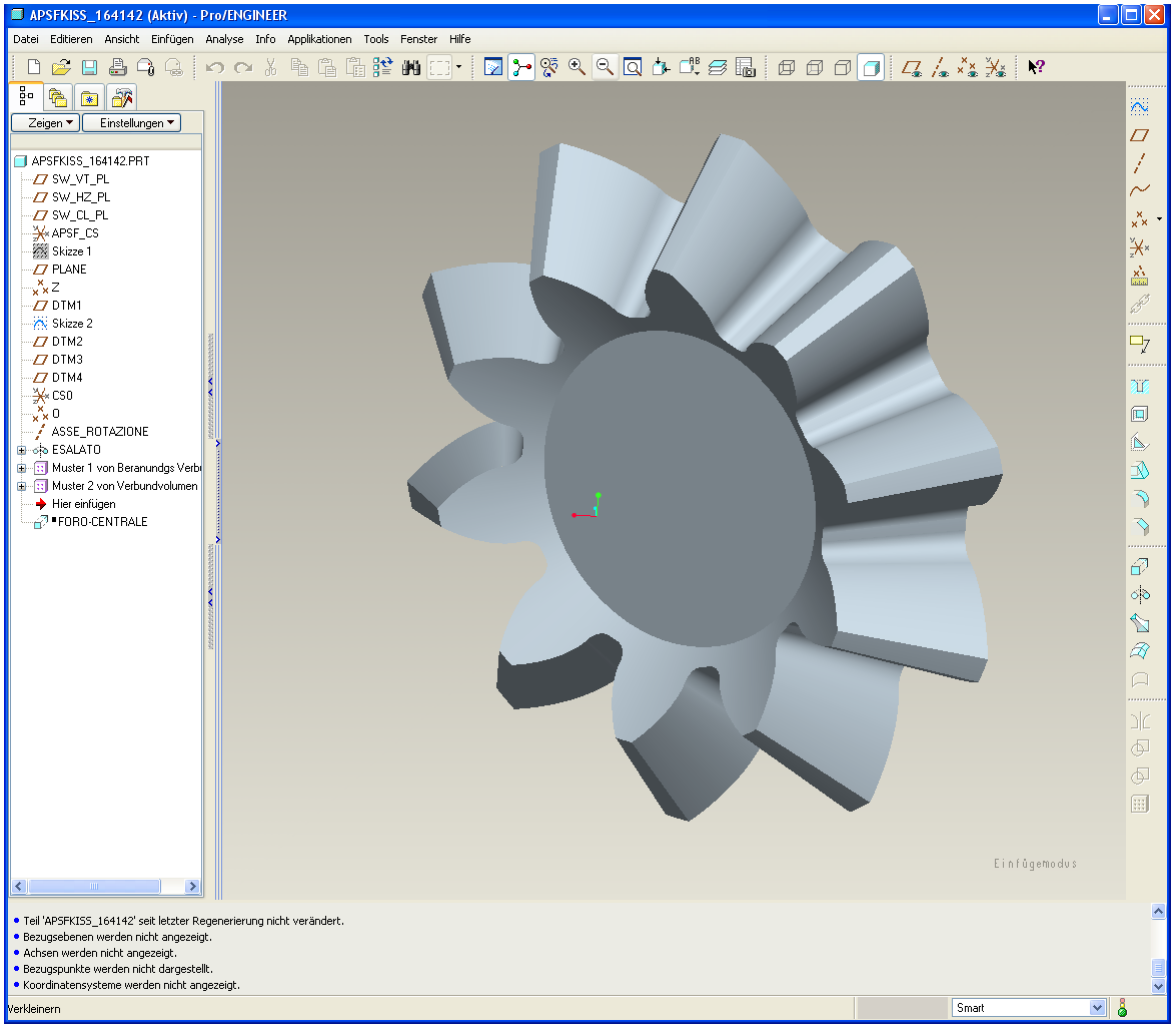
10.8.3 Add-in functions (calls)

10.9 3D interface to Creo Parametric (ProEngineer)

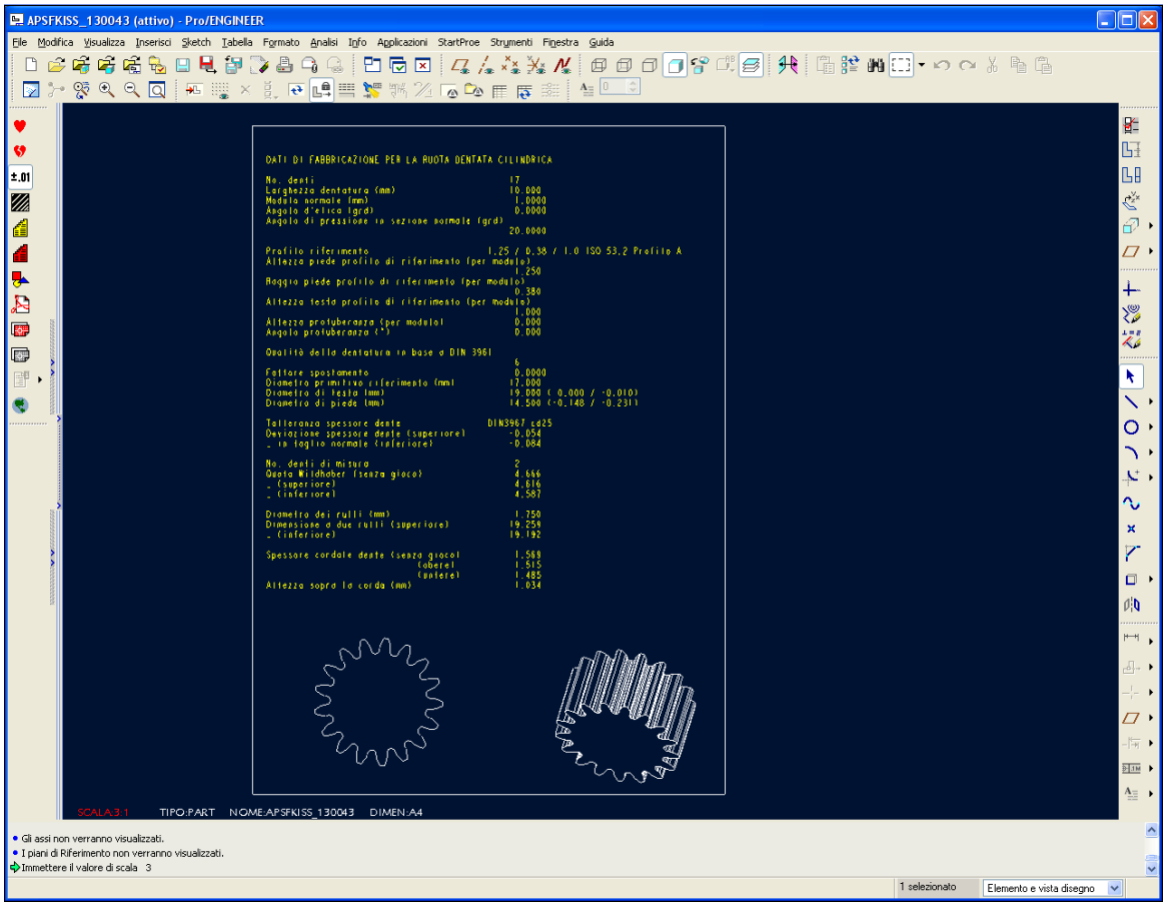
Manufacturer: Applisoft Europe (Italy)



The cylindrical or bevel gears calculated in KISSsoft can then be generated directly in Creo Parametric (ProEngineer) as a 3D part (see page I-175) with a real tooth form. You can create cylindrical gears with straight or helical teeth, which are external or internal, or straight-toothed bevel gears, as defined in DIN 3971, Figure 1.



In addition to the part, the system opens a drawing into which the gear manufacturing data are inserted, in a table. Open the CAD system before you start generating a part in the 3D interface to Creo Parametric (ProEngineer).



In the interface to ProEngineer, you can enter additional variables in the files for the particular gear (e.g. Z10GEAR1CAD.rpt) in the `CAD` directory. These additional variables will later be defined as parameters and saved in ProEngineer. The parameters used for the generation are already defined in ProEngineer and can no longer be used. Predefined parameters:

- pz, z, b, da, d, df, di, elica, USUnit

If you want to create a model of a part in US customary units (not metric), go to the `kiss.ini` (see page I-57) file and set the `USCUSTOMARYUNITS` parameter to 1.

You can also change an existing toothing without actually affecting the part (see Modifying a selected 3D model (see section "Modifying the selected 3D model" on page I-224)).

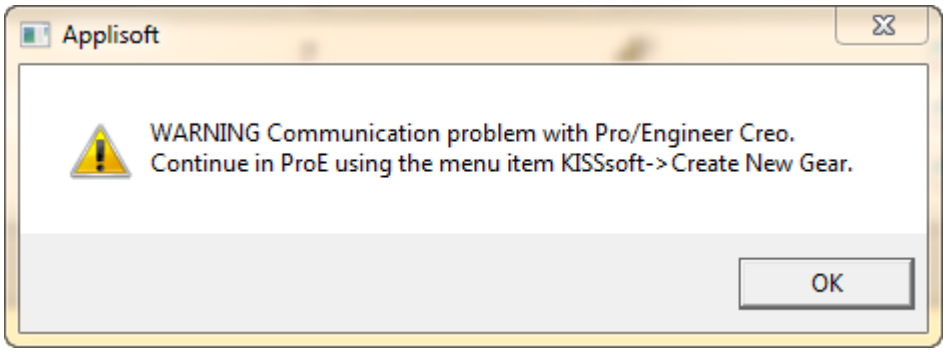
You can also cut a toothing on an existing shaft (see Cutting intermeshing on an existing shaft (see section "Cutting teeth on an existing shaft" on page I-225)).

A new dialog opens as soon as you start the generating process:

here you can select what you require by clicking `Generate gear in new file` to generate the gear in a new part file.

If you cannot set up a communication link to ProEngineer, it may be because the PRO_COMM_MSG.exe file is being blocked by a firewall or an anti-virus program.

In this case, a message is displayed to tell you what to do so that you can still generate the gear:



NOTE:

You can either set your anti-virus program to permit the **pro_comm_msg.exe** and **apsfkissvb.exe** processes to run, or generate the gear directly in the KISSsoft menu in ProEngineer.

If you want to prevent the selection menu or message from being displayed, you can specify this (see Changing base settings in the interface (on page I-228)).

10.9.1 Integrating the KISSsoft Add-in

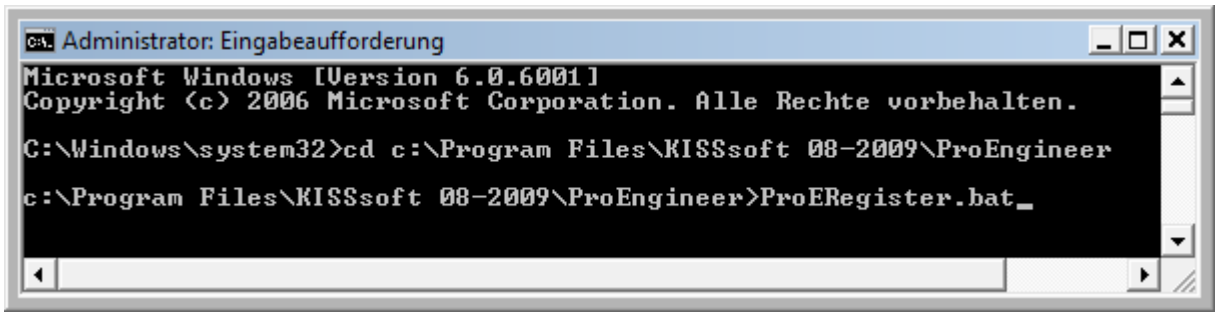
The KISSsoftCOM server should be registered as part of the installation process. However, if this did not succeed, and the KISSsoft interface does not work, you must register KISSsoftCOM manually.

This is how to perform manual registration.

Windows Vista/7/8:

As you must have administrator rights in order to perform the registration, this is only possible here with the command prompt.

1. Start the command prompt as the administrator.
2. Go to the location (`ProEngineer` folder) where the registration file is to be executed. Confirm by pressing `Enter`.
3. Run the registration file. Confirm by pressing `Enter`.

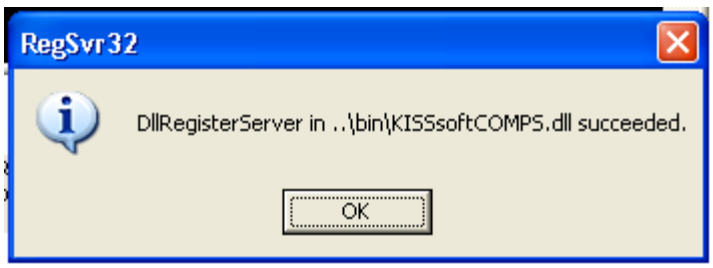


If KISSsoft is running on a server drive, you can enter the following command at the prompt to ensure you can access this drive as an administrator.

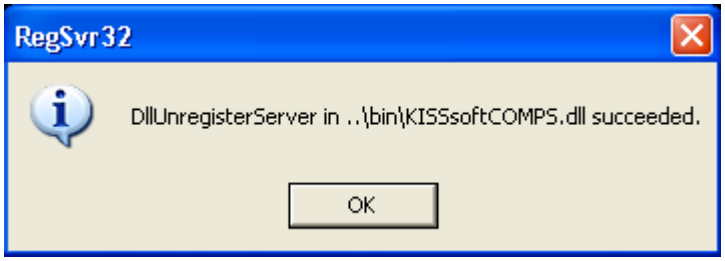
```
Pushd \\SERVER\directory
```

This command assigns a temporary drive letter to the directory. You can then go to where the *.bat file is stored and register the interface.

The following message is displayed if the KISSsoftCom server has been successfully registered.

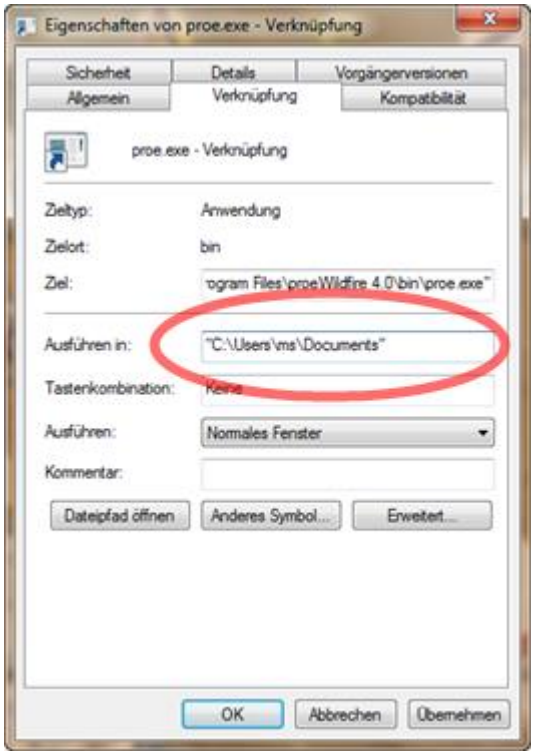


To remove the registration, double-click on the `ProEUnRegister.bat` file in the KISSsoft installation folder. This message is displayed if the process has been performed successfully.



Use one of the three following options to ensure the KISSsoft menu is present every time you start ProEngineer:

1. You can copy the **Protk_EditGear_wf4__64bit.dat** file (depending on the version of ProEngineer) to this Creo subdirectory: **.../Common Files/.../text/** (for ProEngineer in to the **.../text/** subdirectory).
   Then rename the file to **Protk.dat**.
   Using this method you can change your ProEngineer start directory to ensure that the KISSsoft menu always starts at the same time.
   If a different **Protk.dat** file is already present, you can add lines from the **Protk_EditGear_wf4_64bit.dat** file to the **Protk.dat** file.
2. Copy the **Protk_EditGear_wf4_64bit.dat** file to the ProEngineer initial working directory. Rename the file to **Protk.dat**.



This method requires you to copy the **Protk.dat** file to the Start directory (you will find the path displayed under Properties).

3. Then enter the following line in your **config.pro** file (in ProEngineer). (This is where you define your own path):
   protkdat C:\Program Files\KISSsoft 03-2011\ProEngineer\Protk_EditGear_wf4_64bit.dat
   Select the path to the file that contains the name of your version of ProEngineer.
   This method saves you having to copy or rename any files.

Description of the content of the Protk.dat file:

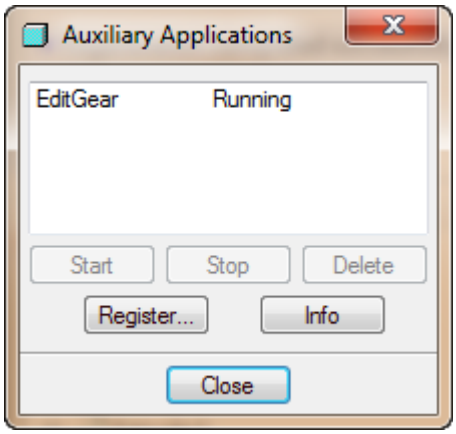
```
NAME EditGear

EXEC_PATH C:\Program Files (x86)\KISSsoft 03-
2011\ProEngineer\EditGear\bin_nt\EditGear_64bit_wf4.dll

TEXT_PATH C:\Program Files (x86)\KISSsoft 03-
2011\ProEngineer\EditGear\text.GB

STARTUP DLL

ALLOW_STOP TRUE

UNICODE_ENCODING FALSE
```

```
END
```

`EXEC_PATH` and `TEXT_PATH` must be the absolute path of the installation.

`STARTUP DLL` and `UNICODE_ENCODING FALSE` are predefined (do not change them).

`ALLOW_STOP TRUE` Use this to stop the ProEngineer program (`Tools->Auxiliary Application->Stop`).



You can delete this line in the **Protk.dat** file to prevent the user from stopping the interface.

`NAME EditGear` and `END` must be present, although you can change the **EditGear** name if required.

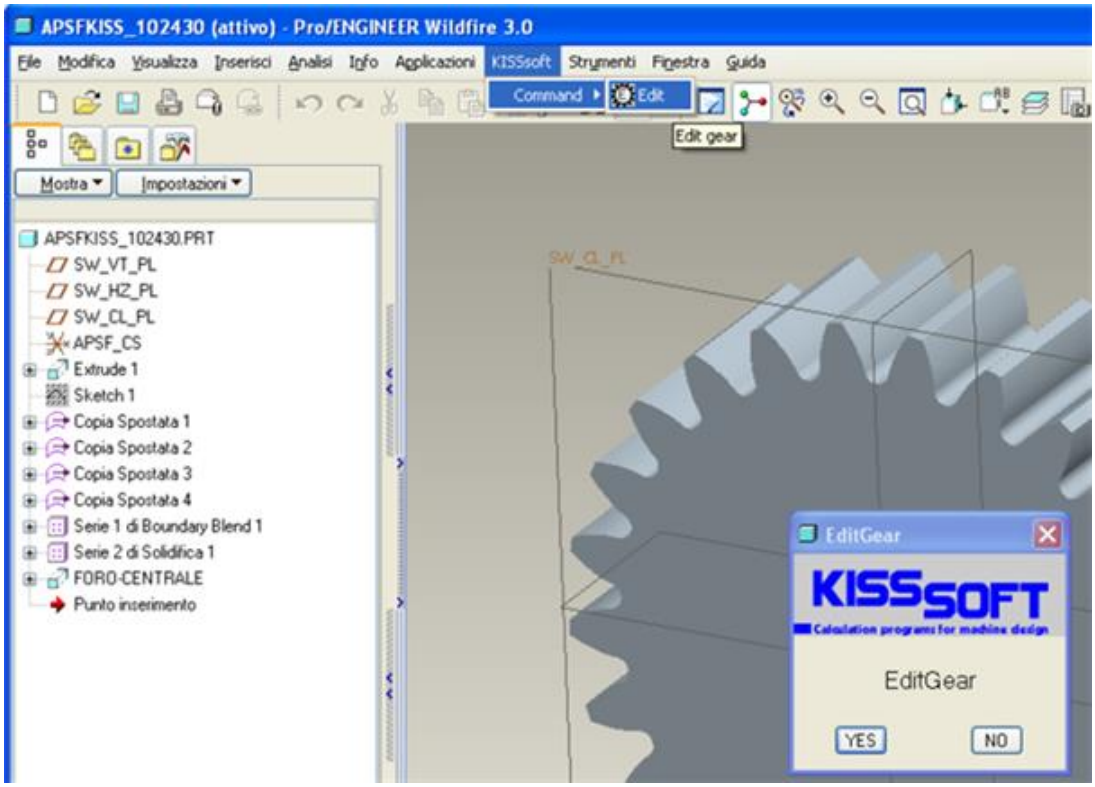
10.9.2 Modifying the selected 3D model

Every time you export a tooth form from KISSsoft, the model is generated in a new part in ProEngineer.

To modify an existing model:

1. Import the model you want to modify into ProEngineer (or use the current part)
2. Go to the KISSsoft menu and select `Edit` and then click `Yes` (to import the current toothing)

3. Then select `Open calculation file`. KISSsoft is then loaded with the corresponding toothing data.



KISSsoft can then regenerate the modified intermeshing and therefore adapt the existing toothing.

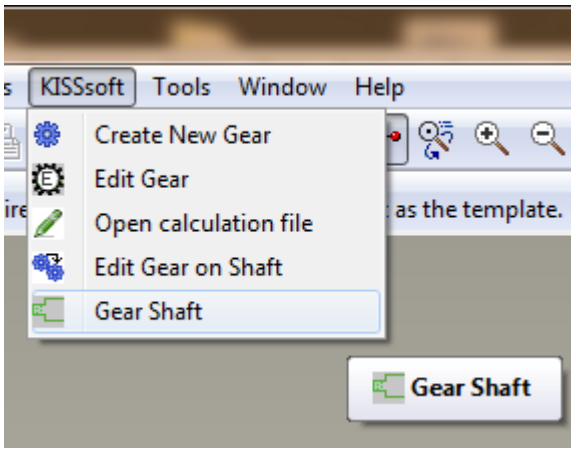
10.9.3 Cutting teeth on an existing shaft

The following menu is displayed if you activate the KISSsoft 3D export:

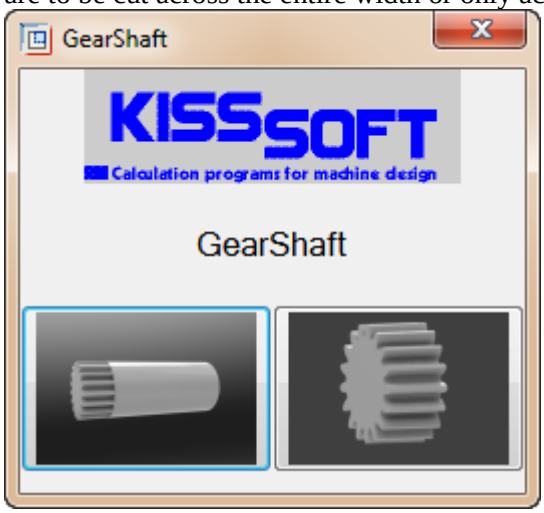
To modify an existing model:

1. Select `Generate gear on shaft`

2. In ProEngineer, open the shaft on which you want to cut the gear teeth.
3. Set a new system of coordinates to describe the point at which the gear teeth are to be cut. You can use this coordinate system if you want the gear teeth to be cut from the point of origin.
4. Select the `GearShaft` menu item in the KISSsoft menu in ProEngineer.



5. This opens another menu in which you can specify whether the gear teeth are to be cut across the entire width or only across part of the shaft.

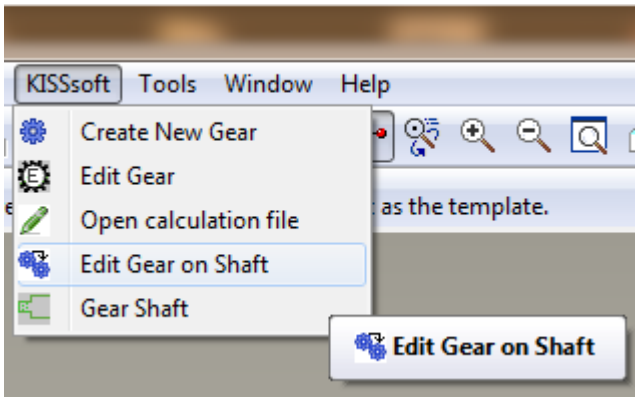


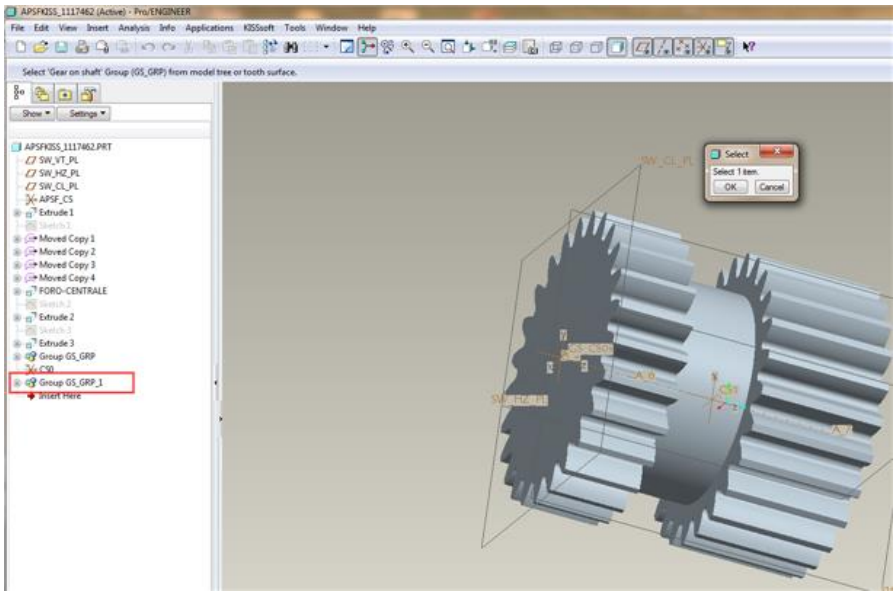
6. After you select the option you require, you can then select the coordinate system into which the toothing is to be inserted. (the co-ordinate system you select must have a Z-axis that is equal to the shaft axis).
7. The gear teeth are then cut on the shaft.

10.9.4 Modifying the teeth on an existing shaft

Follow these steps to modify the teeth of a gear on an existing shaft in a model that has already been generated (gear generated using the KISSsoft interface):

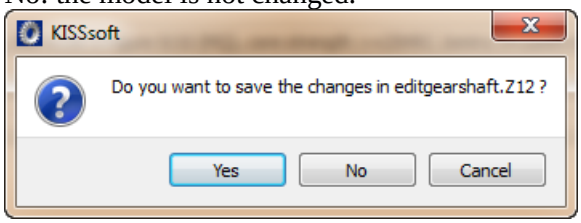
1. Import the model you want to modify into Creo Parametric (ProEngineer), or use the current part.
2. Select the `Edit Gear on Shaft` menu item to select the toothing you want to modify. The system then opens KISSsoft directly with the data that was saved for the toothing element when it was generated.





3. You can then modify or recalculate the toothing in KISSsoft, and restart the 3D export for corresponding toothing.
   Then, click on the cross at the top right to close the KISSsoft window.

You then see this prompt to save or not save the temporary calculation file.
Yes: the model is modified
No: the model is not changed.



4. The system now modifies the model if you confirm the prompt.

10.9.5 Changing base settings in the interface

You can set up your interface in a number of different ways. For example, you can tailor it by setting environment variables:

KISS_PROE_INTERFACE_NO_MENU = YES

This is designed for users who do not have a connection to Creo Parametric (ProEngineer) (via PRO_COMM_MSG.exe).

If you set this environment variable to "Yes", the interface will no longer attempt to use this process to manage the connection. You will also no longer see a warning that this connection is not possible.

KISS_PROE_INTERFACE_NO_MENU = NO

If you set this environment variable to "No", a warning is displayed if no direct connection to ProEngineer can be created.

The warning message describes what you must do to generate the gear even though the connection to ProEngineer is not present.

KISS_PROE_INTERFACE_CLASSIC = YES

The extra dialog in which you can select either "Generate gear in a new file" or "Generate gear on shaft" is not now displayed.

KISS_PROE_INTERFACE__CLASSIC = NO

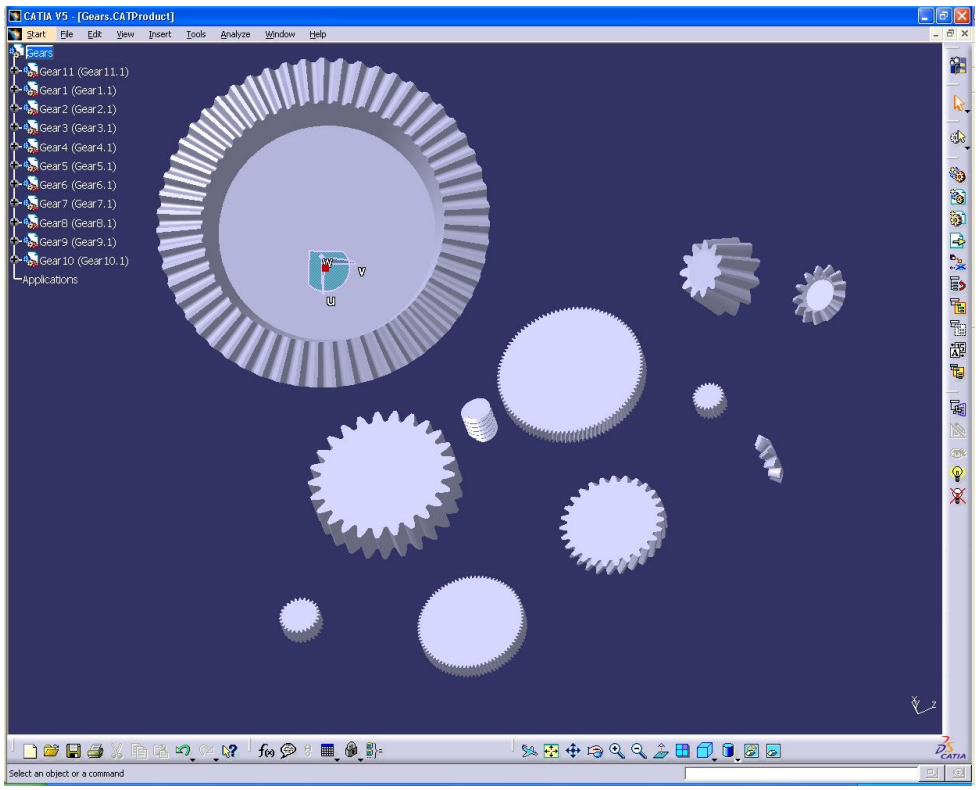
A dialog is displayed in which you can select either "Generate gear in a new file" or "Generate gear on shaft".

If no environment variables are set, both these values are set to **NO**.

10.10 3D interface to CATIA

Manufacturer: SWMS (DE)

Cylindrical or bevel gears calculated in KISSsoft can be generated directly in CATIA as a 3D part (see page I-175) with a real tooth form. You can create cylindrical gears with straight or helical teeth, which are outside or inside, or straight-toothed bevel gears, as defined in DIN 3971, Figure 1.
Furthermore, you have the option of adding toothing to existing shafts at a later point in time.
You will find a more detailed description of the interface in a *.pdf file in the `CATIA` folder in the KISSsoft installation folder.



10.10.1 Registering the interface

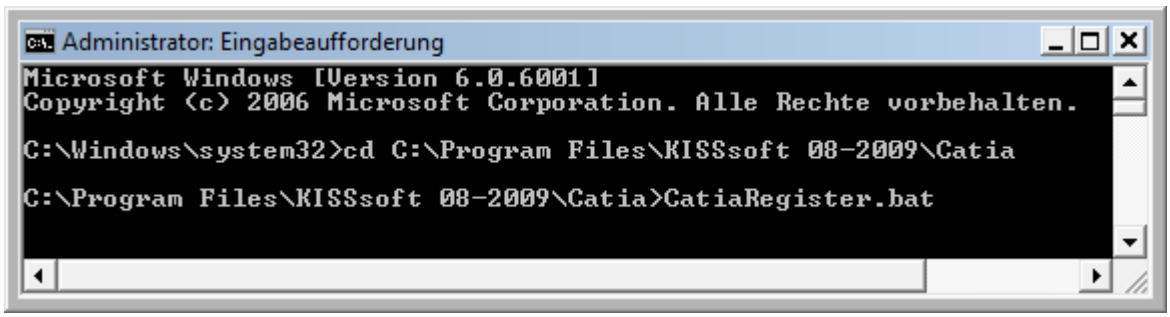
The CATIA interface should be registered as part of the installation process. If this did not succeed, you must register the interface manually.

This is how to perform manual registration.

Windows Vista/7/8:

As you must have administrator rights in order to perform the registration, this is only possible here with the command prompt.

1. Start the command prompt as the administrator.
2. Go to the location (`Catia` folder) where the registration file is to be executed. Confirm by pressing `Enter`.
3. Run the registration file. Confirm by pressing `Enter`.

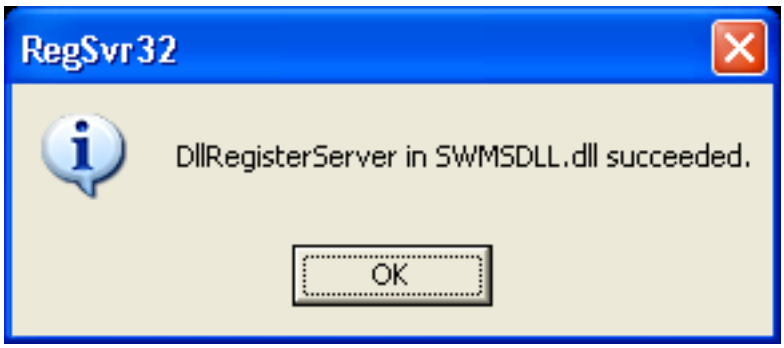


If KISSsoft is running on a server drive, you can enter the following command at the prompt to ensure you can access this drive as an administrator.

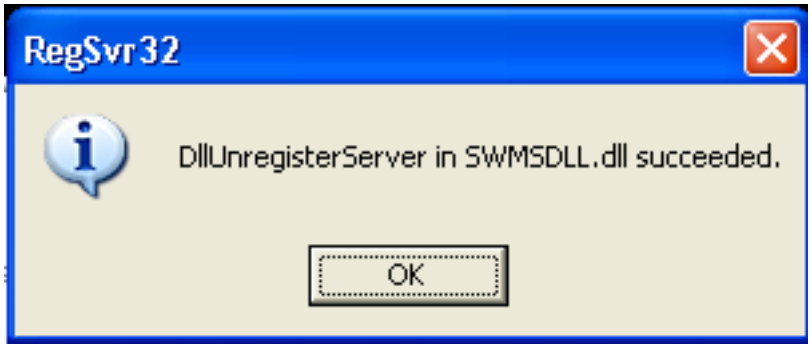
```
Pushd \\SERVER\directory
```

This command assigns a temporary drive letter to the directory. You can then go to where the *.bat file is stored and register the interface.

The following message is displayed if the registration was successful.

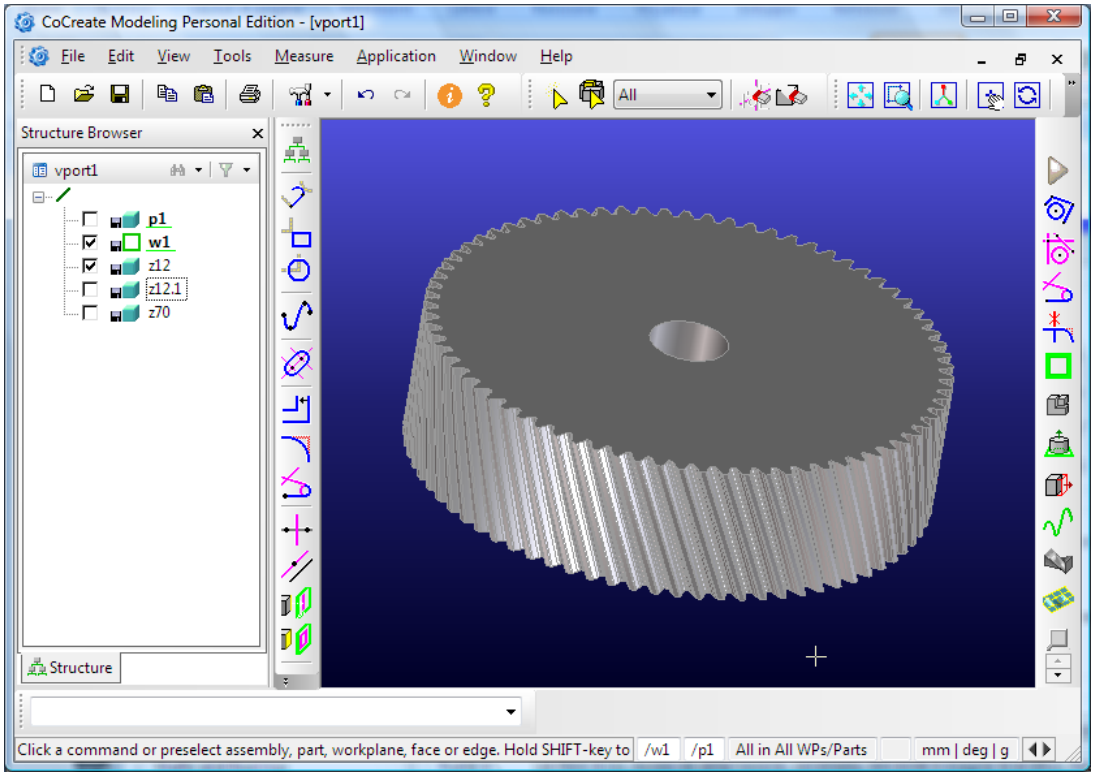


To remove the registration, double-click on the `CatiaUnRegister.bat` file in the KISSsoft installation folder. This message is displayed if the process has been performed successfully.

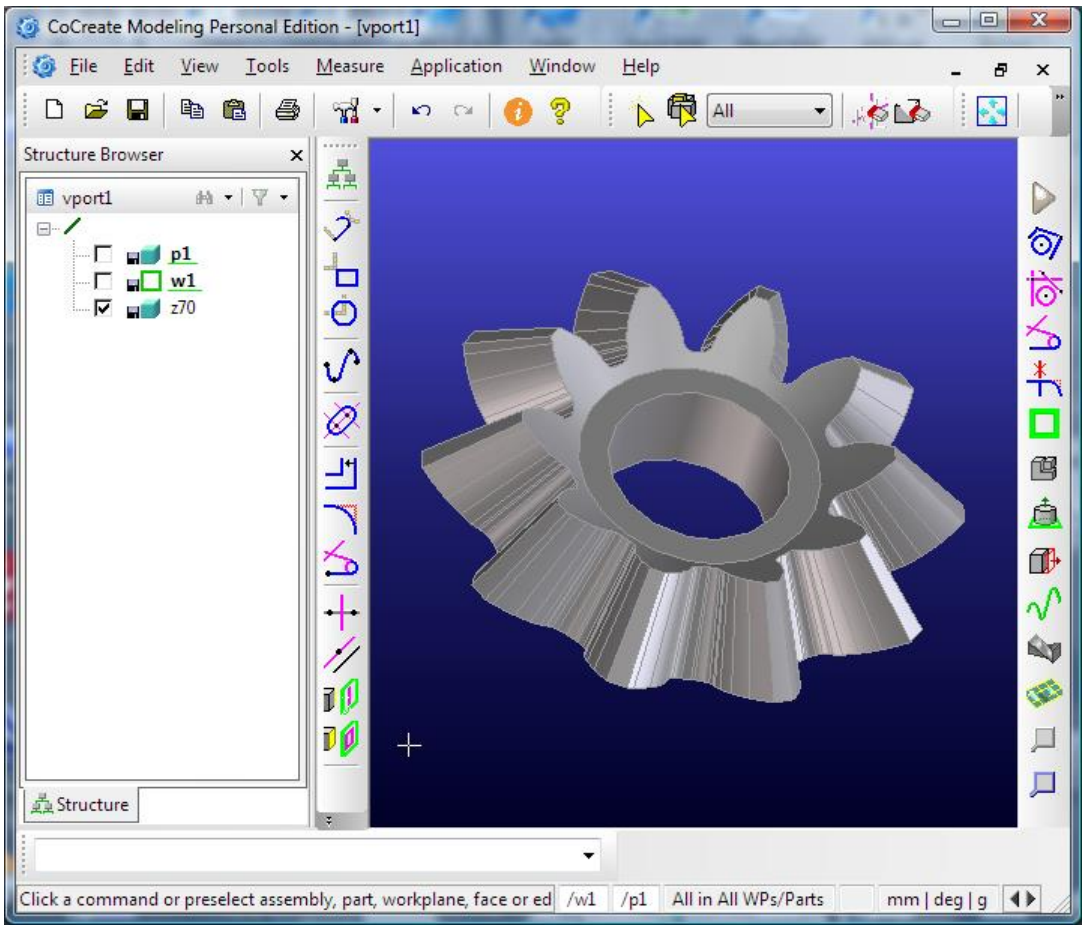


10.11 3D interface to CoCreate

Manufacturer: Studio Tecnico Turci (Italy)

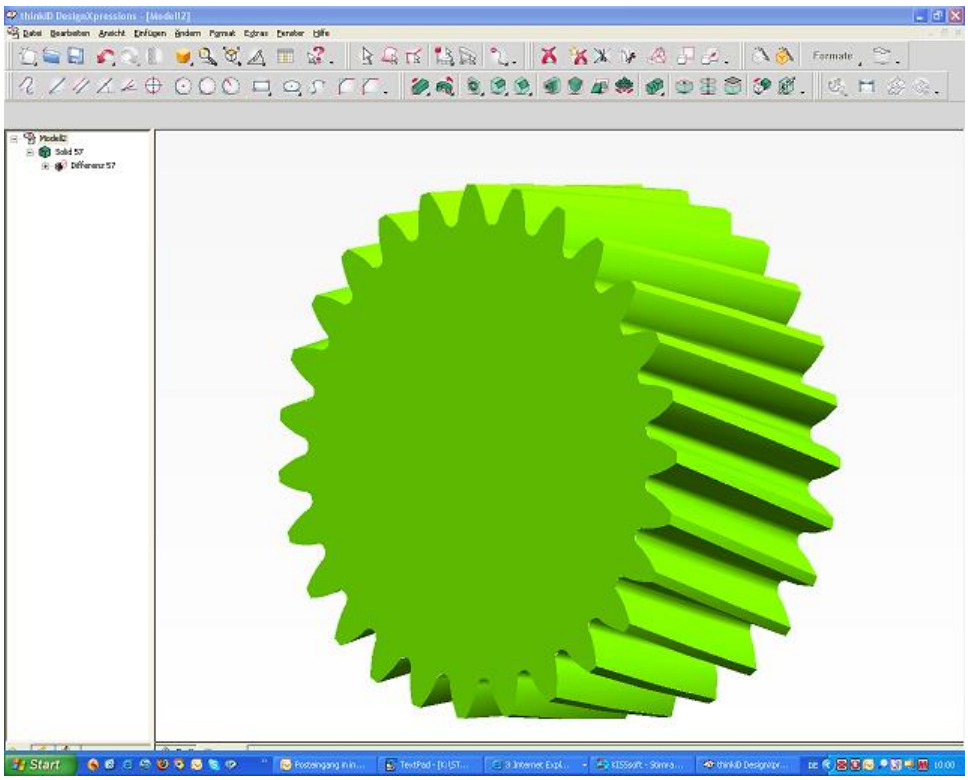


Cylindrical or bevel gears calculated in KISSsoft can be generated directly in the 3D interface to Creo Elements/Direct Modeling (CoCreate) as a 3D part (see page I-175) with a real tooth form. Simply click on the appropriate button to start Creo Elements/Direct Modeling (CoCreate interface) from KISSsoft. This opens a new part and generates the corresponding part. You can create cylindrical gears with straight or helical teeth, which are external or internal, or straight-toothed bevel gears, as defined in DIN 3971, Figure 1.

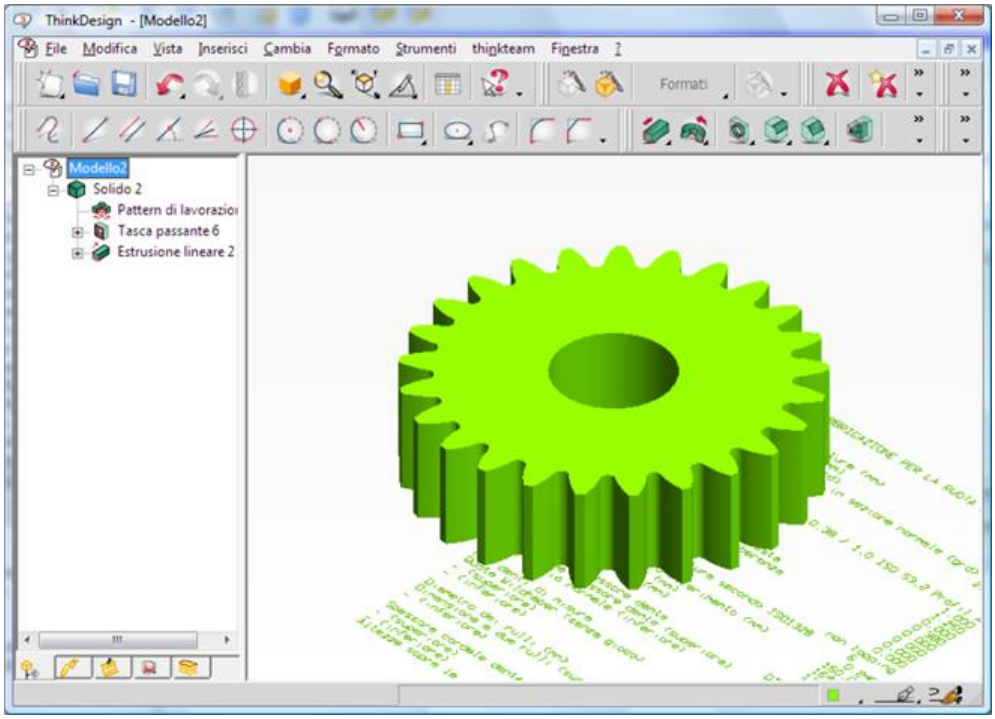


10.12 3D interface to ThinkDesign

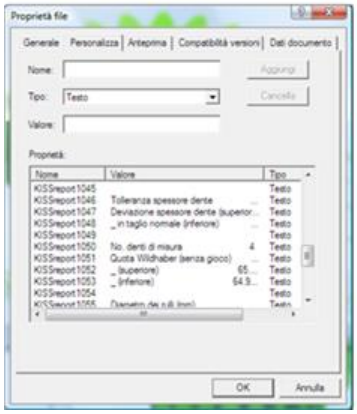
Manufacturer: Studio Tecnico Turci (Italy)

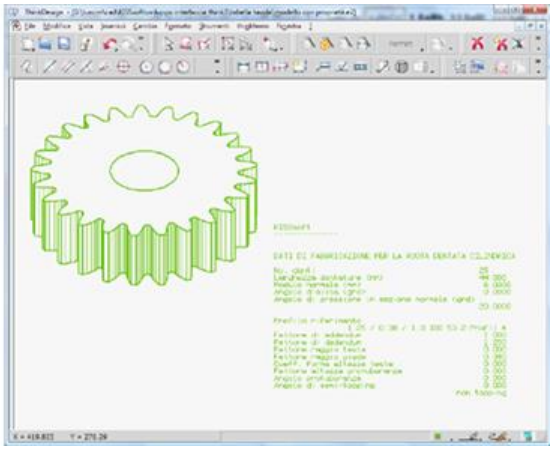


Cylindrical or bevel gears calculated in KISSsoft can then be generated directly in ThinkDesign as a 3D part (see page I-175) with a real tooth form. You can create cylindrical gears with straight or helical teeth, which are external or internal, or straight-toothed bevel gears, as defined in DIN 3971, Figure 1.
You must open the CAD system before you can generate a part in ThinkDesign.



The gear data for the drawing is stored both in the model and as file settings.

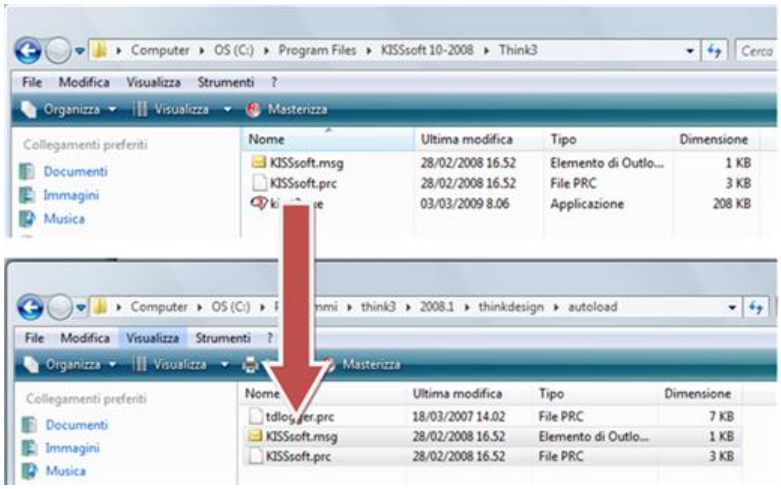




The information can be inserted in the drawing as symbolic text.

10.12.1 Integrating the KISSsoft Add-in

If the KISSsoft menu does not automatically appear in the CAD system, you can copy the two files (KISSsoft.msg, KISSsoft.prc) from the `Think3` folder to the KISSsoft installation folder in the ThinkDesign installation .../`thinkdesign/autoload`.



10.13 3D interface to ASCON Kompas

Manufacturer: KISSsoft AG

Cylindrical or bevel gears calculated in KISSsoft can then be generated directly in Kompas as a 3D part (see page I-175) with a real tooth form. Shafts calculated with KISSsoft can be generated directly in Kompas as a 3D part comprising cylinder and cone elements (see page I-177). From within KISSsoft, you can start Kompas with one click on a button. The system opens a new part and the appropriate part will be generated. You can create cylindrical gears with straight or helical teeth, which are outside or inside, racks with straight or helical teeth, or straight-toothed bevel gears, as defined in DIN 3971, Figure 1, and shafts.

Chapter 11
Answers to Frequently Asked Questions

11.1 Change the output of angles in reports

Can you output angles (in calculations) in the KISSsoft angle report as degree values as well as decimal numbers?

Current form: ##.#### °

Required form: ## ° ## ’ ## ’’

To do this, modify the report template (*.rpt) accordingly. Refer to the notes in the Report templates (see page I-104) manual. The calculation is then performed in the report.

A helix angle is used to show this method:

Current form as a decimal:

Helix angle (grd) %11.4f{Grad(ZS.Geo.beta)}=>

Afterwards, the required form:

Helix angle (grd) %i° %i' %i" {Grad(ZS.Geo.beta)} {(Grad(ZS.Geo.beta)-int(Grad(ZS.Geo.beta)))*60} {(((Grad(ZS.Geo.beta)-int(Grad(ZS.Geo.beta)))*60-int((Grad(ZS.Geo.beta)-int(Grad(ZS.Geo.beta)))*60))*60}

11.2 Input materials for gear calculations in the database

When comparing the materials used for toothings in a particular company, it became evident that not all the required materials were present in the database provided by KISSsoft.

In particular, the following key values necessary for gear calculation are missing. These include $\sigma_{Flim/}S_{at}$, $\sigma_{Hlim/}S_{ac}$, R_{zF}, R_{zH}, $B_{M.}$

When you redefine materials and their properties, you must compare them with similar materials in our materials database.

First of all, define the basic data for a material in the database. Then define the gear-specific data for this base material.

Then calculate the values of $\sigma_{Flim/Sat}$, $\sigma_{Hlim/Sac}$ depending on the hardness values, as described in ISO 6336-5.

To do this, you can use either the relevant material diagram, the conversion function for Own input for materials (see page II-260) or formulae from ISO. The S_{at}, S_{ac} values are converted on the basis of σ_{Flim}, σ_{Hlim}.

If you do not know the thermal contact coefficient BM, simply leave this entry blank so that the default values are used in the calculation.

For medium total heights, specify average values with R_{zF} 10µm and R_{zH} 3µm. You will find more detailed information in ISO 6336-2.

You will find more information about the influence of medium total heights in our article under point 2 http://www.kisssoft.ch / deutsch / downloads / doku_artikelISO6336Neuheiten.pdf.

11.3 How can I test the software?

A demo version (see page I-40) of the software is available. Although the demo version does not have an expiration date, its functionality is limited so that, for example, you cannot change and store material data. The demo version is designed to give you an initial impression of the software. For a detailed trial, request a Test version (see page I-42). The test version runs for 30 days, is free of charge and is the same as the full version (without third party programs).

11.4 What licenses are available?

Individual user licenses and floating licenses are available for both KISSsoft and KISSsys. A floating license allows the software to be used at more than one workplace.

However, floating licenses are not available for some of the third party products, for example, some CAD interfaces.

11.5 Add your own texts in the results window

To do this, define a new file in the KISSsoft installation folder in "…\ext\rpt\". This file must then be named: "Modulname + result.RPT" (e.g. for a cylindrical gear pair Z012result.RPT).

Then define the new parameters or values that are to be added. These values then appear at the end of the "Results" window.

11.6 Restore previous stages of the calculation

(Select File > Restore... (acts like the Undo function) to retrieve an earlier state of the current calculation file. For this reason, every calculation run stores the current state as a point at which it can be restored. The list of restoration points is deleted when you open a different file.

Part II
General

Chapter 12
Introduction

KISSsoft provides calculation modules for different toothing types, ranging from cylindrical gears in different configurations to bevel gears and face gears to worm wheels. The input windows for the different gear calculations are very similar. There are also calculation options for multiple modules. The table below shows you all the input windows in the individual calculation modules.

Input window	Sec.											
Basic data	13.2	is supported by all calculation modules										
Load	13.3											
Factors	13.4											
Reference profile	13.5											
Tolerances	13.6											
Modifications	13.7											
Tooth form	13.8											
Flank breaking	13.9		✔	✔				✔				
Contact analysis	13.10		✔	✔	✔	✔	✔					
Operating backlash	13.12		✔	✔	✔	✔	✔			✔	✔	
Master gear	13.13	✔	✔	✔	✔	✔					✔	
AGMA 925	13.14		✔	✔	✔	✔						

Table 12.1

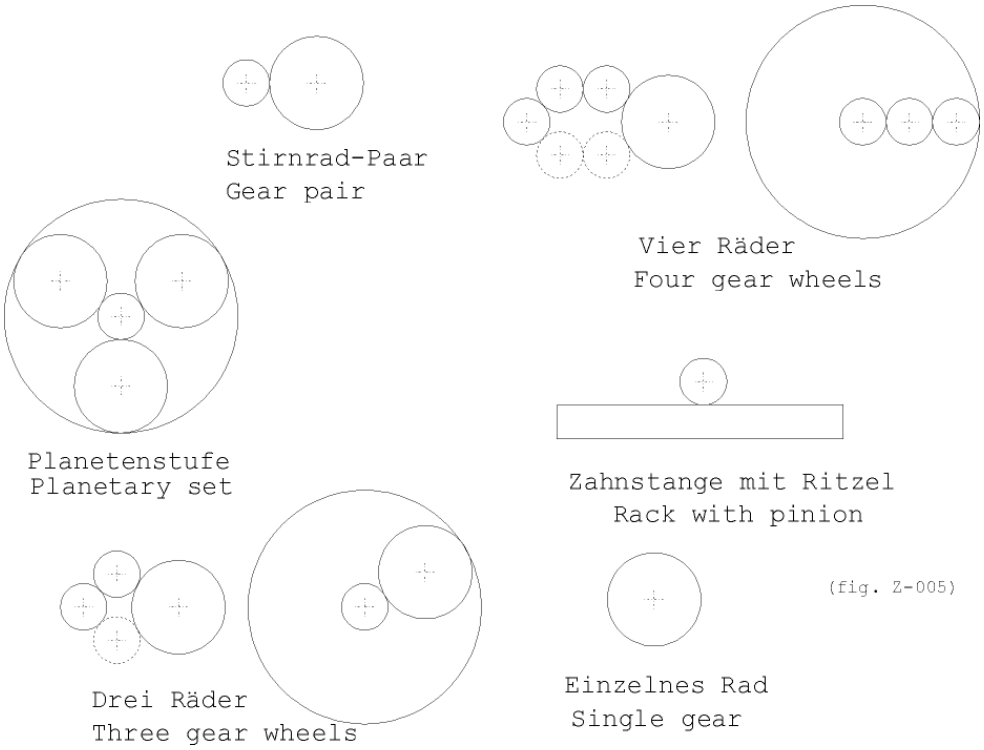
- Single gear, - Cylindrical gear pair, - Pinion with rack, - Planetary gear, - Three gears, - Four gears, - Bevel and Hypoid gears, - Face gears, - Worms with double enveloping worm wheels, - Crossed helical gears and precision mechanics worms, - Splines (Geometry and Strength)

Chapter 13
Cylindrical gears

You can use KISSsoft cylindrical gear calculation software to calculate a range of different configurations.

- The single gear calculation has been developed to calculate the geometry and test dimensions of individual gears
- The cylindrical gear pair is the most important configuration for geometry and strength. You can also use it for additional calculations and several individual calculations at the same time.
- The planetary gear software checks the practical aspects of the configuration and monitors both pairs of gears whilst they are being sized. The Fine Sizing function enables you to optimize the center distance quickly and efficiently. You can usually input your own values here. However, you must take into consideration that, as torque cannot be applied to the planet, it is not possible to perform a strength analysis on a Wolfrom drive or on a Ravigneaux gear set.
- The configurations for three and four gears enable you to calculate a gear wheel chain, in which torque is applied only to the first and last gear.
- The calculation used for a rack and pinion only includes one rack in the geometry calculation and one cylindrical gear with a large number of teeth for the strength calculation.

As the input screens for the different configurations are very similar, they are described together in the sections below.


Stirnrad-Paar
Gear pair
Vier Räder
Four gear wheels
Planetenstufe
Planetary set
Zahnstange mit Ritzel
Rack with pinion
Drei Räder
Three gear wheels
Einzelnes Rad
Single gear

(fig. Z-005)

13.1 Basic data

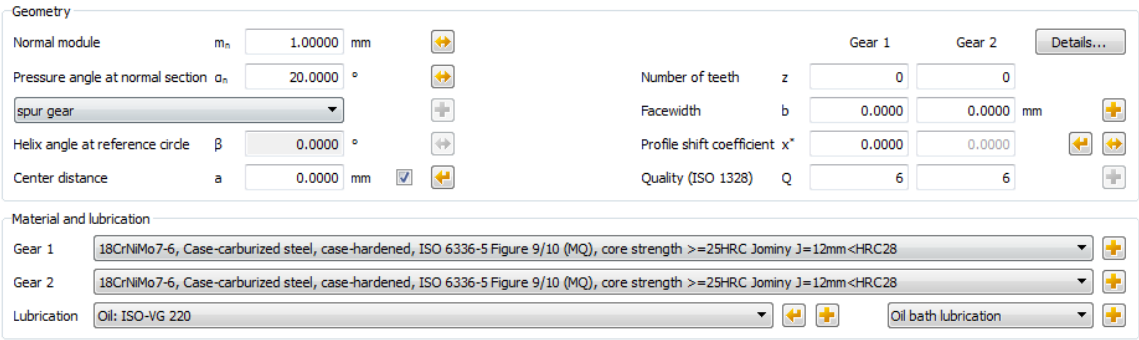


Figure 13.1: `Basic data` input window for cylindrical gear pair

The `Basic data` input window is one of the standard tabs (see page I-86) and is subdivided into the two groups `Geometry`, `Material and Lubrication`.

13.1.1 Normal module

Enter the normal module. The normal module defines the size of the teeth. A standard series is for example defined in DIN 780 or ISO 54. However, if you know the pitch, the transverse module or the diametral pitch instead of the normal module, click the button to open a dialog window in which the conversion will be performed. If you want to transfer the diametral pitch instead of the normal module, you can select `Input` normal diametral pitch instead of normal module by selecting `Calculation > Settings > General`.

13.1.2 Pressure angle at normal section

The normal pressure angle at the reference circle is also the flank angle of the reference profile. For standard toothings the pressure angle is $\alpha_n = 20^o$. Smaller pressure angles can be used for larger numbers of teeth to achieve higher contact ratios and insensitivity to changes in center distance. Larger pressure angles increase the strength and allow a smaller number of teeth to be used without undercut. In this situation, the contact ratio decreases and the radial forces increase.

13.1.3 Hand of gear for gear teeth

Hand of gear for gear teeth (→ see Figure on page II-251) defines the direction of the axial forces. A gear with helical teeth usually produces less noise than a gear

with straight teeth, but it generates an additional bending moment and an axial force. A gear with continuous double helical teeth consists of two halves of a helical gear with a different hand of gear. Although it does not generate any axial forces, it must be possible to adjust the gear along its axis and it is more difficult to manufacture. In a herringbone gear (with continuous double helical teeth), click the button to set the gap width b_n.

13.1.4 Helix angle at reference circle

Enter the helix angle in [°]. Click the button in the `Convert helix angle` window to calculate this angle from other values such as, for example, the overlap ratio and axial force.

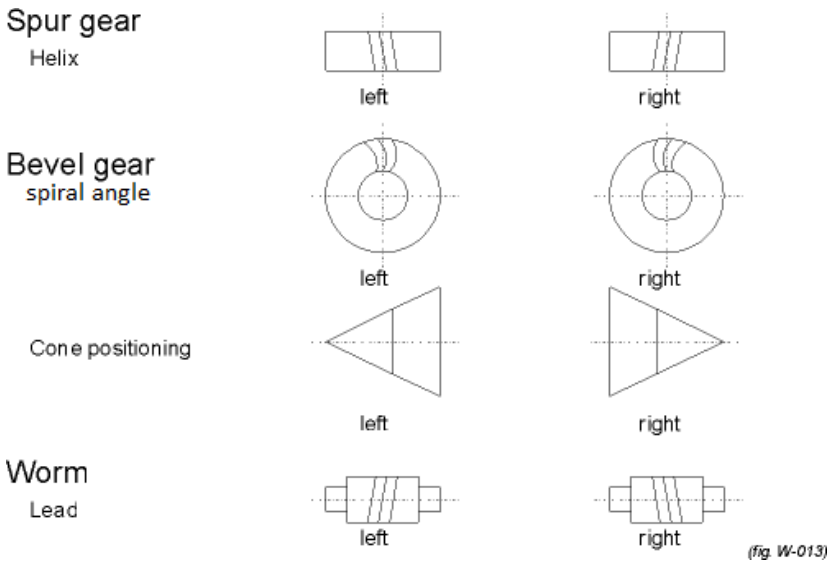


Figure 13.2: Helix angle at reference circle.

13.1.5 Center distance

As stated in ISO 21771, the axis center distance for external and internal gears is positive for two external gears and positive for an external gear paired with an internal gear. For internal teeth, the number of teeth on the internal gear and the axis center distance are always negative.

If you select the checkbox to the right of the axis center distance unit, the value used in the calculation will remain constant. Otherwise, the axis center distance will be calculated from the profile shift total.

Click the button to select one of the following sizing options:

- Fixed sum of profile shift coefficients. The axis center distance is calculated on the basis of a predefined profile shift sum. By clicking the button you can display a suggested value for the profile shift sum (in accordance with DIN 3992). The sum of profile shift influences the profile shift coefficients of both gears as well as the operating pitch circle and the operating pressure angle.
- Fixed profile shift coefficient Gear 1 (or 2), balance specific sliding. Optimize axis center distance with respect to balanced sliding: For a specified profile shift of a (selectable) gear, this option calculates the axis center distance in such a way as to balance gear pair specific sliding (for cylindrical gears). If the `Own input` item is not selected from the Own input drop-down list in the `Reference Profile` input window, this calculation is performed with automatic tip alteration as stated in DIN 3960. You can also enter your own tip alteration value in the `Basic data` input window by clicking the `Details...` button. In the `Define geometry details window` select the checkbox next to the `Tip alteration` input field.

13.1.6 Number of teeth

The number of teeth is, by default, a whole number. You can also enter the number of teeth as an amount with values after the decimal place (see section "Input of number of teeth with decimal places" on page II-429). For internal toothed gears, you must enter the number of teeth as a negative value as stated in ISO 21771. For a pinion-ring internal gear gear pair, the center distance must also be entered as a negative value (e.g. $z_1 = 20$, $z_2 = -35$, $a = -7.5$, $m_n = 1$).

The minimum number of teeth is limited by geometric errors such as undercut or tooth thickness at the tip. For spur gears without profile shift there is for example undercut if there are fewer than 17 teeth.

13.1.7 Facewidth

Normally the facewidth shouldn't be greater than 10 to 20 times the normal module, or also not greater than the reference circle of the pinion. The contact pattern deteriorates if the facewidth is too great. Click the button to the right of the facewidth input field to enter the axial offset b_v (see also Figure 13.3). The axial offset reduces the effective width for the strength calculation. The common width is used to calculate the pressure. A certain amount of overhang is taken into account for the Tooth root strength. The selected pinion width is often somewhat greater than the gear width.

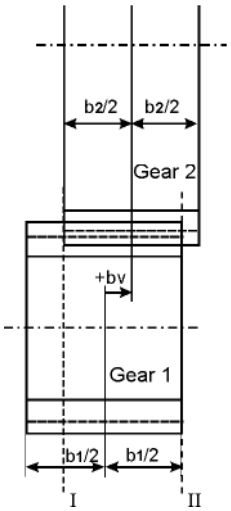


Figure 13.3: Axial offset b_v

In double helical gears[2] you must specify the total width of the gear teeth (i.e. the width of both halves together with the gap). To enter the width of the gap b_n, click the button on the right of the hand of gear drop-down list for the toothing.

13.1.8 Profile shift coefficient

Preliminary note: If the profile shift sum has not yet been specified, click the Sizing button (), to the right of the center distance (see page II-251) input field, to display a suggested value for the distance in the `Sizing center distance` window. The suggested value is based on DIN 3992 recommendations for well balanced toothing (Area P4/P5). You will find more information about this in DIN 3992 or in Niemann [64], Figure 22.1/6.

[2] *Double helical gears are gears that consist of two gear halves; the first half has a left hand helix and the second half a right hand helix.*

The tool can be adjusted for manufacture. The distance between the production pitch circle and the tool reference line is called the profile shift. To create a positive profile shift, the tool is pulled further out of the material, creating a tooth that is thicker at the root and narrower at the tip. To create a negative profile shift the tool is pushed further into the material, with the result that the tooth is narrower and undercutting may occur sooner. In addition to the effect on tooth thickness, the sliding velocities will also be affected by the profile shift coefficient.

The distribution of the total profile shift affects the tooth thickness, sliding movements and strength values. It can be performed in accordance with a range of different criteria. To achieve this, use the various sizing options provided by clicking the button in the `Sizing of profile shift coefficient` window.

- For optimum specific sliding
  The value suggested here shows the profile shift for a cylindrical gear pair that has been balanced for a specific sliding between the pinion and the gear. When more than two gears are involved, the profile shift coefficient is set to the smallest value that corresponds to the specific sliding movement at the root.
- For minimum sliding velocity
  The minimum sliding velocity at the tip of the two gears is often used for speed increasing ratios. In a cylindrical gear pair, this means both gears have the same sliding velocity and that the access and recess length of the path of contact are also the same.
- For maximum root safety
  The profile shift coefficient is defined iteratively for the range x^*_{min}, x^*_{max}.
- For maximum flank safety
  The profile shift coefficient is defined iteratively for the range x^*_{min}, x^*_{max}.
- For maximum scuffing safety
  The profile shift coefficient is defined iteratively for the range x^*_{min}, x^*_{max}.
- For gear 1 without undercut and point at tip (min)
  The minimum value of the profile shift coefficient for gear 1 is calculated from the undercut boundary of gear 1 and the minimum topland of gear 2.
- For gear 1 without undercut and point at tip (max).
  The maximum value of the profile shift coefficient for gear 1 is calculated from the minimum topland of gear 1 and the undercut boundaries of gear 2.
- For undercut boundary per gear.
  The proposed value only refers to the selected gear. No check is performed to

see whether the resulting profile shift is also permitted for the other gear in the pair. For more information, please refer to the explanations above.

- For minimum topland per gear.
  The proposed value only refers to the selected gear. No check is performed to see whether the resulting profile shift is also permitted for the other gear in the pair. You can specify the minimum thickness of the topland in `Calculation > Settings > General > Coefficient for minimum tooth thickness at the tip`. For more information, please refer to the explanations above.

Click the button and KISSsoft will determine whether the profile shift coefficient is to be taken from measured data or from values given in drawings.

The following options are available here:

- **Base tangent length**
  Here you must enter the base tangent length (span) and the number of teeth spanned. This option cannot be used for (internal) helical gear teeth because their base tangent length cannot be measured.
- **Measurement over two balls**
  To do this, enter this dimension and the diameter of the ball. In a gear with helical gear teeth and an odd number of teeth, the measurement over balls is not the same as the measurement over two pins, see Measurement over pins.
- **Measurement over 2 pins**
  To do this, enter this dimension and the diameter of the pin. For helical gear teeth and gears with an odd number of teeth, you must also enter a minimum span. This measurement cannot be calculated in internal helix gears.
- **Measurements over 3 pins**
  Here, enter the measurement over pins and the pin diameter. For helical gear teeth and gears with an odd number of teeth, this is equivalent to the measurement over 2 pins. You cannot use this option for internal and helical gear teeth or gears with an even number of teeth.
- **Tip circle**
  This is a rather imprecise calculation because the tip diameter does not always depend solely on the profile shift.
- **Tooth thickness at reference circle**
  Here, you specify the tooth thickness. You can also enter the arc length or chordal length, and whether the value is in transverse or normal section.

NOTE

If one of the two profile shift values appears in gray, this means it will be calculated by . This is what happens when you activate the checkbox for entering the center distance. If you overwrite a gray field, it will become active and KISSsoft will calculate the value for one of the other gears .

13.1.9 Quality

In this input field, you specify the accuracy grade in accordance with the standard shown in brackets. To change the standard used for this calculation, select `Calculation > Settings > General > Input of quality`. The accuracy grade specified in ISO 1328 is approximately the same as the quality given in DIN 3961 or BS 436/2.

The qualities that can be achieved are displayed in the Quality values (see Table "Quality" on page IV-861) table.

Manufacturing process	**Quality in accordance with DIN/ISO**		
Grinding	2	. . .	7
Shaving	5	. . .	7
Hobbing	(5)6	. . .	9
Milling	(5)6	. . .	9
Shaping	(5)6	. . .	9
Punching, Sintering	8	. . .	12

Table 13.1: Quality values for different manufacturing processes

Conversion of qualities in accordance with AGMA:
When converting qualities as defined in AGMA 2015-1-A01, Annex B.2 the total of the quality figures in version 2015 (comparable with ISO) and version 2000 equals 17.

Quality as specified in ISO 1328 and AGMA: 2015	**Q. in accordance with AGMA 2000**
1	16
2	15
3	14
4	13
5	12
6	11
7	10
8	9
9	8
10	7
11	6

Table 13.2: Quality values in different standards

If you want to define different tolerances, click `Calculation->Settings>General` and set the `Varying qualities` flag.

This activates the Plus button next to Quality in the main screen. Click the Plus button to open a new window in which you can enter the tolerances you require.

You can input the tolerances in standard-specific tabs. The changes in the window are then applied to all the gears in the calculation module.

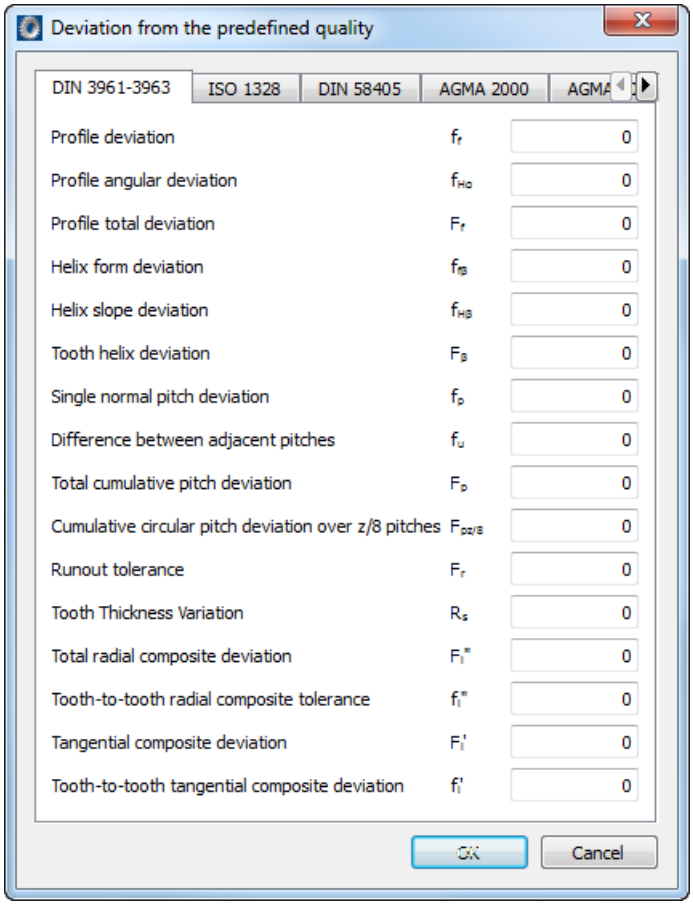


Table 13.3: Input window for different tolerances

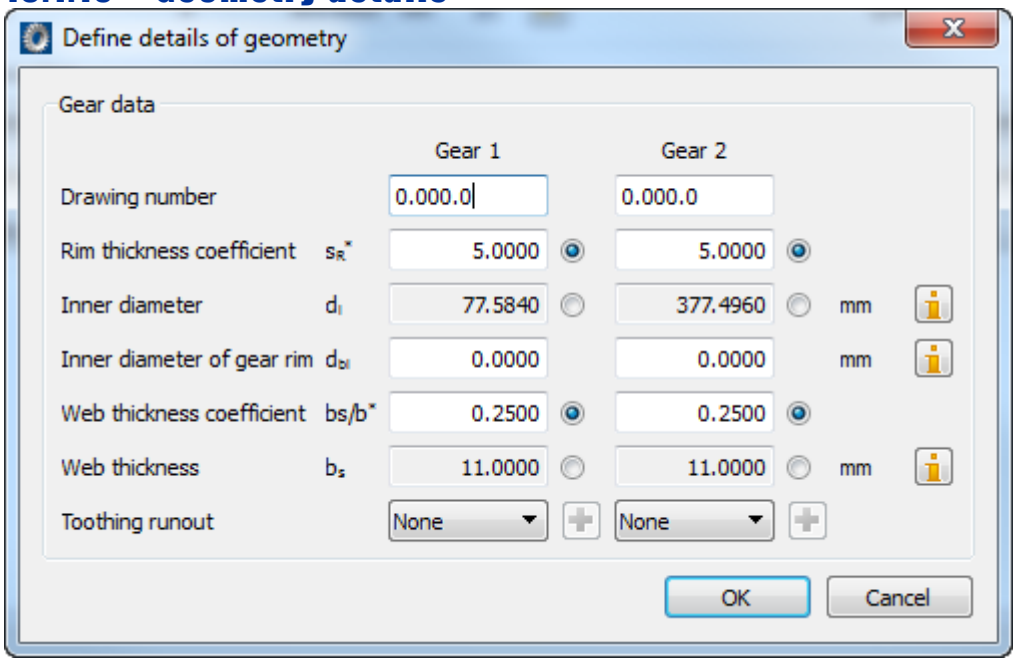
This is the table in which you input any deviation from the base quality (specified in the "Basic data" tab). Example: The base quality of gear 1 is 6. If you then input +2 for the runout tolerance, the runout tolerance will be calculated with a quality of 8.

In every case, only those tabs (standards) are displayed that are possible for the calculation module.

The user entries remain in this window as long as you continue using the same calculation module. You can therefore import a different file, and set the flag.

The same entries will still appear in the window next to the Plus button. You only need to input the data again if you change calculation module.

13.1.10 Geometry details



To open the `Define geometry details` window, click the `Details...` button in the upper right-hand part of the Geometry area. Here you can change the values for:

- Drawing number
- Rim thickness coefficient S_R^*
- Inside diameter d_i
- Inside diameter of rim d_{bi}
- Web thickness coefficient b_s/b^*
- Web thickness b_s

The drawing number is only used for documentation purposes. You can enter any text here.

The inside diameter is needed to calculate the mass moment of inertia. For solid wheels, enter 0, for external wheels with a web, enter the corresponding diameter d_i as shown in Figure 13.4. For internal wheels, enter the external diameter of the gear rim.

In accordance with ISO or AGMA, the gear rim thickness s_r, defined by the inside diameter of rim d_{bi}, affects the strength. If no gear rim thickness is present, you can enter a value of 0 for d_{bi}. In this case the gear rim thickness s_r will be determined

from the diameter d_i. Where thin gear rims are used, this factor can greatly influence the calculation of safety factors. For thin gear rims, this value can also be calculated in accordance with VDI 2737 (see page II-439).

Web thickness coefficient: If the inside diameter di > 0, the value input for the web thickness (bs or bs/b) is taken into account. If bs/b = 1.0, this means no web is present. In this case, the gear body coefficient CR is 1.0. The ratio b/bs can vary between 0.2 and 1.2. In this case, CR is then < 1 (if b/bs < 1) or > 1 (if b/bs > 1). The coefficient CR is then used to calculate the tooth contact stiffness (cγα).

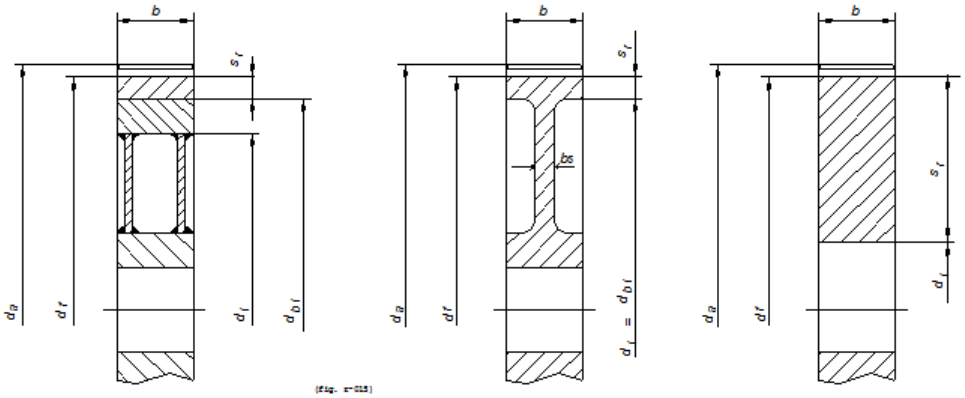


Figure 13.4: Measuring the diameter.

13.1.11 Material and lubrication

13.1.11.1 Materials

The materials displayed in the drop-down lists are taken from the materials database. If you cannot find the material you require in this list, you can either select `Own Input` from the list or enter the material in the database (see section "External tables" on page I-122) first. Click the button to the right of the materials drop-down list to open the `Define material, Gear 1(2)` window in which you can select the material you require from the database list of available materials. Select the `Own Input` option to enter specific material characteristics. This option corresponds to the `Create a new entry` window in the database tool.

Strength calculation with unusual materials:

The cylindrical gear strength calculation formulae defined in ISO 6336, DIN 3990 or AGMA 2001 only involve specific (most commonly used) materials and treatment methods. These are:

- Heat treatable steel
- Case hardening steel
- Nitriding steel
- Structural steel
- Gray cast iron with spheroidal graphite
- Cast iron with flake graphite

Materials not included in the strength calculation standards:

- Stainless steel
- Free cutting steel
- Aluminum and bronze alloys

KISSsoft handles these materials in the same way as heat treatable steels. This affects some of the less important values that are used to calculate the permitted tooth root and flank resistance (e.g. the support value). The maximum possible error is minimal.

Plastics

The strength of plastic gears is calculated in accordance with Niemann or VDI 2545. The permitted demands and Young's modulus used for plastics are largely dependent on the temperature and lubrication type. As a consequence, calculating the characteristics of plastics requires a great deal of time and effort. At present, there are only a few reliable values that can be applied solely to the following materials:

- POM, PA12, PA66
- Laminated fabric
- Plywood

You can add additional materials quite easily because the specific data is already present in files (the file name is present in the materials database). As yet no really reliable data is available for the new generation of plastics (such as fiber-reinforced and other plastics), even from the manufacturers.

It takes a significant amount of time and effort to determine all the data for calculating the strength of plastics. For this reason, you can also enter plastics with a limited amount of data in the database.

For this reason, the comment you input for gear strength data for all plastics can state which data is present and therefore which type of calculations can be performed.

The entry has this format:

[SBFoFgFdWoWgWd]

Abbreviations:

S data for the static root strength calculation is present

B Woehler lines for calculating the root endurance limit (VDI) are present

F Woehler lines for all lubrication types for flank endurance calculation (VDI) are present

Fo Woehler lines for oil lubrication for flank endurance calculation (VDI) are present

Fg Woehler lines for the grease lubrication for flank endurance calculation (VDI) are present

Fd Woehler lines for the dry run for the flank endurance calculation (VDI) are present

Fgd means: Woehler lines for grease and dry run for the flank are present, etc.

W Wear coefficients for all lubrication types are present for wear calculation

Wo Wear coefficients for oil lubrication are present for wear calculation

Wg Wear coefficients for grease lubrication are present for wear calculation

Wd Wear coefficients for dry runs are present for wear calculation

NOTE:

When you select a calculation method either in accordance with VDI or Niemann, the root, tooth flank and wear values are calculated automatically, if the relevant data is available for them. However, if data is not present for one or more of these methods, only those calculations for which data is available are performed.

Converting hardness to endurance limit values σHlim, σFlim

When you enter data for your own material, the hardness can be taken for conversion into the endurance limit values σHlim, σFlim. To open the conversion dialog, click the appropriate conversion button next to the input fields for the endurance limit values σHlim, σFlim. The data is converted in accordance with the ISO 6336-5:2003 formula described in section 5.

(The data for forged steels is used for heat treatable steels "not alloyed/through hardened" and "alloyed/through hardened".)

σHlim, σFlim=A*x+B

x: Hardness value in the unit used in the table (depending on the HV or HBW material type)

A,B: Factors for the particular material type and processing. (from Table 1, ISO 6336-5)

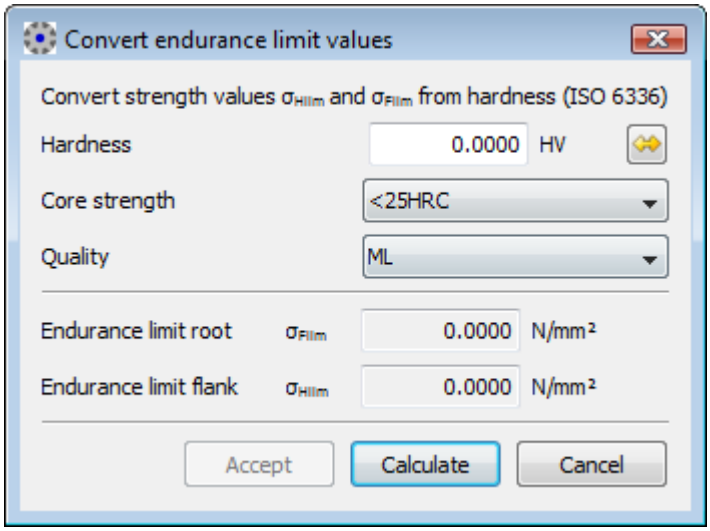


Figure 13.13: `Convert endurance limit values` dialog window

Values for σHlim and σFlim that are required for the conversion specified in ISO 6336-5 are displayed directly in the material screen under "Own Input" if these values are possible with the specified hardness and material type.

In the next conversion dialog, click on another conversion button next to the hardness input field to start converting the hardness value. In the case of materials that are not alloys you can calculate the hardness from the tensile strength value or other hardness values.

13.1.11.2 Calculation of the wear coefficient kw for steel

According to Niemann [65], Table 21.6/5, and Plewe's dissertation (Plewe, H-J: "Untersuchung über den Abriebverschleiss von geschmierten, langsam laufenden Zahnrädern" (Abrasive wear and endurance calculation for lubricated, low-speed gears). Technical University of Munich, 1980) which calculates an approximate guide value for coefficient of wear kw. kw depends on the size of the lubricant gap in the operating pitch circle hc. The function defined by Plewe, kw = f(hmin), is valid for standard mineral oil without high pressure additives.

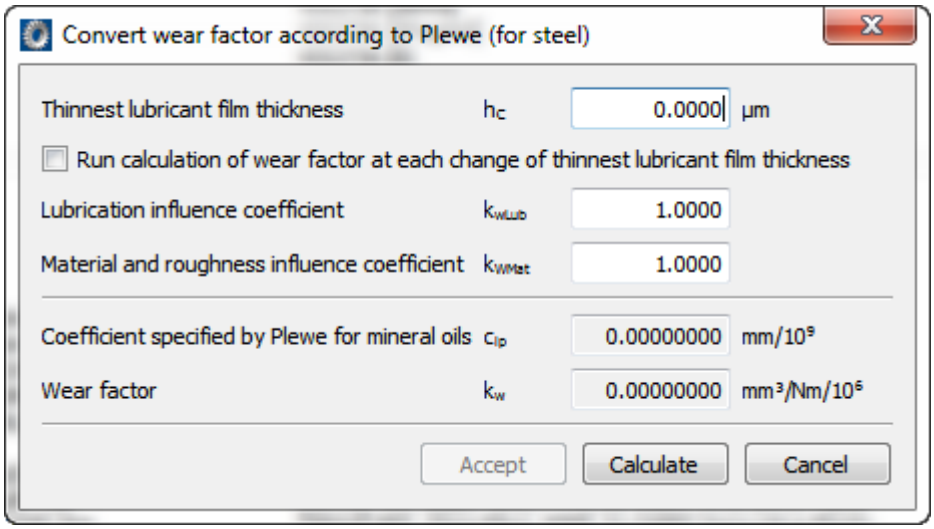


Figure 13: Input window for Proposed value for wear coefficient

You should take care when using this guide value because the existing information is far from complete. In particular, very little is known about the influence of surface roughness and the influence of lubricant additives. You should take careful measurements to check the wear coefficient to ensure reliable results from the calculations.

Influence coefficient of lubricant: As stated in [65], adding suitable additives to a lubricant can significantly reduce the amount of wear. The influence coefficient of the lubricant can therefore lie in a range between 0.01 and 1,000.

Influence coefficient of material: Plewe took measurements from various different material pairings: Gear made of heat treatable steel paired with a hard or soft counter gear, gear pairs made of case hardening steel, and gear pairs made of nitriding steel. The kw as defined by Plewe was then determined for these

combinations. The influence coefficient (if known) can be used for other combinations. For more information see [65].

13.1.11.3 Lubrication

Select the lubricant from a list. If you select Own Input, click the button to specify your own lubricant.

If you see the note (with kw info) after the lubricant description, this means an influence coefficient k_{wlub} is present for this lubricant. This coefficient can then be used to determine the wear factor k_w more accurately.

You can select oil bath or oil spray lubrication, or grease lubrication, or none at all (dry run). You can select dry run only when using a calculation method for plastics.

Click the button to the right of the lubrication type drop-down list to open the `Define temperatures` window (see Figure 13.13)

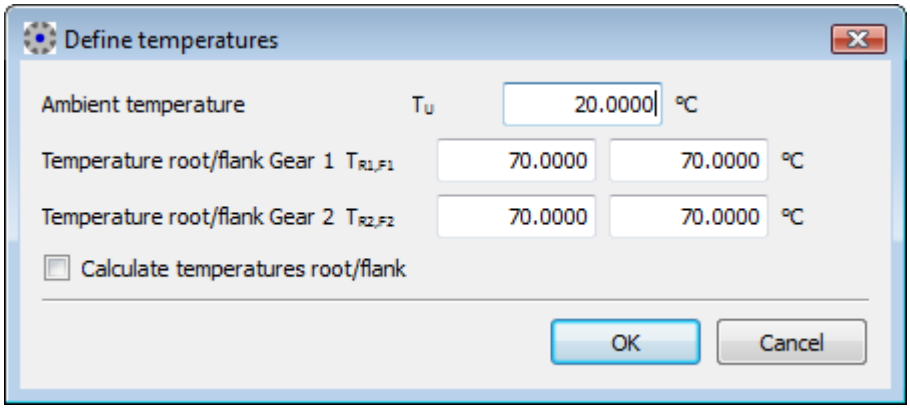


Figure 13.13: `Define temperatures` for `dry run` dialog window

Here you can either specify your own lubricant temperature or enter the root and flank temperatures for a dry run in case of plastics. Usually, these temperatures will be calculated for plastics. However, you can also switch off the calculation and define your own temperatures.

13.2 Load

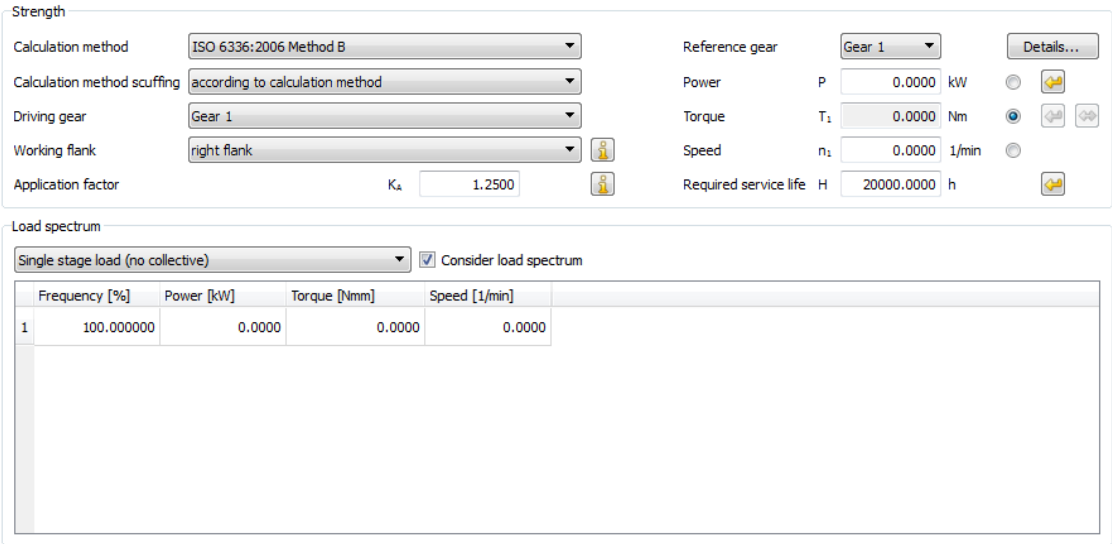


Figure 13.14: `Load` input window for cylindrical gear pair

The `Load` input window is one of the standard (see page I-86) tabs and is subdivided into 2 areas: `Strength` and `Load spectrum`.

13.2.1 Calculation method

In the drop-down list, you can select the following calculation methods:

1. **Geometry calculation only.** If the Rating module is not selected in the Calculation menu, only the geometry is calculated.
2. **Static calculation**. Unlike DIN 743 which, for example, has a specific method for static shaft calculations, ISO 6336 does not have its own calculation method for static calculation. In a static calculation, the nominal stress is usually compared with the permitted material parameters (yield point and/or tensile strength). This runs a static calculation of cylindrical gears in KISSsoft where the nominal stress in the tooth root (calculated by tooth form factor Y_F) is compared with the yield point and tensile strength.

   Each coefficient (application factor, face load factor, transverse coefficient, dynamic factor) is set to 1.0. The load at the tooth root is calculated in accordance with ISO 6336 method B with the tooth form and the helix

angle (without the stress correction factor).

$$\sigma_{F0} = \frac{F_t}{b_{eff} \cdot m_n} \cdot Y_F \cdot Y_\varepsilon \cdot Y_\beta \tag{12.1}$$

$$\sigma_F = K_A \cdot K_V \cdot K_{F\alpha} \cdot K_{F\beta} \cdot \sigma_{F0} \tag{12.2}$$

It also calculates the local tooth root stress multiplied by the stress correction factor Y_S. This stress is approximately the same as the normal stress calculated in an FEM model. This stress is then also output in the report:

$$\sigma_{F0} = \frac{F_t}{b_{eff} \cdot m_n} \cdot Y_F \cdot Y_S \cdot Y_\varepsilon \cdot Y_\beta \tag{12.3}$$

3. **ISO 6336:2006 method B** (Calculation of load capacity of spur and helical gears). Method B is used for this calculation.
4. **DIN 3990 method B** (Calculation of load capacity of cylindrical gears). This calculation is also performed using method B. However, either method B or method C can be used to calculate the tooth form factor (we recommend method C for internal meshings; otherwise, use method B).
5. **DIN 3990 method B** (YF method C).
   (See DIN 3990, method B)
6. **DIN 3990 Part 41 (Vehicle gearbox) method B** (Load capacity calculation for vehicle gearboxes). Method B is used for this calculation. You must enter two application factors (see page II-272) to accurately represent load spectra.
7. **AGMA 2001-B88**. (See AGMA 2001-C95)
8. **AGMA 2001-C95**. This edition of the AGMA 2001-C95 American national calculation guideline replaces AGMA 2001-B88. The previous version of the AGMA guideline has been retained because many companies still use old AGMA guidelines. In fact, there are very few differences between the old edition of B88 and the new C95 edition. However, the new edition does include the service factor calculation.

The standard is implemented in its complete form and the dynamic factor and the face load factor are calculated in accordance with AGMA recommendations. The geometry factors (for tooth root and flank) are calculated entirely in accordance with ANSI/AGMA 908-B89.

In addition to all the relevant intermediate results, the following values are also supplied:
Pitting Resistance Power Rating, Contact Load Factor, Bending Strength Power Rating, Unit Load for Bending Strength, Service Factor.

This calculation can also be used for every other cylindrical gear configuration (including planetary stages). However, it is remarkable that AGMA Standards do not permit the direct calculation of tooth root strength in internal gear pairs. In this case the calculation must be performed using the graphical method (see page II-286).

9. **AGMA 2001-D04**.
   Most recent version of AGMA 2001. Differs only slightly from the previous version C95.
10. **AGMA 2101-D04**. (Metric Edition)
    Equivalent to AGMA 2001-D04, but all values in SI units.
11. **Special AGMA standards: 6004-F88, AGMA 6014-A06, AGMA 6011-I03**
    Special standards used in the USA to calculate the strength of open gear rims. These calculation methods are based on the AGMA 2001 or 2101 basic standards. However, some factors have been specifically defined for special applications.
    AGMA 6014 replaces the old AGMA 6004 standard, but both methods are still available because AGMA 6004 is still requested separately.
12. **AGMA 6011-I03: For turbo drives (High Speed Helical Gear Units) and API 613**
    The AGMA 6011 standard is a special edition for high speed drives and is less complex than AGMA 2001 (or the metric AGMA 2101) base standards. In this case, less complex means that some data is already predefined. For example, to define the face load factor, AGMA 2001 has the options "Open gearing", "Commercial gear unit" and "Precision gear unit" whereas AGMA 6011 has "Precision gear unit" as a predefined requirement. In addition, AGMA 6011 also has information to help you select the application factor KA for specific turbo-driven applications and other useful notes about this type of gear (lubrication arrangement etc.). It is therefore always possible to perform the calculation according to AGMA

6011 using AGMA 2001 or 2101 without causing any problems. To input data correctly for AGMA 2001, as implemented in KISSsoft, that is also correct for AGMA 6011 you must be aware of the constraints and take them into consideration when entering the parameters. Select the AGMA 6011 method to save the user having to do this. In this situation, the program checks whether all the constraints are set and, if not, queries the user to see if they want to make any modifications.

Calculation in accordance with API613 (Special Purpose Gear Units for Petroleum, Chemical and Gas Industry Services, 2003). API613 states that the calculation must be performed in accordance with AGMA 6011. However, this also involves a number of special features. To perform the calculation correctly, you must use our information sheet which describes the necessary inputs and checks: kisssoft-anl-078-E-CylindricalGears API613.docx. The values required by API613 such as flank load K or the permitted value Ka, bending load S and the permitted value Sa, as specified in Appendix J of API613, are documented.

13. **Plastic as defined in Niemann**
    Please refer to [65] and calculation method No. 13 to see the differences.

14. **Plastic as defined in VDI 2545 (YF, method B)** (thermoplastic materials used in gears). This directive defines how calculations are performed on gears made of plastic or combinations of plastic and steel. The calculation methods used for plastics pay particular attention to the fact that these materials are very sensitive to extremes of temperature. The types of lubrication used here include oil, grease or none at all (dry run). The acceptable load for each material is calculated from figures in data tables whilst taking into consideration the local temperatures at the tooth flank and root as well as the number of load cycles. The local temperature can be calculated when grease is used as the lubricant or during a dry run. However, when oil is used as the lubricant, the oil temperature is used as the local temperature. The calculation is performed for combinations of plastic/plastic and also steel/plastic. The acceptable deformation is also checked. KISSsoft supplies data for the following materials:

- Plywood
- Laminated fabric
- Polyamide (PA12, PA66)
- Polyoxymethylene (POM)

All the specific properties of each material are stored in text tables to allow for the integration of own materials (see page I-114). Strength calculations for

plastics can be performed in accordance with Niemann [66] or VDI 2545 (1981)[3] (tooth form factor using Method B or C). You can also use the modified calculation method as detailed in VDI 2545. This calculates the stress using the tooth root stress correction factor Y_s. The major differences between the two methods are:

Root	**Niemann**	**VDI 2545**	**VDI 2545-mod.**
Y_F	C	B or C	B or C
Y_S	DIN 3990	1.0	DIN 3990
Y_ε	1.0 [8)]	$1/\varepsilon_\alpha$ [7) 9)]	$1/\varepsilon_\alpha$ [7) 9)]
Y_β	1.0	DIN 3990 [10)]	DIN 3990 [10)]
σ_{FE}	$2 * \sigma_{Flim}$	σ_{Flim}	$2 * \sigma_{Flim}$

Table 13.3: Differences between the various calculation methods used for plastics and tooth root

Flank	**Niemann**	**VDI 2545**	**VDI 2545-mod.**
Z_ε	1.0	DIN 3990	DIN 3990
Z_V	DIN 3990 [5) 10)]	1.0	1.0
Z_R	DIN 3990 [6) 10)]	1.0	1.0

Table 13.4: Differences between the various calculation methods used for plastics and tooth flank

Tooth deformation: Very different calculation methods!
[5)] For plywood only, otherwise 1.0
[6)] For steel/plastic combinations only, otherwise 1.0
[7)] For tooth form factor Y_F as defined in method B: 1.0
[8)] the method sets the face contact ratio for the tooth root stress to the value 1.0. According to Niemann, this is because the material data is not always precise. The formulae used in VDI 2545 correspond to those used in ISO 6336:1996.
[9)] For crossed helical gears = $0.25 + 0.75/\varepsilon\gamma$
[10)] For crossed helical gears = 1.0

[3] *Calculation method VDI 2545 has been withdrawn because it has not been revised as required. A new calculation guideline, VDI 2736, is currently being developed. We recommend the "VDI 2545-mod." method is used until the new standard is launched. Its announced publication date is 2014. We are unaware of a better version.*

14. **Plastic as defined in VDI 2545 (YF, method C)**.
    In this calculation method, the tooth form factor Y_F is calculated in accordance with method C.
15. **Plastic as defined in VDI 2545-modified (YF, Method B).**
    This method is recommended for plastics with normal toothing. Transverse contact ratio $\varepsilon_\alpha < 1.9$. See table in 13.4. for the differences between VDI and VDI modified.
16. **Plastic in accordance with VDI 2545-modified (YF, Method C).**
    This method is recommended for plastics with deep toothing. Transverse contact ratio $\varepsilon_\alpha > 1.9$. See table in 13.4. for the differences between VDI and VDI modified. See table in 13.4. for the differences between VDI and VDI modified. In this calculation method, the tooth form factor Y_F is calculated in accordance with method C.
17. **As in FVA program (DIN 3990)**. Supplies the same results as the FVA (Forschungsverein Antriebstechnik: German Research Society for Transmission Techniques) Reference Program. Based on DIN 3990 method B with minor differences.
18. **BV/Rina FREMM 3.1 Naval Ships and Rina 2010 (ISO 6336)**
    Calculation guidelines for ships' engines.
19. **DNV41.2, Calculation standard for ships' engines**
    The Det Norske Veritas calculation guideline [93] for ships' engines corresponds in principle to ISO6336 (root, flank) and ISO 13989 (scuffing). However, it does have some significant differences, especially where Woehler lines are concerned. These differences are detailed our kisssoft-anl-076-DE-Application_of_DNV42_1.pdf information sheet, which is available on request.
20. **Calculation of the safety of the hard surface layer**
    The calculation specified in DNV41.2 [93] is also performed automatically in every strength calculation method if a hardening depth has been specified (hardening depth (see page II-284)).

13.2.2 Service life

Enter the required service life directly in the input field.

Click the button to size this value. Based upon the minimum safety value for the tooth root and flank strength, this process calculates the service life (in hours) for every gear and for every load you specify. The service life is calculated in

accordance with ISO 6336-6:2006 using the Palmgren-Miner Rule. The system service life and the minimum service life of all the gears used in the configuration is displayed. You can size the service life using the button either with or without defining a load spectrum (see section "Define load spectrum" on page II-287).

13.2.3 Application factor

The application factor compensates for any uncertainties in loads and impacts, whereby $K_A \geq 1.0$. Table 13.5. illustrates the values that can be used for this factor. You will find more detailed comments in ISO 6336, DIN 3990 and DIN 3991.

When deciding which application factor should be selected, you must take into account the required safety values, assumed loads and application factor in one context.

Operational behavior of the driving machine	**Operational behavior of the driven machine**			
	uniform	**moderate shocks**	**medium shocks**	**heavy shocks**
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 13.5: Assignment of operational behavior to application factor

DIN 3990, Part 41 (car gearboxes), distinguishes between application factors for flank strength K_{AH} and for tooth root strength K_{AF}. Except for flank strength calculations, all other calculations (e.g. resistance to scoring) use application factor K_{AF}.

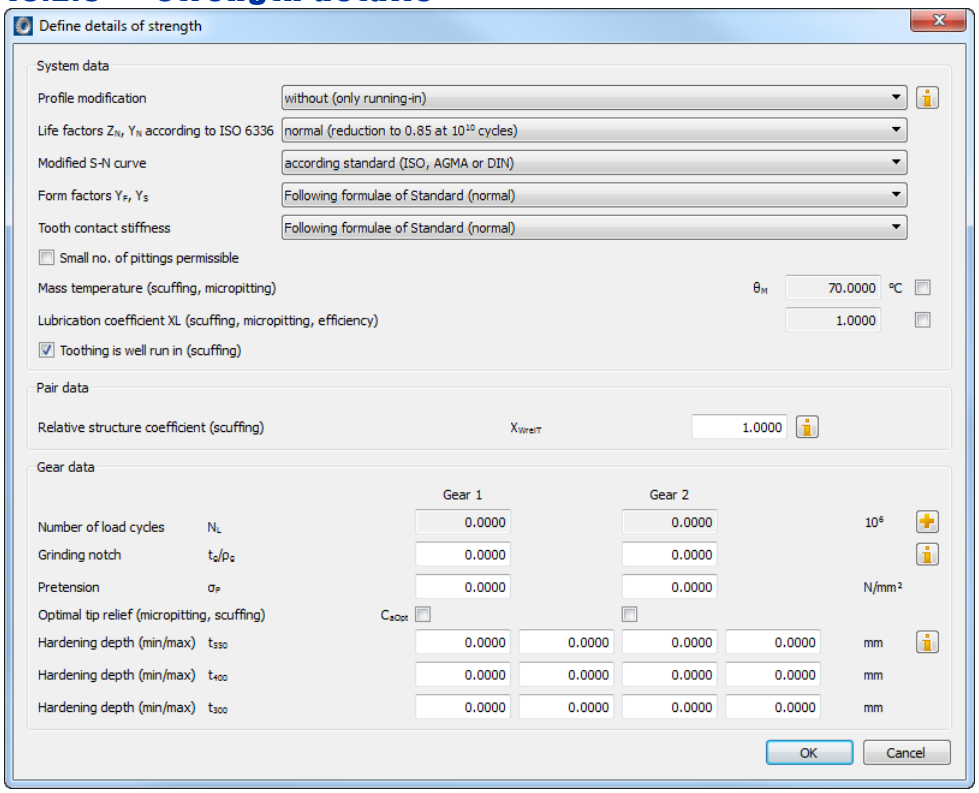
However, in accordance with DIN 3990 Part 41, the application factor can also be less than 1.0. This is intended to avoid the need to perform a calculation involving a load spectrum. For example, DIN 3990, Part 41, Appendix A, suggests the following values for a 4-speed car gearbox:

Gear	**R**	**1**	**2**	**3**	**4**
N_L	10^5	$2 * 10^6$	$1.5 * 10^7$	$3 * 10^7$	$2 * 10^8$
K_{AH}	0.65	0.65	0.65	0.65	
K_{AF}	0.70	0.70	0.80	0.80	

13.2.4 Power, torque and speed

Click the button next to the power input field (for the torque) to calculate the power (torque) so that a predefined safety minimum (see page II-441) can be maintained.

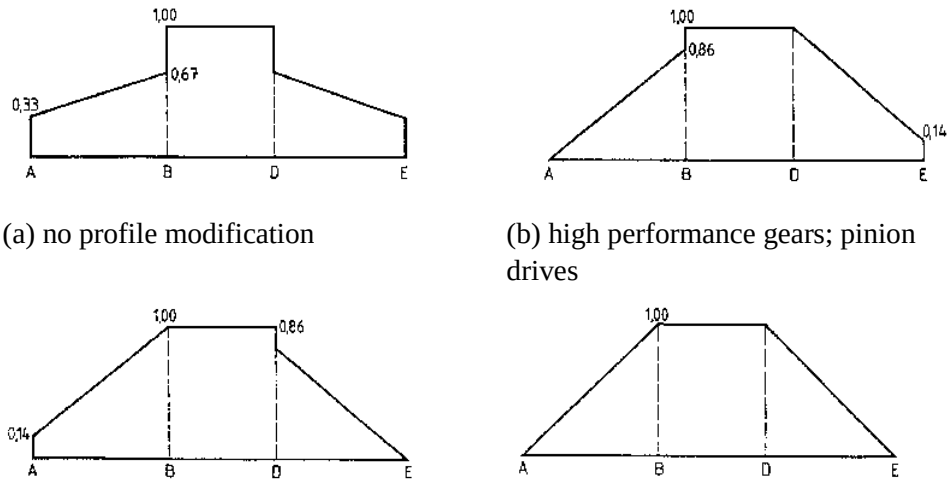
13.2.5 Strength details



Click on the `Details...` button to open the `Define details of strength` window which is divided into `System data`, `Pair data` and `Gear data`. Please note that the window layout used for calculations in accordance with AGMA (see page II-286) is different.

13.2.5.1 Profile modification

You can modify the theoretical involute in high load capacity gears by grinding/polishing the toothing. You will find suggestions for sensible modifications (for cylindrical gears) in KISSsoft Module Z15 (see section "Modifications" on page II-326). The type of profile modification has an effect on how scuffing safety is calculated. The load sharing factor X_Γ is calculated differently according to the type of profile modification used. The main difference is whether the profile has been modified or not. However, the differences between `for high load capacity` and `for smooth meshing` are relatively small. The strength calculation standard presumes that the tip relief C_a is properly dimensioned but does not provide any concrete guidelines. The resulting load sharing factor X_Γ in accordance with DIN 3990, depends on the type of profile modification:



(a) no profile modification

(b) high performance gears; pinion drives

(c) high performance gears; gear drives

(d) balanced meshing

Figure 13.9: Load sharing factor X_Γ for different profile modifications

13.2.5.2 Lifetime factors as defined in ISO 6336

The fatigue limit factor Z_{NT} reduces the permitted material stress in accordance with ISO 6336-2:2006:

$$\sigma_{H\lim red} = Z_{NT} \cdot \sigma_{H\lim} \tag{12.14}$$

$$\sigma_{F\lim red} = Y_{NT} \cdot \sigma_{F\lim} \tag{12.15}$$

As stated in ISO 6336, this value is important for cylindrical gear calculations and is the reason for the lower safety values in the range of endurance limit when compared with DIN 3990.

1. **normal (reduction to 0.85 at 10^{10} cycles):** The permitted material stress in the range of endurance limit (root and flank) is reduced again. Fatigue strength factors Y_{NT} and Z_{NT} are set to 0.85 for $\geq 10^{10}$ load cycles.
2. **increased with better quality (reduction to 0.92):** Y_{NT} and Z_{NT} at $\geq 10^{10}$ load cycles are set to 0.92 (in accordance with the data in ISO 9085).
3. **with optimum quality and experience (always 1.0):** This removes the reduction and therefore corresponds to DIN 3990. However, this assumes the optimum treatment and monitoring of the materials.

13.2.5.3 Form factors

The tooth form factor Y_F takes into account how the tooth form affects the nominal tooth root stress σ_{F0}. The stress correction factor Y_S takes into account the effect of the notch on the tooth root. These two factors can be calculated in three different ways:

1. **In accordance with the formulae in the standard (normal)**
   As defined in ISO 6336 or DIN 3990, the tooth form and the stress correction factors are calculated at the tooth root at the point at which the tangent and the tooth center line form an angle of 30°. However, it is generally acknowledged that this method is rather imprecise, for deep toothings in particular.
2. **Using graphical method**
   According to Obsieger [68], there is a more precise approach in which the product of the tooth form factor Y_F and the stress correction factor Y_S is calculated and the maximum value is determined. This method is based on the production procedure used for a specific tooth form and is applied to all points in the whole root area. This maximum value is then used in calculating the strength. Factors Y_F and Y_S are calculated in accordance with the formulae in ISO 6336 or DIN 3990.

   This is the recommended method, particularly for unusual tooth forms and internal teeth. If required, this calculation procedure can also be applied in strength calculations as defined in ISO 6336 and DIN 3990, as well as in fine sizing.

   Note:
   If you use the graphical method here, KISSsoft will calculate the tooth

form before it calculates the strength, each time. It takes its parameters either from the cutter data you entered previously in the `Tooth form` (see section "`Gear tooth forms`" on page II-574) input window or from the default settings in the `Reference profile` input window. The maximum value of the product of the tooth form and stress modification factor is calculated at the same time and included in the strength calculation.

$$Y_F(r) = \frac{6h_F(r)\cos\alpha_F}{[s_F(r)]^2\cos\alpha_n}$$
$$q_s = 0{,}5\, s_F(r)/r_F(r)$$
$$a = 1{,}2 + 0{,}13[s_F(r)/h_F(r)]$$
$$b = 1{,}21 + 2{,}3/[s_F(r)/h_F(r)]$$
$$Y_S(r) = a \cdot q_s^{(1/b)}$$

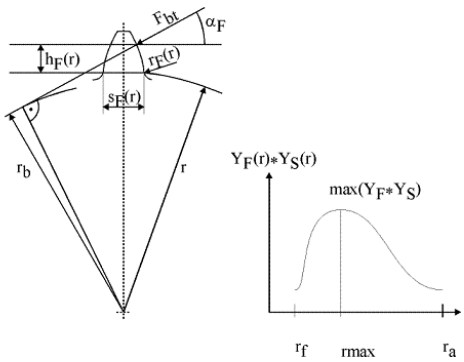


Figure 13.15: Form factors using graphical method

3. **for internal toothing, in accordance with VDI Proposal 2737**
When calculating strength in accordance with ISO 6336 or DIN 3990, selecting this option allows you to use the tooth form factor as defined in VDI 2737, which is more precise for internal teeth, because it evaluates the stress at the point of the 60° tangent and derives the tooth form from the manufacturing process with the pinion type cutter.

The tooth root stress calculation specified in ISO 6336 is more accurate than the one implemented in DIN 3990. However, the calculation applied to the root rounding in the critical point (for a 60° tangent) is still incorrect. The method defined in VDI 2737, Appendix B is much more accurate, which is why we recommend you use this method. If you select this option, only the root rounding ρF and the root thickness sFn in the critical cross section is calculated in accordance with the formulae in 2737. All other sizes are calculated in accordance with ISO 6336.

The table (below) uses 4 examples to show the large variations that still arise in root rounding between the result defined in ISO 6336 and the effective values measured on the tooth form. However, the calculation method stated in 2737 is very suitable for this.

Gear x=	Pinion	ρF in ISO	ρF in ISO 6336-	ρF measured	ρF with VDI

	Cutter x0=	6336-3 2006 and 2007-02	3 2007-04	on the tooth flank	2737
-0.75	0.1	0.201	**0.426**	**0.233**	**0.233**
-0.75	0.0	0.175	**0.403**	**0.220**	**0.220**
0.0	0.1	0.298	**0.364**	**0.284**	**0.286**
0.0	0.0	0.274	**0.343**	**0.265**	**0.264**

Table13.10: Comparison of root roundings

Note about calculating YF:

- The theoretical profile shift is used for the calculation of the allowance is As < 0.05*mn (in accordance with ISO 6336-3). Otherwise the larger manufacturing profile shift xE.e is used. This corresponds to the procedure used in the STplus program (from Munich, Germany). An exact definition is not provided in the ISO standard. However, if this is specified in Settings `Strength calculation using mean position in tolerance field (of tooth form)`, the calculation will always be performed with the average manufacturing allowance xE.m.
- According to the ISO standard, the reference profile for the entire toothing is to be used for the calculation. For this reason, if you input the reference profile for preliminary treatment with protuberance, and a manufactured profile with remaining protuberance is left after deduction of the grinding allowance, the reference profile for final treatment is used for the calculation. In the case of the reference profile for preliminary treatment without a protuberance (or a protuberance that is too small), a grinding notch is produced. To ensure that this situation can be correctly taken into consideration the preliminary treatment reference profile (with preliminary treatment manufacturing profile shift) is used to calculate YF. Furthermore, the final treatment reference profile is used to calculate the grinding notch and therefore define YSg (section 7.3 in ISO 6336-3).

13.2.5.4 Tooth contact stiffness

Tooth contact stiffness is required to calculate the dynamic factor and the face load factor. You can use one of these calculation options:

1. **In accordance with the formulae in the standard (normal)**
   In the standard calculation, the tooth contact stiffness c_g is calculated using empirical formulae (in ISO 6336, DIN 3990, etc.).

2. **Using the tooth form**
   In this option, the tooth form stiffness c' is calculated in accordance with Weber/Banaschek's dissertation [69]. This takes into consideration tooth bending, basic solid deformation, and Hertzian pressure. The last condition determines the load dependency of c'. The contact stiffness is determined using the effective tooth form (see Meshing stiffness (Z24)). The mean value of the stiffness curve that is calculated using this method is then included in the calculation. If required, this calculation procedure can also be applied in strength calculations as defined in ISO 6336 and DIN 3990, as well as in fine sizing (Z04). The single spring stiffness c' is calculated from the c_g, by extrapolating c' from the formula for c_g (ISO or DIN).
3. **constant (20 N/mm/μm)**
   In this option, the tooth contact stiffness constant is replaced by

$$c_g = 20 N / mm / \mu m$$

13.2.5.5 Small no. of pittings permissible

In specific cases, the appearance of a small number of pittings on the flank may be permissible. In a range of materials this results in higher flank safeties in fatigue strength range due to the changed S-N curve (Woehler line), as can be seen in either ISO 6336-2, Figure 6, curve 1 or DIN 3990-2, Figure 8.1.

13.2.5.6 Lubrication coefficient

The lubricant coefficient is needed to calculate the friction factor, loss, micropitting and scuffing.

These values are specified in ISO15144:

- 1.0 for mineral oils
- 0.6 for water-soluble polyglycols
- 0.7 for non-water-soluble polyglycols
- 0.8 for polyalphaolefins
- 1.3 for phosphate esters
- 1.5 for traction fluids

13.2.5.7 Relative structure coefficient (scuffing)

The relative structure phase coefficient takes into account differences in materials and heat treatment at scoring temperature. However, the standards do not provide any details about how to proceed when different types of material have been combined in pairs. You must input this coefficient yourself because it is not set automatically by KISSsoft.

Relative structure phase coefficient as defined in DIN 3990, Part 4:

Through hardened steels	1.00
Phosphated steel	1.25
Coppered steel	1.50
Nitrided steel	1.50
Case-hardened steels	1.15 (with low austenite content)
Case-hardened steels	1.00 (with normal austenite content)
Case-hardened steels	0.85 (with high austenite content)

Stainless steels	0.45

The standard does not provide any details about how to apply the coefficient when the pinion and gear are made of different material types. In this case it is safer to take the lower value for the pair.

13.2.5.8 Number of load cycles

KISSsoft calculates the number of load cycles from the speed and the required service life. If you want to influence the value, you can define it in the `Number of load cycles for gear n window`. Click the button to access this. In this window you can select one of five different options for calculating the number of load cycles.

1. **Automatically** The number of load cycles is calculated automatically from the service life, speed, and number of idler gears.
2. **Number of load cycles** Here you enter the number of load cycles in millions. You must select this option for all the gears involved in the calculation to ensure this value is taken into account.
3. **Load cycles per revolution** Here you enter the number of load cycles per revolution. For a planetary gear units with three planets, enter `3` for the sun and `1` for the planets in the input field.

   Note:
   If the `Automatically` selection button in the calculation module is enabled, KISSsoft will determine the number of planets in the `Planetary stage calculation` module.
4. **Load cycles per minute** Here you enter the number of load cycles per minute. This may be useful, for example, for racks or gear stages where the direction of rotation changes frequently, but for which no permanent speed has been defined.
5. **Effective length of rack** The rack length entered here is used to calculate the number of load cycles for the rack. The rack length must be greater than the gear's perimeter. Otherwise, the calculation must take into account that not every gear tooth will mesh with another. You must enter a value here for rack and pinion pairs. Otherwise the values N_L(rack) = N_L(pinion)/100 are set.

NOTE

This calculation method is used for transmissions with a slight rotation angle.

This scenario assumes that a reduction is present

$$i = \frac{z_2}{z_1}$$

and a pivoting angle w in [°] from gear 2, where gear 2 constantly performs forwards and backwards movements by the angle value w. The effective endurance is given as the service life. The two coefficients N_1 and N_2, which reduce the absolute number of load cycles, N_L, are now calculated. To do this:

a) Set the alternating bending factor of the pinion and gear to 0.7, or calculate it as defined in ISO 6336-3:2006. In this case, a complete forwards/backwards movement is counted as a load cycle.

b) For the pinion, coefficient N_1 is determined as follows:

$$N_1 = 0.5 \qquad \text{für } w \cdot i > 360$$

$$N_1 = 0.5 \cdot \max(1.0, 360/w/i) \qquad \text{für } w \cdot i \leq 360$$

c) The number of load cycles of teeth in contact in gear 2 is smaller by a coefficient of N_2 when compared with the number of load cycles during continuous turning.

$$N_2 = 0.5 \cdot 360/w$$

Factor 0.5 takes into account both the forwards and backwards movements.

d) Enter coefficients N_1 and N_2 in the `Load cycles per revolution` input field.

The correct number of load cycles can now be calculated on the basis of the data entered in steps a to d.

13.2.5.9 Grinding notch

As defined in DIN 3990 or ISO 6336, the effect of the grinding notch can be taken into account by the coefficient Y_{Sg}. Here, you input the ratio t_g to the radius of grinding notch ρ_g in accordance with the figure in DIN3990-3, Section 4.4 or ISO 6336-3, Figure 5. KISSsoft then calculates the coefficient $Y_g = Y_{Sg}/Y_S$ (a factor, which was multiplied with Y_S).

The distance between the 30_o tangents for the initial and final contour is used as the grinding notch depth t^g. If a preliminary treatment allowance has been entered in KISSsoft (see Figure 13.11), you can no longer enter the ratio t_g/ρ_g. It is calculated by the software instead. A grinding notch occurs when a grinding depth (see section "Modifications" on page II-326) was entered and no protuberances remain, either because no protuberance tool was used, or the selected allowance was too small. The fillet radius ρ_g is then calculated by generating the grinding wheel at the 30° tangent (or, for internal teeth, at the 60° tangent).

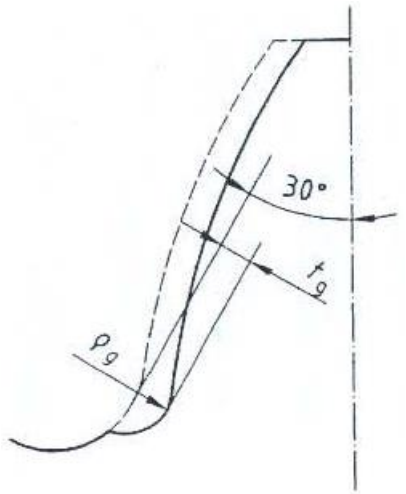


Figure 12.11: Grinding notch

13.2.5.10 Pretension

The influence of a press fit or other processing methods that influence tooth root stress can be taken into account with the pretension σ_P. This value influences the calculated tooth root stress as well as the safety in accordance with the following formulae:

For static strength:

$$\sigma'_F = \sigma_F + \sigma_P$$

$$S'_S = \frac{R_P}{\sigma'_F}$$

$$S'_B = \frac{R_m}{\sigma'_F}$$

For fatigue strength:

$$\sigma'_{FG} = \sigma_{FG} \cdot \left(1 - \frac{\sigma_P}{R_m}\right)$$

$$S'_F = \frac{\sigma'_{FG}}{\sigma_F}$$

The pretension σ_P merely generates additional results in the reports. The results in the results window remain unchanged. You define this under "Strength" > "Details".

13.2.5.11 Optimal tip relief

To calculate safety against micropitting as specified by Method B in ISO 15144, you must specify whether or not the profile modification is to be assumed to be optimal. The same applies to calculating the safety against scuffing. The software checks whether the effective tip relief (Ca) roughly corresponds to the optimum tip relief (Ceff). If this check reveals large discrepancies, i.e. Ca < 0.333*Ceff or Ca > 2.5*Ceff, a warning is displayed. In this case, the value you input is ignored and documented accordingly in the report.

13.2.5.12 Hardening depth, known by its abbreviation "EHT"

You can input the intended hardening depth (for hardness HV400, for nitrided steels, or HV550 for all other steels). You can also input the hardness HV300. This value is then used to display the hardening curve as a graphic. The input applies to the depth measured during final treatment (after grinding).

When you input this data, the safety of the hardened surface layer is calculated automatically according to DNV 41.2 [93]. Here a minimum value of t400 (nitrided steel) or t550 (all other steels) is used. If only the value for HV300 is known, this value is then used. However, the calculation should then only be seen as an indication. The calculation is performed as described in the section in [93] "Subsurface fatigue". The values required to define the EHT coefficient YC as specified in DNV 41.2 are also needed. The calculation is performed using different solutions than the calculation of the proposal for the recommended hardening depth, but still returns similar results (proposal for hardening depth). To achieve a proposal for a sensible hardening depth, we recommend you call the calculation in `Report -> Proposals for hardening depth`. A maximum value for the hardening depth is only used to check the hardening depth at the tooth tip. It is mainly used for documentation purposes.

13.2.5.13 Modification of S-N curve (Woehler line) in the range of endurance limit

In a standard Woehler diagram, the range of endurance limit is reached at a particular number of load cycles. From this point onwards, the dynamic strength no longer changes even when the number of load cycles increases. This behavior is called "according to Miner".

However, more recent investigations have revealed that there is actually no such thing as an endurance limit and that the S-N curve (Woehler line) should be modified in the endurance limit range.

In the range of endurance limit, you can therefore select the following modified forms:

- Miner (corresponds to DIN 3990, Parts 2, 3 and 6)
- According to Corten/Dolan
- According to Haibach

Figure 13.73 shows the relevant characteristics. Experience has shown that performing a service life calculation with load spectra using the Miner method returns results that are far too optimistic. We recommend you use the Haibach method of approach.

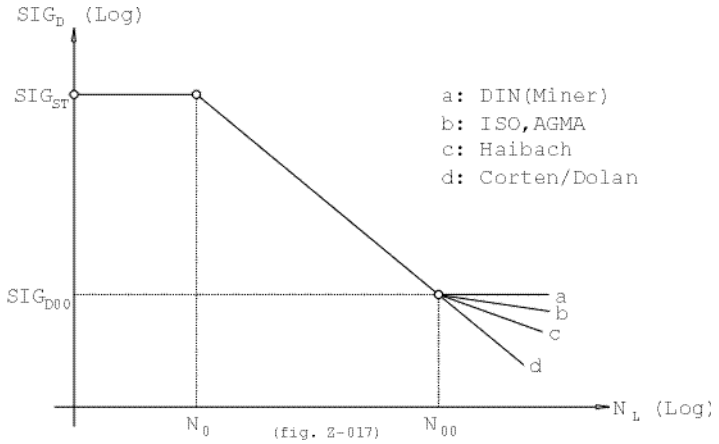


Figure 13.73: Endurance limit models

Note concerning calculations according to ISO or DIN:

The slope of the S-N curve (Woehler line) for bending in the time-dependent domain (between N0 and N00) is defined using the YNT, YdrelT, YRrelT and YX factors for the static and endurance cases, but in the endurance domain (NL > N00), only the YNT factor is used for the static and endurance cases. The same applies to pitting with the ZNT, ZL, ZV, ZR and ZW factors. This corresponds to the method used in ISO 6336 for the endurance domain. However, this does mean that buckling occurs on the S-N curve (Woehler line) at N00, according to the Cortan/Dolan rule.

As an example: for case-carburized steel the slope of the S-N curve (Woehler line) in the endurance domain is 13.22, but in the time-dependent domain it is approximately 10, depending on the precise values for YdrelT, etc.

If all the factors, YdrelT, etc., are set to 1.0 using "Own input" then there will be no buckling of the S-N curve (Woehler line) at N00.

13.2.6 Strength details (AGMA)

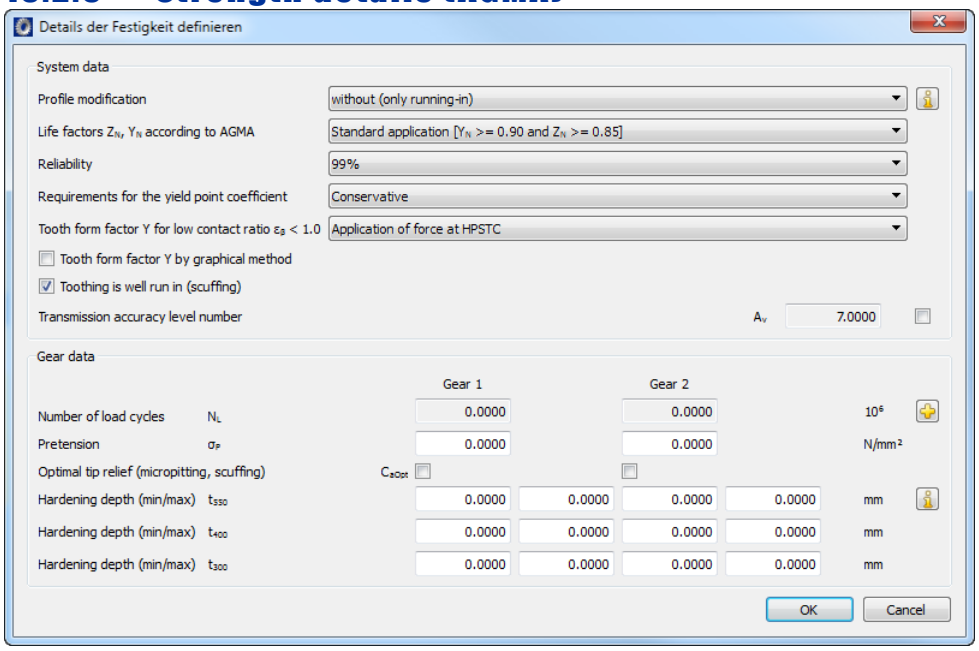


Figure 13.12: `Define details of strength` input window for calculating strength as defined in AGMA

NOTE

Only values in the input window that differ from those defined in ISO are described here.

13.2.6.1 Lifetime factors

The endurance limit factors determine which material values can be entered in the field for limited time and strength. In standard applications, endurance strength values up to 10^{10} load cycles are reduced from 100% to 90% for the root and to 85% for the flank. As stated in AGMA, the reduction in strength also extends beyond 10^{10} load cycles. In critical application areas, where a gear breakdown must be prevented at all cost, the material values are further reduced in comparison to those used in standard application areas.

13.2.6.2 Form factors

For cylindrical spur gears, or spur gears with low helix angles, you can specify that the load is to be applied either at the tip or at a single meshing point (the more precise option). For cylindrical gears with a large helix angle ($\varepsilon\beta \geq 1$) in

accordance with AGMA the force is always applied to a single meshing point (HPSTC).

Calculating with the HPSTC results in a lower load at tooth root because the load is divided between the two teeth. However, if large single pitch deviations occur, this load distribution does not take place and therefore the force should be assumed to be placed at the tooth tip.

As stated in AGMA, the contact point between the tooth form and the Lewis parabola is selected as the critical root cross section. The stresses are determined here. AGMA does not provide a formula for calculating internal teeth. Instead, it recommends you use the graphical method to calculate the tooth form. The required data is to be taken from measurements. If you click the checkbox to select the graphical method of calculating the tooth form factor, the software automatically calculates the tooth form at the point where the K_f or I factor is greatest. In contrast to the method defined by Lewis, where the calculation is only performed at the contact point of the parabola, the calculation using the cross section with the greatest stresses gives more precise results and is therefore the method we recommend for external gears too.

13.2.6.3 Transmission accuracy level number

A_V (or Q_V for AGMA 2001-C95 or earlier) is calculated in accordance with the formulae defined in AGMA 2001 or 2101 and is extremely dependent on the accuracy grade. However, the A_V may be one level higher or less than the accuracy grade and is needed to calculate the dynamic factor. You can overwrite this value if required.

13.2.7 Define load spectrum

Load spectrum

Standardised load spectrum, binominal distribution, p=5/6

☑ Consider load spectrum

	Frequency [%]	Power [kW]	Torque [Nm]	Speed [1/min]	$K_{H\beta}$
1	0.000200	75.0000	1624.7669	440.8000	1.0000
2	0.001600	74.4000	1611.7688	440.8000	1.0000
3	0.028000	73.1250	1584.1477	440.8000	1.0000
4	0.272000	71.5500	1550.0276	440.8000	1.0000
5	2.000000	69.6750	1509.4084	440.8000	1.0000

Figure 13.72: `Load spectrum` group

In this group, you can also access load spectra that have been stored in the database. You can also define the load spectra directly.

If you select `Read`, you can import a file (in either *.txt or *.dat format) with a load spectrum.

The "Example_DutyCycle.dat" file in the `dat` sub-folder in the KISSsoft installation directory is an example of a file that shows how a load spectrum can be defined.

If you want the calculation with load spectra to include separate factors ($K_{H\beta}$, K_{γ}, etc.) for each load spectrum element, you must make the appropriate settings in the `Factors` tab for the load distribution coefficient (on page II-292) K_{γ}, the alternating bending factor (on page II-293) Y_M and the face load factor (on page II-296) $K_{H\beta}$. You will find an example file that shows how a load spectrum with factors ($K_{H\beta}$, K_{γ}, etc.) can be defined in the "Example_DutyCycleWithFactors.dat" file in the `dat` sub-folder in the KISSsoft installation folder.

13.2.7.1 Type of load spectrum

The calculation of service life for load spectra is performed as specified in ISO 6336, Part 6, and is based on the Palmgren-Miner rule.

Here, three load spectra are predefined as shown in DIN 15020 (Lifting Appliances) along with many other standard spectra. You can enter your own load spectra.

A load spectrum consists of several elements (up to 50 in the database or an unlimited number if imported from a file). Each element consists of the frequency, speed, and power or torque. The data always refers to the reference gear you selected when you input the nominal power (Performance- Torque- Speed screen). The program stores these values as coefficients so that they are modified automatically when the nominal power changes.

If two speeds that are not equal to zero have been predefined for planetary stages, you can select two load spectra. In this case, only the speed factor is important for the second load spectrum.

NOTE

The load dependency of the K-factors are included in the calculation (K-factors: dynamic, face load and transverse coefficients). If you want to examine the result in greater detail, you will find the most interesting interim results in the Z18-H1.TMP text file (in the TMP directory).

13.2.7.2 Load spectra with negative elements

Load spectra with negative load spectrum elements (T < 0 and/or n < 0) are calculated as follows (is only applied to elements whose alternating bending factor is YM=1.0).

IMPORTANT:

A load spectrum element is evaluated as negative if the mating flank is placed under load. The same definition is used for the speed and torque (or power) prefix as in KISSsys.

Consequently, the following elements are evaluated as negative:

Factor for torque or power	Factor for speed	Flank under load	Actual load spectrum element
+	+	Working flank (*)	Evaluated as positive
+	-	Working flank (*)	Evaluated as positive
-	+	Mating flank	Evaluated as negative
-	-	Mating flank	Evaluated as negative

(*) Working flank as entered in the Strength tab

Under `Details` in the `Rating` tab, group `Strength`, you can select the following:

For calculating pitting safety

Evaluate all negative load spectrum elements as positive (as up to now)

Only evaluate positive load spectrum elements

Only evaluate negative load spectrum elements

For calculating the tooth root safety:

Evaluate all load spectrum elements (as up to now)

For negative load spectrum elements Root stress with 1/0.7 increase

For positive load spectrum elements Root stress with 1/0.7 increase

13.2.8 Calculate scuffing

The following selection options are available here:

- Corresponding to the strength calculation method

Here, if the DIN strength calculation method is used, scuffing is calculated as defined in DIN 3990-4, for all other calculation methods, scuffing is calculated as specified in ISO TR 13989.

- `Always in accordance with ISO TR 13989`
  Scuffing is always calculated as specified in ISO TR 13989.

- `Always in accordance with DIN 3990-4`
  Scuffing is always calculated as specified in DIN 3990-4.

Depending on which option is selected, the integral temperature and flash temperature are calculated in accordance with the corresponding standard.

13.3 Factors

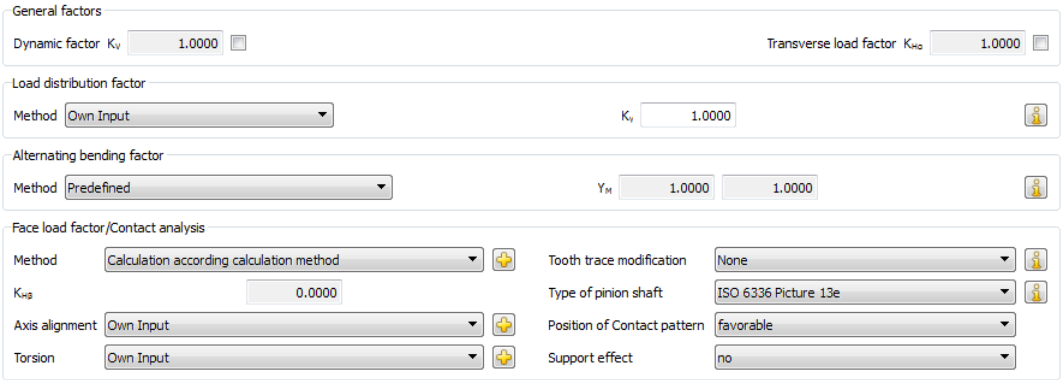


Figure 13.15: `Factors` input window for a cylindrical gear pair

The `Factors` input window is one of the standard (see page I-86) tabs.

13.3.1 Transverse coefficient

The transverse coefficient $K_{H\alpha}$ is calculated in accordance with the calculation method you selected. The transverse coefficient takes into account irregular contact characteristics across a number of teeth. When the contact ratio increases, the transverse coefficient also becomes larger depending on the predefined accuracy grade. A high contact ratio will result in a reduction of the root stresses. Large single pitch deviations, the transverse coefficient will compensate this effect.

In unusual cases, the transverse coefficient will be unrealistically high. If you want to reduce the transverse coefficient in this situation, simply click the checkbox to the right of the input field. You can then change this value.

13.3.2 Dynamic factor

The dynamic factor takes into account additional forces caused by natural frequencies (resonance) in the tooth meshing. It is usually calculated using the calculation method you selected, however you can also input the value if it has already been derived from more precise measurements. To change the value, click the checkbox next to the input field.

13.3.3 Load distribution coefficient

The load distribution coefficient takes into consideration the uneven load distribution across multiple planets or idler gears. In this case the load is multiplied by this coefficient. Dimensioning suggestion in accordance with AGMA 6123-B06:

	Number of planets									
Application Level	2	3	4	5	6	7	8	9	Quality ISO 1328	Flexible Mounting
1	1.16	1.23	1.32	1.35	1.38	1.47	1.60	-	≥ 7	without
2	1.00	1.00	1.25	1.35	1.44	1.47	1.60	1.61	5 ÷ 6	without
3	1.00	1.00	1.15	1.19	1.23	1.27	1.30	1.33	≤ 4	without
4	1.00	1.00	1.08	1.12	1.16	1.20	1.23	1.26	≤ 4	with

Table 13.9: Load distribution coefficient K_γ defined by the number of planets

Level of application	**Description**
1	Typical of large, slow-turning planetary gear units
2	Average quality, typical of industrial gears
3	High quality gears, e.g. for gas turbines

Table 13.10: Meaning of the level of application

NOTE

Level 2, or higher, requires at least one floating element.
Level 3, or higher, requires a flexible gear rim.
In a flexible assembly, the planets must be supported on flexible pins/flexible shafts or on a support with couplings.

The `Calculated in accordance with AGMA 6123` method is used to calculate the load distribution coefficient K_γ for application level 1 ÷ 3 depending on the accuracy grade and the number of planets.

If a different load distribution coefficient is input for each element when load spectra are in use, you should select the `Own input, per load stage` method.

13.3.4 Alternating bending factor

The tooth root strength calculation is designed for pulsating load on the tooth root. However, in some cases, the tooth root is subject to alternating bending loads (e.g. a planet gear in planetary gear units). In this scenario you can change the alternating bending coefficient of individual gears by selecting either the `Own input` or `Own input, per load spectrum element` methods. As an alternative to transferring these values directly, select the `Calculate in accordance with ISO 6336-3 Annex B` method to calculate the coefficient. To do this, you must then open the `Rating` tab, go to the Load spectrum section and input the f_{low} and f_{high} parameters for each gear. f_{high} must always have the fixed default value of 100%.

ISO 6336-5:2003, Section 5.3.3, and DIN 3990-5, Section 4.3, specify the value 0.7 for Y_M for a purely cyclic load. In ISO 6336-3:2006, Annex B, the stress ratio *R* for idler and planetary gears is taken into account by using these formulae:

$$Y_M = \frac{1}{1 - R \cdot \frac{1-M}{1+M}} \tag{12.16}$$

$$R = -1.2 \cdot \frac{f_{low}}{f_{high}} \tag{12.17}$$

f_{high}	Load on the flank side that is subject to the higher load (must always have the fixed default value of 100%)
f_{low}	Load on the flank side that is subject to the lower load
M	Dimensionless number depending on the type of treatment and load type (see Table B.1 in ISO 6336:2006-3, Appendix B)
R	Stress ratio
Y_M	Alternating bending factor

Treatment	Endurance strength	Coefficient for static proof
Steels		

case-hardened	0.8 ÷ 0.15 Y_S	0.7
case-hardened and shot peened	0.4	0.6
nitrided	0.3	0.3
heat-/induction-hardened	0.4	0.6
not surface-hardened steel	0.3	0.5
Cast steel	0.4	0.6

Table 13.11: Mean stress ratio M as specified in Table B.1 - Mean Stress Ratio - in ISO 6336:2006-3

According to Linke [58] the alternating bending factor (described there as Y_A) is determined as shown in Figure 13.10. For plastics, Niemann recommends 0.8 for laminated fabric and 0.667 for PA (polyamide) and POM (polyoxymethylene).

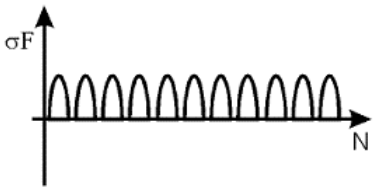
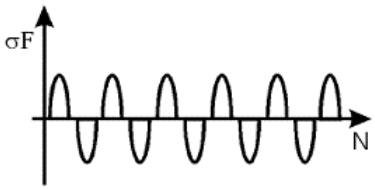
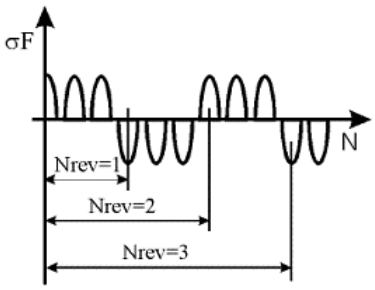
Operating Mode	Operating Factor YA	Load Direction
Pulsating	1	
Alternating	0.7	
Oscillating	$0.85 - 0.15 \cdot \frac{\lg N_{rev}}{6}$ $(1 \le N_{rev} \le 10^6)$ 0.7 $(N_{rev} > 10^6)$	

Figure 13.10: Alternating bending factor in accordance with Linke [58]

13.3.4.1 Load spectrum with changing torque

You can also enter load spectrum elements with negative torque, but then the prefix operator is NOT taken into account.

The problem:

Until now, no calculation guidelines have been drawn up to describe how to calculate gears with changing load spectrums.

The only unambiguous data is that, during every cycle (and in each element of the collective) a change in torque takes place. At this point, the load change corresponds exactly to a double-load with +torque and then with –torque. This instance can be calculated correctly by entering the load spectrum of the +moments and the alternating bending factor YM for the tooth root. The flank is also calculated correctly, because the +moments always apply to the same flank.

If, in contrast, the drive runs forwards for a specific period of time and then runs backwards, the experts agree that the tooth root is not subjected purely to an alternating load (and possibly this is the only point at which an alternating load change takes place). However, discussions are still raging as to how this case can be evaluated mathematically. It is even more difficult to define how mixed load spectra with unequal + moments and –moments for the tooth root are to be handled. For this type of case, only the +moments are observed for the flank (with the prerequisite that the +moments are equal to or larger than the –moments).

Note on the handling of load spectra with reversing torque:

A load progression as represented in Figure 13.10 below, where the tooth is subjected to a load a few times on the left flank, and then a few times on the right flank, can be converted into a load spectrum as shown below. This is represented in an example here.

Load progression (example):

13 loads with 100% of the nominal load (100 Nm) on the left flank, then

9 loads with 80% of the nominal load (80 Nm) on the right flank, etc.

This results in the following process:

11 load cycles with 100% load, positive torque, pulsating; then

1 load cycle with 100% load on the left and 80% load on the right; then

7 load cycles with 80% load, negative torque, pulsating; then

1 load cycle with 80% load on the right and 100% load on the left;

then repeated again from the start.

This can be represented as a load spectrum as follows:

Frequency	Torque	Load left flank	Load right flank
11/20 = 0.55	100 Nm	100%	0%
7/20 = 0.35	80 Nm	0%	100%
2/20 = 0.10	100 Nm	100%	80%

13.3.5 Face load factor

The face load factors $K_{H\beta}$,$K_{F\beta}$,$K_{B\beta}$ take into consideration the influence of an uneven load distribution upon the facewidth on the flank surface pressure, the scoring, and the tooth root stresses. You can specify that the face load factor is either to be set as a constant value or calculated from other values. If you already know the face load factor $K_{H\beta}$, select the `Own input` method and input this value. During a calculation according to DIN/ISO, click the button to open the `Define face load factor` window, in which you can use a number of parameters to calculate the value you require.

The usual setting here is "Calculation according to calculation method". The face load factor is then calculated according to the formulae used in the strength calculation standard (ISO, AGMA or DIN). You will need to input some values for this. These values are displayed on the right of the window (tooth trace modification, etc.) and are described in the sections that follow. You can input other values by clicking the button in the "Define face load factor" window.

The formulae proposed in the standards for defining face load factor KHb enable you to determine KHb very quickly (but only empirically, and so not very accurately). The KHb coefficient calculated using these formulae is usually higher than it actually is, so the calculated value is therefore on the conservative side. If you consider the coefficient to be too high (> 1.5), it is a good idea to perform a more accurate calculation. You can use the "Calculation according to ISO 6336 Annex E" method to do this.

Although the "Calculation according to ISO 6336 Annex E" method is very accurate, it requires quite a lot of time and effort. As described in [44], it calculates any gaping in the meshing, and therefore defines the load distribution over the entire facewidth. To perform this calculation, you will need to know the exact dimensions of the shafts and support. Click the button to input the shaft values stored in the shaft calculation program for the relevant shafts.

Figure 13.6: Inputting shaft data for calculating KHb accurately as defined in ISO 6336 Annex E.

There is no point in inputting shaft data for planetary stages. In their case, the (linear) axial offset in the axial plane and in the normal plane to the axial plane is predefined. This enables you to take into account of any planet carrier deformations that might occur (if known, for example, if you know of this from an FE (Finite Element) calculation). You can also use this variant to calculate a gear pair, if you know the axial offset.

The "Calculation with manufacturing allowance according to ISO 6336 Annex E" method is the most accurate. However, if you use this method, you must click the button to input the toothing tolerance fHb (tooth trace deviation over the carrying facewidth) and the axis alignment tolerance fma (angular deviation of the axis alignment in the plane of action). In this case, the load distribution over the facewidth is calculated 5 times (according to [44]): Firstly without variation, then sequentially using (+fHb,+ fma), (+fHb,- fma), (-fHb,+ fma) and (-fHb,- fma). The largest face load factor KHb determined here is then taken as the end result.

Note: See `Module specific settings -> Face load factor` for settings involved in the calculation according to ISO 6336 Annex E.

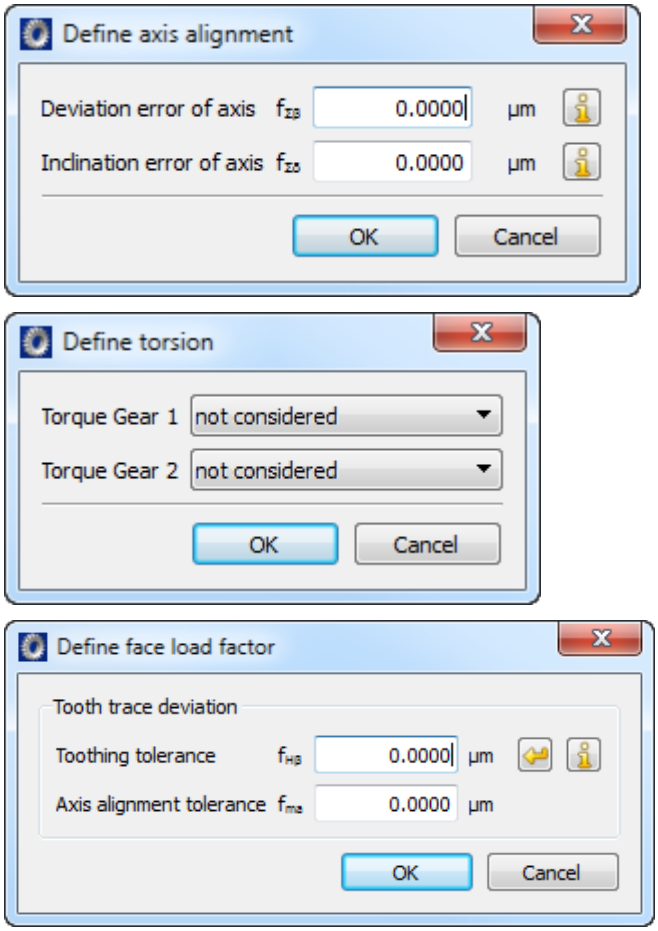


Figure 13.7: Inputting the axis alignment and torsion application for accurately calculating KHb according to ISO 6336 Annex E.

NOTE

Section <Kap19.3>.3 (see page II-634) gives an overview of the characters used in the formulae in this section.

If you want to calculate the face load factor by applying load spectra for each element, select either the `Own input, per load stage`, `Calculation according calculation method` or `Calculation with/without manufacturing allowance according to ISO 6336-1 Annex E, per load stage` method.

For cylindrical gear pairs (Z012), shaft calculation files (W010) can be used to calculate the relative displacement between the gear flanks more accurately, based on the corresponding shaft bending lines (see page II-380). The torque, power, and force, for all the load elements involved in the shaft calculation are then modified according to the partial load coefficient w_t. In planetary gear units you can define the deviation error of axis and the inclination error of axis between the sun/internal

gear and planets/internal gear. Meshing is then calculated simultaneously for both gear sets (sun-planets and planets-internal gear).

This means you can include any torsion that occurs in the gear. Here the calculation assumes a solid cylinder or hollow cylinder (external diameter = root circle + 0.4*normal module or operating pitch circle, depending on what has been predefined under "Settings", bore = inside diameter). In other words, the inside diameter is taken into account and the torque on one side is zero. The torque is distributed in a linear fashion along the facewidth (parabolic course of the deformation by torsion). You can select which side is to be subjected to torsion moment. In this case, I and II refer to the same side (as is the case when you enter the toothing corrections). The increase in torque for a sun in planetary stages is taken into account by using multiple contacts (several planets). Multiple contact is not taken into consideration in any other configuration (e.g. for pairs of gears). In such situations, the correct torque curve can be used if the deformation is taken from shaft calculation.

To help you calculate the face load factor as defined in ISO 6336, Appendix E, the facewidth is divided into slices:

You can set the accuracy of the face load factor calculation according to Appendix E in the "Define number of slices" dialog. Click the plus button next to where you select the calculation method to open this dialog.

13.3.5.1 Tooth trace modification

You can achieve balanced contact characteristics if you perform targeted tooth trace modifications. Figure 1.5 shows the two most frequently used modifications.

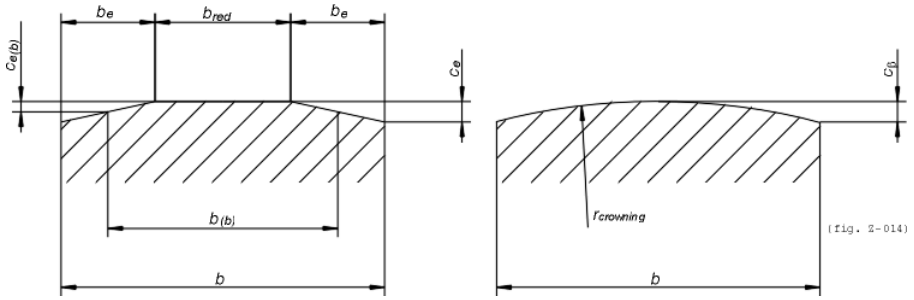


Figure 13.5: End relief and crowning

13.3.5.2 Cylindrical gear pairs

The calculation, as specified in ISO 6336, is based on an approximate estimate of the pinion deformation. In many cases, this is extremely inaccurate and usually results in face load factors that are much too high.

The face load factor is the ratio between the maximum and average line load. The basic equation used for the face load factor corresponds to equation (41) in the standard[4] :

$$K_{H\beta} = \frac{(F_m / b)_{\max}}{F_m / b} = 1 + \frac{F_{\beta y} \cdot c_{\gamma\beta}}{2 \cdot (F_m / b)} \tag{13.4}$$

The effective tooth trace deviation $F_{\beta y}$, see equation (52) in the standard, is defined with the inclusion of a linearized, specific deformation component f_{sh}. The multiplier 1.33 in the equation stands for the conversion of the linearized specific deformation progression into the real parabolic progression - see equation. (13.5).

$$F_{\beta y} = F_{\beta x} \cdot \kappa_\beta = (1.33 \cdot f_{sh} + f_{ma}) \cdot \kappa_\beta \tag{13.5}$$

The manufacturer component of the tooth trace deviation f_{ma} is derived from tolerances specified by the manufacturer. If a usual procedure for checking the accuracy grade is used, you can apply this formula (equation (64) in the standard):

$$f_{ma} = \sqrt{f_{H\beta 1}^2 + f_{H\beta 2}^2} \tag{13.6}$$

If you have used KISSsoft's shaft calculation software to calculate the exact tooth trace deviation due to deformation (torsion and bending) in the plane of action, you can correct the approximate value f_{sh} extrapolated from the standard and therefore calculate the width factors much more precisely! The formula as specified in ISO6336 only applies to solid shafts or hollow shafts that have an inside diameter that is less than half of the external diameter.

[4] *The equation numbers used in this section refer to ISO 6336:2006*

In Method C2, the face load factor is calculated using these equations:

Size	Drop-down list	Selection	Equation	No.
$K_{H\beta}$				(7.04)/ (7.06)
F_β				(7.08)
F_β	position of the contact pattern	not verified or inappropriate		(7.26)
		favorable		(7.27)
		optimal		(7.28)
f_{sh}				(7.39)
f_{sh0}	Tooth trace modification	None	$0.023 \cdot \gamma$	(7.31)
		Crowning	$0.012 \cdot \gamma$	(7.34)
		End relief	$0.016 \cdot \gamma$	(7.35)
		Solid	$0 \cdot \gamma$	a)
		Slight crowning	$0.023 \cdot \gamma$	b)
		Helix angle modification	$0.0023 \cdot \gamma$	b)
		Crowning + helix angle correction	$0.0023 \cdot \gamma$	b)
γ	Toothing	straight/angled		(7.32)
		helical		(7.33)
f_{ma}	Tooth trace modification	None	$1.0 \cdot f_H\beta$	(7.51)
		Crowning	$0.5 \cdot f_H\beta$	(7.53)
		End relief	$0.7 \cdot f_H\beta$	(7.52)
		Total tooth trace modification	$0.5 \cdot f_H\beta$	a)
		Slight crowning	$0.5 \cdot f_H\beta$	b)
		Helix angle modification	$1.0 \cdot f_H\beta$	b)
		Crowning + helix angle correction	$0.5 \cdot f_H\beta$	b)

Table 13.6: Overview of equations used in accordance with DIN 3990:1987

a) same as DIN 3990, Equation (6.20)
b) same as ISO 9085, Table 4

Size	Drop-down list	Selection	Value	No.
$K_{H\beta}$				(39)/ (41)
F_β				(43)
F_β	position of the contact pattern	not verified or inappropriate favorable optimal		(52) (53) (56)
f_{sh}				(57)/ (58)
f_{ma}				(64)
	Tooth trace modification	None	1 / 1	Table 8
		Crowning	0.5 / 0.5	
		End relief	0.7 / 0.7	
B_1/B_2		Full	0 / 0.5	(56)
		Slight crowning	1 / 0.5	Table 8
		Helix angle modification	0.1 / 1.0	
		Crowning + helix angle correction	0.1 / 0.5	

Table 13.7: Overview of equations used in accordance with ISO 6336:2006

Type of pinion shaft

Load as defined in ISO 6336:2006, Figure 13 (DIN 3990/1, Figure 6.8) or the bearing positioning is shown in Figure 13.6.

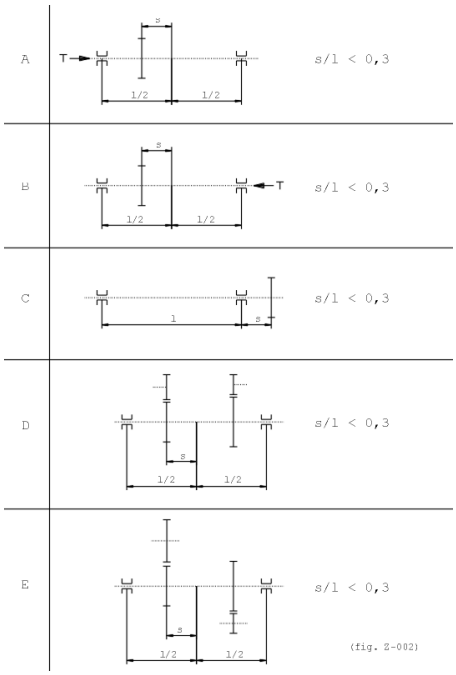


Figure 13.6: Load as defined in ISO 6336:2006, Figure 13.

Load in accordance with AGMA 2001

Definition of s and s_1 in accordance with AGMA 2001, Figure 13-3. Figure 13.7 shows the bearing positioning as described in AGMA 2001.

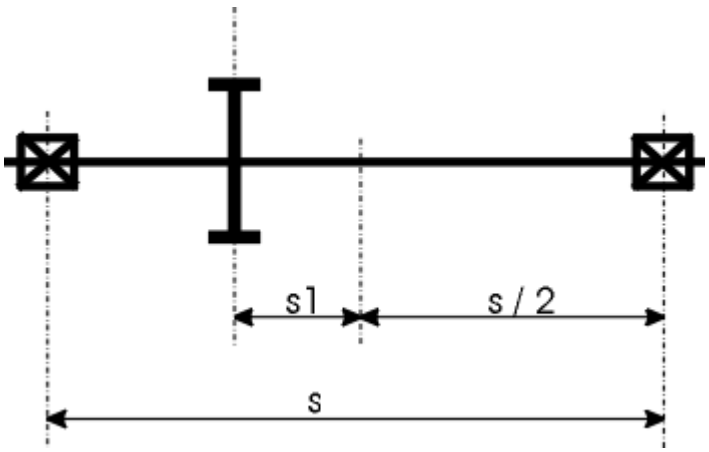


Figure 13.7: Load as defined in AGMA 2001, Figure 13-3

13.3.5.3 Planetary stages

The face load factor for planetary stages is calculated in a different way than for cylindrical gears. The deformation component f_{sh} is derived from the deformation of the mating gears on the shaft due to torsion and bending. In order to simplify the situation for a pinion-gear pair, only the pinion deformation (which is much greater) is taken into account.

Planetary stages are subject to the following sizable deformations: Since the sun has several tooth meshings, all radial forces are canceled out No bending takes place because deformation is caused solely by torsion. However, the multiple meshing which corresponds to the number of planets means this is greater than for normal pinion shafts. - A planet gear has two meshings with opposed torques, which prevents deformation due to torsion. Bending may be calculated in the same way as for pinion shafts; however, the circumferential force must be doubled because of the sun/planet and planet/internal gear. - In most cases, rim deformation can be ignored. As a result, the torsion at the pinion and the bending at the planet bolt must be taken into consideration for sun/planet meshing whereas, for planet/internal gear, only the bending at the planet bolt is important. For most support arrangements for planets, bending can be determined analytically using a procedure similar to that specified in ISO 6336. Figure <12>.8 shows the four most common cases.

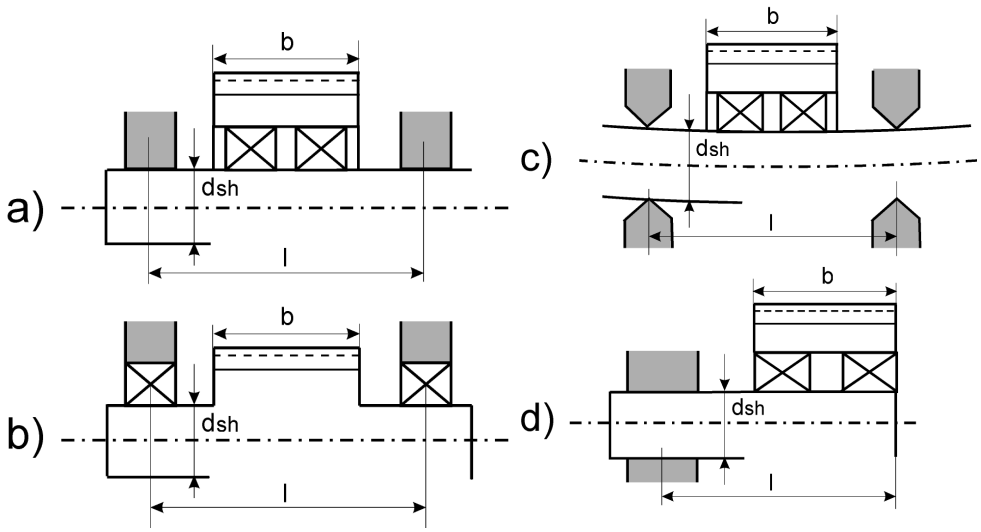


Figure 13.8: Support arrangement for planets

a) Planets mounted with fixed clamped bolts on both sides

b) Planets are on bolts, which have flexible bearings on planet carrier

c) Planets mounted with gently tightened bolts (flexible bearings) on both sides

d) Planets mounted with fixed clamped bolts on one side

Configuration	ISO 6336	DIN 3990	AGMA 2001
a	Part 1,	Formulae	Chapter 15, (37)
	Appendix D	6.20/6.21/6.24/6.25/	
b	Part 1,	Formulae	Chapter 15, (37)
	Appendix D	6.24A/6.24B/6.25A/6.25B	
c and d	Part 1,	Formulae as defined in Part 1,	Chapter 15, (37)
	Appendix D	Appendix C, see [49]	

Table 13.8: Configuration of planetary stages as defined in ISO, DIN and AGMA

For ISO 6336 see also the explanation in [49].

Equations (13.7a - 13.7d) show the bending components in relationship to the distance x from the planet's face width. As we are only interested in bending variation across the facewidth, the constant term was left out of the equations so that $f_b(x = 0)$ is zero. Similar formulae can be found in other technical documentation [38]. For cases a to d as illustrated in Figure 1.8 the following equations apply.

$$f_{bpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m / b}{d_{sh}^4 E_p} \cdot \left[\frac{x^4}{24} - \frac{bx^3}{12} - \frac{bx^2\left(3l - 6b + b^2\right)}{48} + \frac{b^2 x\left(3l - 4b + b^2 / l\right)}{48} \right] \quad (13.7a)$$

$$f_{bpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m / b}{d_{sh}^4 E_p} \cdot \left[\frac{x^4}{24} - \frac{bx^3}{12} - \frac{bx^2\left(l - b\right)}{8} + b^2 x\left(\frac{l}{8} - \frac{b}{12}\right) \right] \quad (13.7b)$$

$$f_{bpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m / b}{d_{sh}^4 E_p} \cdot \left[\frac{x^4}{24} - \frac{bx^3}{12} - \frac{bx^2\left(l - b\right)}{8} + b^2 x\left(\frac{l}{8} - \frac{b}{12}\right) \right] \quad (13.7c)$$

$$f_{bpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m / b}{d_{sh}^4 E_p} \cdot \left[\frac{x^4}{24} - \frac{bx^3}{6} - \frac{b^2 x^2}{4} + \frac{blx\left(l - b\right)}{2} \right] \quad (13.7d)$$

The sun's deformation due to torsion, as described in equation (13.8), can be calculated from Appendix D (f_T in accordance with formula D.1).

$$f_{tso} = p \cdot \frac{8}{\pi} \cdot \frac{F_m / b}{0.39 \cdot E_{so}} \left(\frac{b}{d_{so}} \right)^2 \cdot \frac{x}{b} \left(1 - \frac{x}{2b} \right) \tag{13.8}$$

In order to stay as close as possible to the methods used in ISO 6336 (and be able to apply formula 2), the average deformation components f_{bmpla} (bending at the planet) and f_{tmso} (torsion at the sun) will be determined.

$$f_{tso} = \frac{1}{b} \int_0^b f_{tso}(x) dx \qquad f_{bmpla} = \frac{1}{b} \int_0^b f_{bmpla}(x) dx \tag{13.9}$$

$$f_{bmpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m / b}{d_{sh}^4 E_p} \cdot \frac{b^3}{16} \left(-\frac{b}{5} + \frac{l}{6} + \frac{b^2}{18l} \right) \tag{13.10a}$$

$$f_{bmpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m / b}{d_{sh}^4 E_p} \cdot \frac{b^3}{16} \left(\frac{l}{3} - \frac{b}{5} \right) \tag{13.10b}$$

$$f_{bmpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m / b}{d_{sh}^4 E_p} \cdot \frac{b^3}{16} \left(\frac{l}{3} - \frac{b}{5} \right) \tag{13.10c}$$

$$f_{bmpla} = 2 \cdot \frac{64}{\pi} \cdot \frac{F_m / b}{d_{sh}^4 E_p} \cdot \frac{b^2}{4} \left(\frac{b^2}{5} - bl + l^2 \right) \tag{13.10d}$$

$$f_{tmso} = p \cdot \frac{8}{3\pi} \cdot \frac{F_m / b}{0.39 E_{so}} \cdot \left(\frac{b}{d_{so}} \right)^2 \tag{13.11}$$

According to ISO 6336:2006, equation D.8, the linearized deformation components of the tooth trace deviation f_{sh} (in mm) will be:

$$f_{sh}(Paarung\ Sonne - Planet) = 2000 \cdot \left(f_{tmso} + f_{bmpla} \right) \tag{13.12}$$

$$f_{sh}(Paarung\ Planet - Ring) = 2000 \cdot f_{bmpla} \tag{13.13}$$

This can then be used with equations (13.4) and (13.5) to calculate face load factors for the sun/planet and planet/internal gear.

Formula symbol	**Unit**	**Meaning**
b	mm	Meshing face width
$c\gamma\beta$	N/(mm µm)	Meshing stiffness
d_{pla}	mm	Planet reference circle
d_{sh}	mm	Planet shaft diameter
d_{so}	mm	Sun reference circle
E_p	N/mm^2	Young's modulus planet bolt/shaft
E_{so}	N/mm^2	Young's modulus sun
f_{bpla}	mm	Planet shaft deflection
$f_H\beta$	μm	Helix slope deviation in accordance with ISO 1328
$f_m\alpha$	μm	Tooth trace deviation
		manufacture error
f_{sh}	μm	(Linearized) deformation components of the
		tooth trace deviation
f_{tso}	mm	Sun torsion deviation
F_m/b	N/mm	Average line load
$(F_m/b)_{max}$	N/mm	Maximum local line load
$F\beta_y$	μm	Actual tooth trace deviation
$K_H\beta$	[-]	Face load factor
l	mm	Planet bolt/shaft length
p	mm	Number of planets
x	mm	Distance to the left side of the facewidth
$\kappa\beta$	[-]	Run-in factor

13.3.5.4 Calculation of KHβ with manufacturing errors

According to ISO 6336-1(E), the lead variation (fHb) and shaft misalignment (fma) errors are applied in the plane of action in such a way that they increase the meshing gap. In such a case, their combined effect is taken into account for the flank gap in three cases:

- Case 1: fma = fHb = 0, i.e. no error
- Case 2: fma = |fma|, fHb = |fHb|, so positive values for both errors
- Case 3: fma = -|fma|, fHb = -|fHb|, so negative values for both errors

The face load factor $K_{H\beta}$ is calculated for all three cases, and the maximum value is selected as the face load factor of the gear pair.

The positive direction always lies in the direction of the pinion's material, seen from a common point of contact.

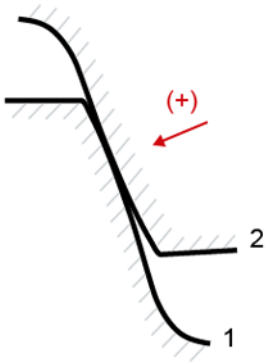
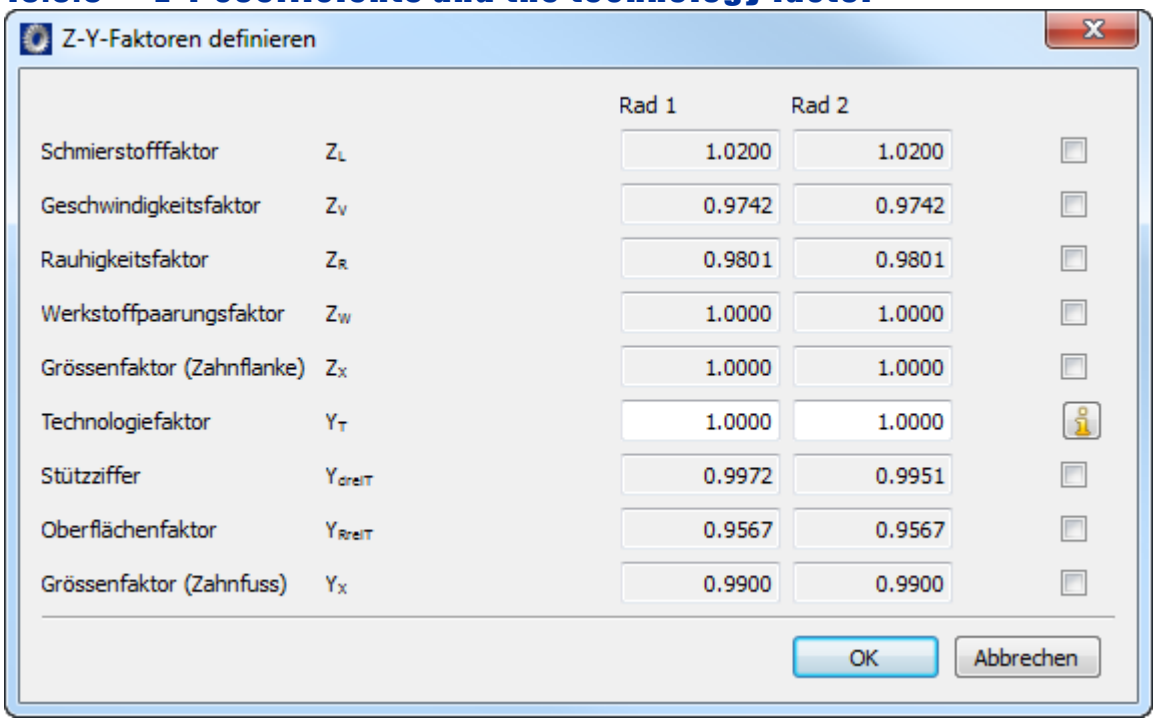


Figure 13.9: Definition of the positive direction

In all three cases, the manufacturing error is documented in the report and in the gaping and load distribution graphics.

13.3.6 Z-Y coefficients and the technology factor



If necessary, you can modify any of the coefficients that affect the permitted material values (root and flank) as specified in ISO or DIN in the "Z-Y coefficients" window.

Coefficients ZL, ZV, ZR, ZW and ZX affect the safety against pitting limit sigHG. Coefficients YT, YdrelT, YRrelT and YX influence the root safety limit sigFG.

You can predefine any of these coefficients in the range 0.5 to 2.0. However, if you input a value that lies outside this range, it will be set automatically to 1.0.

The technology factor takes into account the change in tooth root strength caused by manufacturing. In this situation the material's permissible stress is multiplied by $Y_T \geq 1.0$. This coefficient is not specified in the DIN or AGMA standards and is therefore set to 1.0.

Treatment of tooth root area	**Technology factor Y_T**
Shot peening Case-hardened/carbonitrided toothing Not ground in the reinforced areas	1.2
Rollers Flame- and induction-hardened toothing Not ground in the reinforced areas	1.3

Grinding For case-hardened or carbonitrided toothing	 0.7 (general) 1.0 (CBN grinding discs)
Cutting machining Not for profile ground teeth!	 1.0

Table 13.12: Technology factor according to Linke

According to Bureau Veritas/RINA [70], the technology factors in Table 13.13 shall be applied.

Treatment of tooth root area		**Technology factor Y_T**
Shot peening,	Case hardening steel	1.2
Shot peening,	Heat treatable steel	1.1
Shot peening,	Nitriding steel	1.0

Table 13.13: Technology factors as defined by Bureau Veritas/RINA Directives

Table 13.14 shows the technology factors as defined in ISO 6336-5:2003, Section 6.7. These only apply to tooth root bending stresses and shot peened case hardening steel.

Material class	**Technology factor Y_T**
ML	1.0
MQ	1.1
ME	1.05

Table 13.14: Technology factor according to ISO 6336-5:2003, Section 6.7

13.3.7 General calculation procedure for KHbeta as specified in ISO 6336-1, Appendix E.

1. Import the shaft files and select the correct gears, perform the initialization
2. Calculate the shafts and determine the diagram of bending and torsion in the point of contact (if uniform load distribution is present, determine these values along the facewidth of the gear)
3. Take into account flank modifications from Z012 (not W010)
4. Calculate the gaps in the tooth contact, then the load distribution with tooth contact stiffness and finally calculate $K_{H\beta}$
5. Use the calculated load distribution to correct the load distribution on the original gears
6. Divide the gears into "sections" whose load values are defined in the previous step
7. Use the flank contact ratio (as a vector) from the previous iteration g_{k-1} and the current flank contact ratio g_k to calculate the root of the sum of the square error

$$\lambda = \sqrt{\sum \left(100 \cdot \frac{g_k^i - g_{k-1}^i}{g_{k-1}^i} \right)^2}$$

If λ>0.1%, go back to step 2 and perform further iterations. Otherwise finish.

This procedure exactly follows the method described in ISO 6336-1, Appendix E, but uses a stricter iteration criterion.

13.4 Reference profile

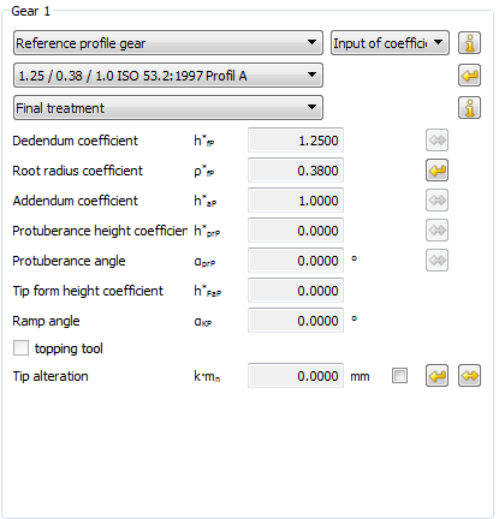


Figure 13.16: `Reference profile` input window

In contrast to traditional mechanical engineering, where a predefined standard reference profile is most commonly used, in precision mechanics the reference profile is often modified. Input the toothing reference profile or the appropriate tool in the `Reference profile` input window. You can input this data either as coefficients, as lengths or as the diameter.

13.4.1 Configuration

The reference profile of the gear toothing is usually predefined. However, you can also define your own hobbing cutter or pinion type cutter. The pinion type cutter parameters are also used in the strength calculation to calculate the tooth form factor. You can also select the `Constructed involute` for precision engineering. In this case, the involute is defined directly together with a root radius.

13.4.1.1 Cutter/Tool: Hobbing cutter

Select the hobbing cutter you require from the selection list and then click the button, (see Figure 13.17.)

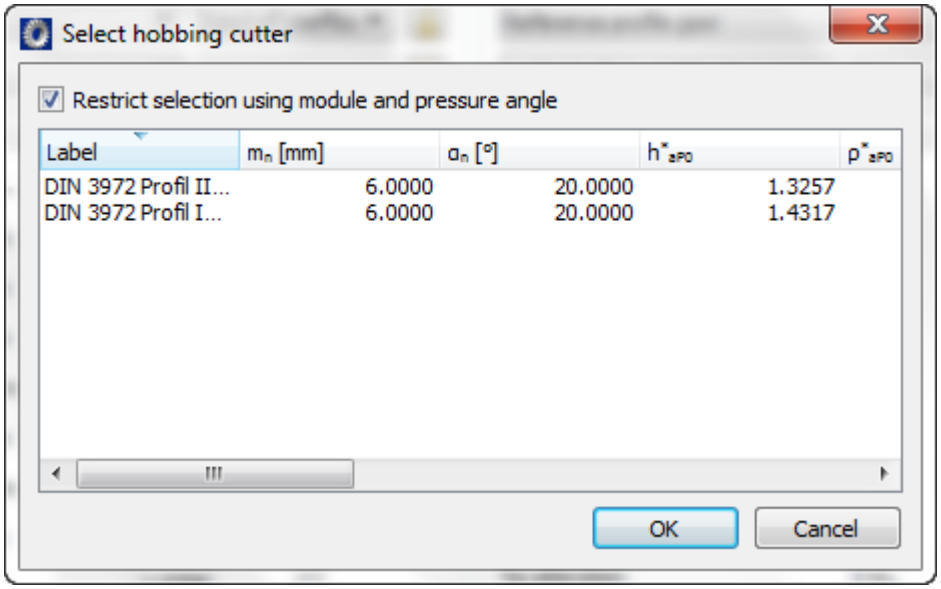


Figure 13.17: `Select hobbing cutter` window

If you select a standardized profile (e.g. DIN 3972III), the list displays the tools that are present in the database. The name of the cutter file list is entered in the database. Click on the `Restrict selection using module and Pressure angle` checkbox to limit the display to tools whose modules and pressure angles match those defined in the gear geometry. Therefore, only tools that match the selected module and pressure angle are displayed.

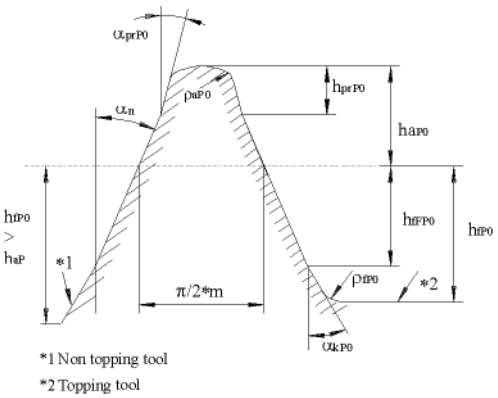


Figure 13.18: Reference profile for the `Tool:` configuration `Hobbing cutter`

Select `Own Input` to directly define your own cutter:

- The cutter addendum coefficient h^*_{aP0} defines the cutter addendum which defines the gear root circle. A usual value is 1.25.
- The cutter tip radius factor ρ^*_{aP0} defines the cutter tip radius which then defines the gear root radius. The tip fillet radius is limited by the maximum, geometrically possible radius, depending upon the profile addendum and the pressure angle. This value usually lies in the range 0.2 to 0.38.
- The dedendum coefficient h^*_{fP0} defines the dedendum that, with a topping tool, determines the tip circle. A usual value for this is 1. In a non topping tool, there has to be a certain amount of clearance between the tool and the gear tip circle, which the software checks. 1.2 is a usual value for an addendum of the reference profile of 1.
- The root radius coefficient ρ^*_{fP0} defines the cutter root fillet radius. In a topping tool, the root radius cuts a tip rounding on the gear in most cases. Depending on the geometric conditions, a chamfer or corner may occur on the tip.
- The protuberance height coefficient h^*_{prP0} defines the protuberance length measured from the addendum. The protuberance is used as an artificial undercut to avoid the creation of a grinding notch. The protuberance height can be calculated from the protuberance size and angle.
- The protuberance angle α^*_{*prP0} is usually smaller than the pressure angle, however, in some special cutters it may also be larger. In this case no undercut is present, but the tooth thickness at the root of the gear is larger. The protuberance angle can be calculated from the protuberance height and size. If you enter the value "0", no protuberance will be present.
- When calculating the contact ratio, protuberance is not taken into account until it reaches a certain value because a contact under load is assumed in the profile modification. You can specify the threshold used to take into account the protuberance and buckling root flank for diameters in the `Calculation -> Settings (see page II-435)` menu item.
- The root form height coefficient h_{FfP0}^* defines the end of the straight flank part of the tool with pressure angle α_n. The height is measured from the tool reference line.
- The ramp angle a_{KP0}^* defines a ramp or a profile modification that is present in the cutter. The length is determined by the root form height coefficient. The angle must be greater than the pressure angle α_n. If you enter the value "0", this part will be ignored.

- The threshold value used for protuberance is also taken into consideration here when calculating the diameter and the contact ratio (→ more information (see page II-435)).
- The tooth thickness factor of the reference line s^*_{P0} for the usual tools $s^*_{P0} = \pi/2$. The value can be overwritten for special tools.
- The addendum coefficient of the gear reference profile h^*_{aP} for a non topping cutter/tool is defined with the usual value of $h^*_{aP} = 1$ of the gear reference profile or by the gear's tip circle. The value can be converted from the tip circle.

13.4.1.2 Cutter/Tool: Pinion type cutter

Click the button next to the pinion type cutter designation to select a pinion type cutter for inside and outside gears from a list. Pinion type cutters as specified in DIN 1825, 1826 and 1827 are listed here. You use this window in the same way as the `Define milling cutter` window in Figure 13.19. The default setting is for the list to display only those tools that match the selected module, meshing and helix angle.

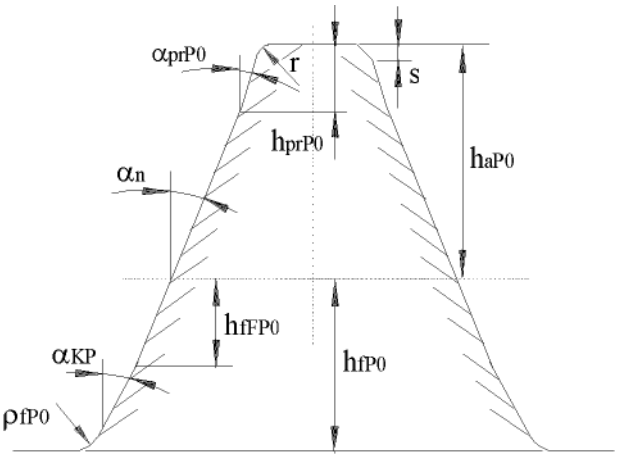


Figure 13.19: Reference profile for the `Tool:` configuration `Pinion type cutter`

Select `Own Input` to directly define your own pinion type cutter:

- KISSsoft can prompt the number of teeth z_0 for the cutter. If the number of teeth is too small, it may not be possible to manufacture the tip form circle and/or the root form diameter of the cylindrical gear. If the number of teeth is too great, it may cause collisions during manufacture.

- The pinion type cutter profile shift coefficient x_0 is often unknown. However, it does influence the root circle of the resulting gear. This value is set automatically, together with the number of teeth.
- A pinion type cutter tip often takes the form of a radius or a chamfer. Click the button to define the corresponding numerical value.
- The pinion type cutter addendum coefficient h^*_{aP0} defines the pinion type cutter addendum that determines the pinion type cutter tip circle and the gear root circle. A usual value is 1.25.
- The pinion type cutter dedendum coefficient h^*_{fP0} defines the pinion type cutter dedendum height that determines the tip circle for a topping tool. A usual value for this is 1. In a non topping tool, there has to be a certain amount of clearance between the tool and the gear tip circle, which the software checks. 1.2 is a usual value for an addendum of the reference profile of 1.
- The root radius coefficient of the pinion type cutter ρ^*_{fP0} defines the radius at the cutter root. In a topping tool, the root radius cuts a tip rounding on the gear in most cases. The input value is only displayed for a topping tool.
- The protuberance height coefficient h^*_{prP0} defines the protuberance length measured from the addendum. The protuberance is used as an artificial undercut to avoid the creation of a grinding notch.
- The protuberance angle α^*_{prP0} is usually smaller than the pressure angle. If 0 is input, no protuberance is present.
- When calculating the contact ratio, protuberance is not taken into account until it reaches a certain value because a contact under load is assumed in the profile modification. You can specify the threshold used to take into account the protuberance and buckling root flank for diameters in the `Calculation -> Settings (see page II-435)` menu item.
- The root form height coefficient h_{FfP0}^* defines the end of the tool involute with the pressure angle α_n. The height is measured from the tool reference line.
- The ramp angle α_{KP0}^* defines a ramp flank or a profile modification that is present in the cutter. The length is determined by the root form height coefficient. The angle must be greater than the pressure angle α_n. If you enter the value "0", this part will be ignored.
- The threshold value used for protuberance is also taken into consideration here when calculating the diameter and the contact ratio (→ more information (see page II-435)).
- The addendum coefficient of the gear reference profile h_{aP}^* with the usual value of $h_{aP}^* = 1$ defines the tip circle of the gear for a non topping tool. The value can be calculated from the tip circle.

13.4.1.3 Reference profile

The reference profiles displayed here are taken from the database. If you can't find a suitable reference profile here, you must first enter it in the database (see page I-114) (`Z000.ZPROF`). Alternatively, select `Own Input` from the drop-down list so you can edit all the input fields, and therefore change all the reference profile parameters. The `Label` input field is displayed under the `Reference profile` drop-down list. There you can enter the name of your own profile, which will then appear in the calculation report.

NOTE

You do not create a new entry in the database when you define your own profile in the `Own Input` field.

The reference profile details are according to ISO 53, DIN 867 or DIN 58400. This is the reference profile data for the gear. You can calculate the corresponding values in [mm] by multiplying it with the normal module. Please note the following points:

- If the reference profile is set to `Own Input`, the tip alteration (see section "Modifications" on page II-326) is set to zero. For this reason the addendum may change when you toggle from one window to another.
- If you are using reference profile `BS4582-1:1970 Rack 2` to determine the correct tip and root diameters, you must input an appropriate tooth thickness tolerance of

  $$A_s = -0.1572 \cdot m_n$$

  . The tip and root diameter will then match the values defined in BS4582-1(8)).
- The ramp flank is usually used to generate a tip chamfer[5]. Alternatively, you can also use a small buckling root flank to generate a profile modification. However, profile corrections are usually defined in the `Modifications` (on page II-326) window.
- If the angle of the ramp flanks is only slightly different to the pressure angle, it is not taken into account in the contact ratio because the assumption for profile modifications is that the contact ratio will not decrease under load. In contrast, the contact ratio should be reduced accordingly for a chamfer. In `Settings` (see page II-435), you can specify the difference in angle that is to be used as the threshold in profile modifications and chamfers.

[5] *also called semi-topping.*

- If a premachining tool is used, the additional measure for the preliminary treatment `must` be entered separately. You must input the gear's reference profile for the preliminary treatment. This data, and the grinding wheel data, is used to calculate the reference profile used during the final treatment and

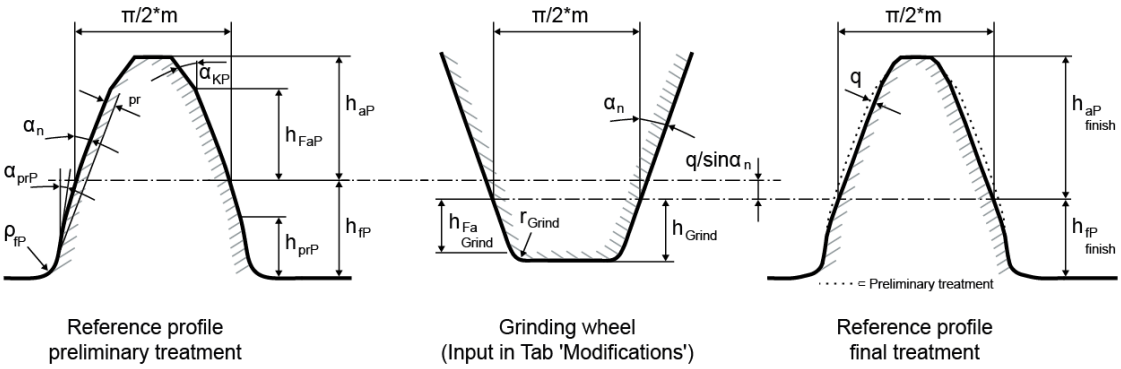


Figure 13.20: Reference profile for configuration: Reference profile gear

- Click the button next to the reference profile drop-down list to display a reference profile for a deep tooth form with the predefined required transverse contact ratio. You can then transfer a value for the required transverse contact ratio in `Calculation > Settings` in the `Sizings` (see page II-434) tab.
- h_{aP}^* always applies as the normal gear reference profile. The tooth thickness on the reference line is

$$s_P = \frac{\pi}{2} \cdot m_n \tag{12.19}$$

13.4.1.4 Constructed involute

When you select `Constructed involute`, you do not need to enter as many parameters as you do when you select `Reference profile`. The essential difference is that no simulation of manufacture is performed, and the involute is generated directly.

In the gear root, the involute is closed by a radius that is defined by the root radius coefficient ρ_{fP}. In theoretical involutes, the root radius coefficient is usually greater than the coefficient for a reference profile, because the manufacturing process does not involve a meshing movement.

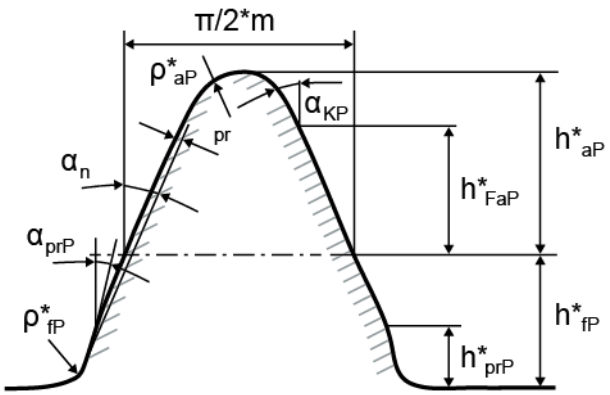


Figure 13.13: Reference profile for configuration: Constructed involute

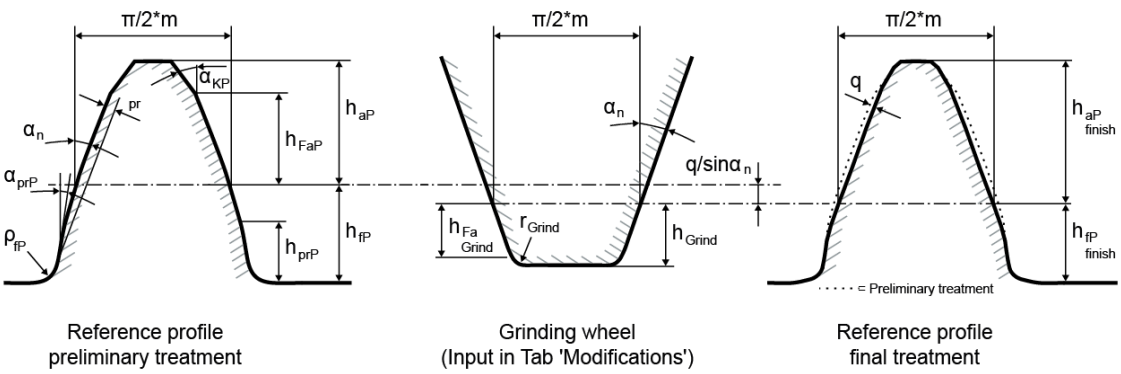


Figure 13.21: Gear reference profiles during preliminary treatment; grinding wheel; reference profile during final treatment

Note: If a cutter, pinion type cutter or constructed involute is selected as the preliminary treatment tool, the gear reference profile for preliminary treatment is calculated internally from the tool data.

In this case, the root circle is created by the premachining cutter and the flank by the grinding process. To complete this process correctly, select either `Preliminary treatment` (with own input, or with reference profile for grinding allowance III or IV as specified in DIN 3972) and then select `final treatment`. If you decide to use preliminary treatment, the `Grinding`

`allowance` field appears. You can also add your own tolerances to the database. Enter the profile of the premachining tool (except: h_{aP}^*) as the reference profile. As the tooth thickness deviations (tolerances) you have to enter the tooth thickness allowance of the finished gear teeth (A_s). In KISSsoft the grinding allowance is then calculated for the finished gear teeth. The premachining is then performed using the total deviation of tooth thickness:

$$A_s' = A_s + \frac{2 \cdot q}{\cos \alpha_n} \tag{12.20}$$

For special requirements, click the button in the "Define grinding allowance tolerance" window to increase the tolerance. If you input data for qmax-qmin, the program applies qmax = q+(qmax-qmin)/2 and qmin = q-(qmax-qmin)/2 to calculate the allowances during preliminary treatment.

In the `Modifications` (on page II-326) input window you can enter the infeed and the radius of a grinding wheel.

KISSsoft then determines the reference profile that corresponds to the finished tooth form. It does this by calculating factors Y_F and Y_S for the tooth root strength. The tooth form is then defined automatically by overlaying the premachining contour with the subsequent grinding process. The root diameters are derived from the reference profile for premachining. The control data (e.g. base tangent length) is calculated and printed out for both the premachined and the finished gear teeth.

IMPORTANT EXCEPTION

The addendum coefficient h_{aP}^* is the theoretical addendum coefficient that is used to calculate the theoretical tip diameter coefficient. The appropriate minimum root height of the hobbing cutter h^*_{fP0}, which is required to create the tooth form without topping, is printed in the report. h_{aP}^* always applies as the final treatment reference profile for the gears. The tooth thickness on the reference line is $\pi/2 * m_n$.

13.4.3 Tip alteration

The tip alteration k*mn is usually calculated from the profile shift total to ensure that the tip clearance does not change. However, if the reference profile is set to `Own Input`, the tip alteration will not be calculated. In an external gear pair, a reduction in the tip alteration in a negative value for the tip circle reduction. In contrast, in internal teeth, the result is a positive value for both gears, and therefore

also an increase in the tooth depth. In KISSsoft, the tooth depth of internal teeth is not increased and therefore the tip alteration is limited to 0.

Alternatively, you can specify your own tip alteration, however, this only has an effect on non-topping tools. Otherwise the value is set to 0 when it is calculated. Click a Sizing button to calculate the proposed value for a constant tip clearance.

Click the Recalculate button to input the tip diameter (either d_a, d_{aE} or d_{ai}) to calculate the tip alteration using the current reference profile.

13.5 Tolerances

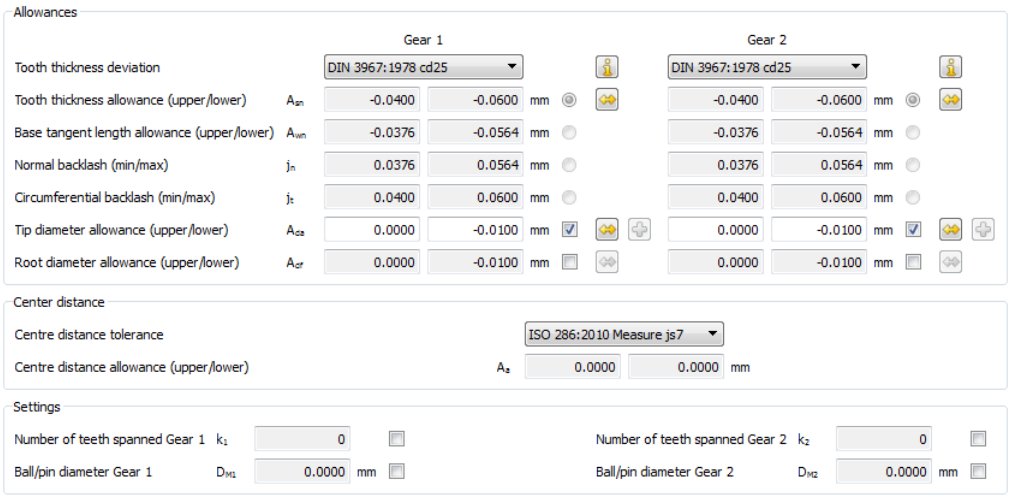


Figure 13.22: `Tolerances` input window

The toothing geometry is calculated for a backlash-free state. A slightly smaller tooth thickness is manufactured, to prevent the gears jamming in practice. This reduction in tooth thickness (in contrast to the backlash-free state) is known as the "tooth thickness allowance". The upper tooth thickness allowance is the upper limit of the tooth thickness. The lower tooth thickness allowance is the lower limit of the tooth thickness.

EXAMPLE

Tooth thickness in a backlash-free state:	4.560 mm
Upper deviation of tooth thickness (top limit):	-0.050 mm
Lower tooth thickness allowance:	-0.060 mm
This results in the actual tooth thickness:	4.500 to 4.510 mm

13.5.1 Tooth thickness tolerance

This drop-down list contains the tolerances listed below. You can also include your own tolerance tables. You will find more detailed information about this in the section about the KISSsoft Database tool (see section "External tables" on page I-122).

13.5.1.1 DIN 3967

Selection a tolerance as specified n DIN 3967 (for gearbox a module greater than 0.5 mm). Suggestions as defined by Niemann [65 (see section "Gear teeth when existing shaft data is present" on page I-210)](page 84):

Cast ring gears	a29, a30
Ring gears (normal clearance)	a28
Ring gears (narrow clearance)	bc26
Turbo gears (high temperatures)	ab25
Plastic machines	c25, cd25
Locomotive gears	cd25
General mechanical engineering,	
Heavy machines, non-reversing	b26
General mechanical engineering,	
Heavy machines, reversing	c25,c24,cd25,cd24,d25,d24,e25,e24
Vehicles	d26
Agricultural vehicles	e27, e28
Machine tools	f24, f25
Printing presses	f24, g24
Measuring gear units	g22

13.5.1.2 ISO 1328

The current edition of ISO 1328 no longer includes tolerance classes for tooth thickness allowances. For this reason, many companies have continued to use the tolerance classes specified in the old 1980 edition.

13.5.1.3 DIN 58405

Proposals as specified in DIN 58405, Part 2: Allowances for precision mechanics; usual gear modifications as defined in DIN 58405 Sheet 2

Material	Processing	Center distance tolerance	Base tangent length

			tolerance
Hardened steel	Ground	5J	5f
Through hardened steel	finest milled	6J	6f
Light metal	finely milled	7J	7f
Light metal	finely milled	8J	8f
Steel/laminate	finely milled	6J	6e
Steel/laminate	finely milled	7J	7d/7c
Light metal	finely milled	8J	8d/8c
Plastic	milled	9J	9e/9d
Plastic	injection molded	10J	10e

13.5.1.4 Own Input

Select this option to input your own data. However, you should note that the values for tooth thickness allowance, the normal or circumferential backlash (per gear) and the base tangent length allowance all depend on each other. The (negative) base tangent length allowance corresponds to the normal backlash.

13.5.2 Tip diameter allowances

You can specify the tip diameter allowances if a non-topping tool was defined. In contrast, the tip diameter allowances for a topping tool are defined from the tooth thickness allowances. These allowances influence the effective contact ratio due to the effective tip circle.

Click the button to specify a tolerance field in accordance with ISO 286. The tolerances prefix is changed in internal teeth because the tip circle is used as a negative value in the calculation.

Click the button to specify the minimum and maximum tip diameter from which the allowances are to be calculated.

13.5.3 Root diameter allowances

Root diameter allowances are usually calculated from the tooth thickness allowances. In the gear cutting process, the backlash is produced by reducing the manufacturing distance of the tool. This is why the root diameter allowances depend on the tooth thickness allowances.

In special cases, a different manufacturing process is used e.g. for sintered gears or extruded plastic gears. The user can then input their own root diameter allowances.

Click the button to specify the minimum and maximum root diameter from which the allowances are to be calculated.

13.5.4 Center distance tolerances

The center distance tolerances are defined either by a standard tolerance taken from the database or the value you enter in the `Own Input` field. They influence the intermeshing allowance and the contact ratio.

13.5.5 Settings

The base tangent length and the mass across balls and rollers for the most suitable number of teeth over which the measurement is to be taken or the roller diameters is specified in the report. If you want to use a different number of teeth spanned or a different diameter of ball/pin in an existing drawing, this is where you can overwrite the values selected by the software.

However, no results are output if you enter values for which a measurement cannot be performed. If the `Do not cancel when geometry errors occur (see page II-429)` option is selected, test masses are also output for cases in which they could not be measured, for example, for points of contact above the tip circle.

NOTE

The proposed ball/pin diameters are taken from the `Z0ROLLEN.dat` file. In the case of splines as defined in ANSI 92.1 these values are taken from the `Z0ROLLENANSI.dat` file. This file corresponds to the recommended diameters specified in DIN 3977. You can then use an Editor to modify the existing ball/pin. You will find more detailed information about how to handle external data records in the External tables (on page I-122) section.

13.6 Modifications

The `Modifications` input window is where you define the profile and tooth trace modifications, and a tip chamfer or a tip rounding, and specify the depth of immersion of the grinding wheel.

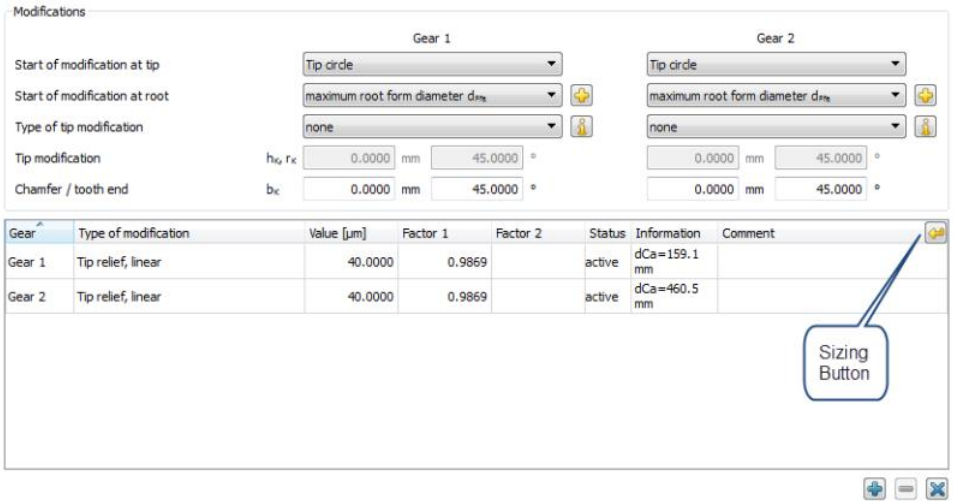


Figure 13.23: `Modifications` input window

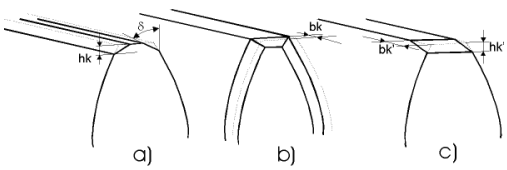


Figure 13.24: `Definition of modifications to the tooth end`

- a) tip chamfer
- b) chamfer at tooth end
- c) tip end chamfer

NOTE:

The tip end chamfer is not specified for gear calculations because it does not affect the strength. However, if an unusually large chamfer is involved, hk' and bk' can be simulated by inputting e.g. hk=0.3*hk'. The standards do not offer any guidance for this.

13.6.1 Define grinding wheel for gear n dialog window

For gears which have an entry for the grinding process (see section "Preliminary treatment and grinding allowance" on page II-319) you can click the button on the right of the `Start modification at root` input field to trigger the grinding process. The most important predefined value in this window is the radius of the tip of the grinding wheel (see Figure 13.24). You can input the remaining data either as coefficients, as lengths or as the diameter.

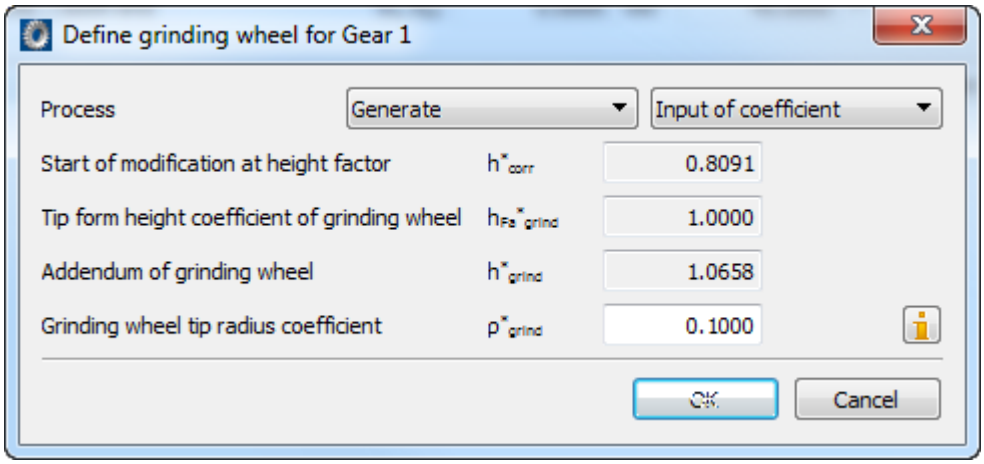


Figure 13.24: `Define grinding wheel for gear n dialog window`

NOTE

Recommendation for "Generate" or "Form grinding" settings:

If you input finished teeth without a preliminary treatment tool, we recommend you select the "Form grinding" process. However, if a preliminary treatment tool is involved, you should select "Generate".

13.6.2 Type of modification

To create a new entry in the list of corrections to be performed, click the button. Double-click on a cell in the `Type of modification` column to open a drop-down list if you want to change the value in that cell. Figure 13.25 shows an extract of the range of possible tooth corrections.

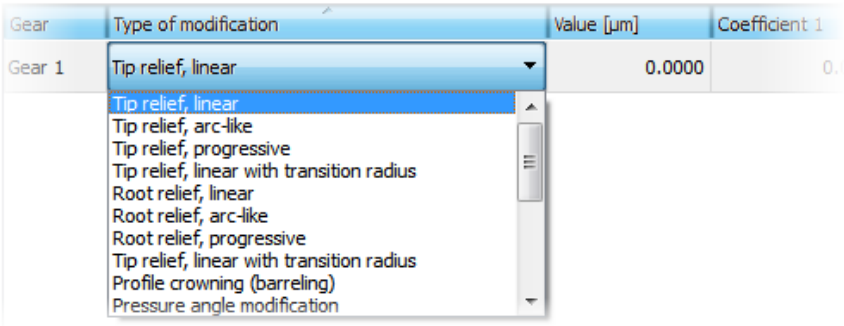


Figure 13.25: `Type of modification` drop-down list

The next two sections, 13.7.3 (see section "Profile modifications" on page II-331) and 13.7.4 (see section "Tooth trace modifications" on page II-336), provide descriptions of the corrections defined in ISO 21771.

Input different corrections for right or left flank: to do this, go to `Settings > General` and set the `Unsymmetrical Profile modifications` flag.

Defining the right-hand/left-hand tooth flank (in accordance with ISO 21771):

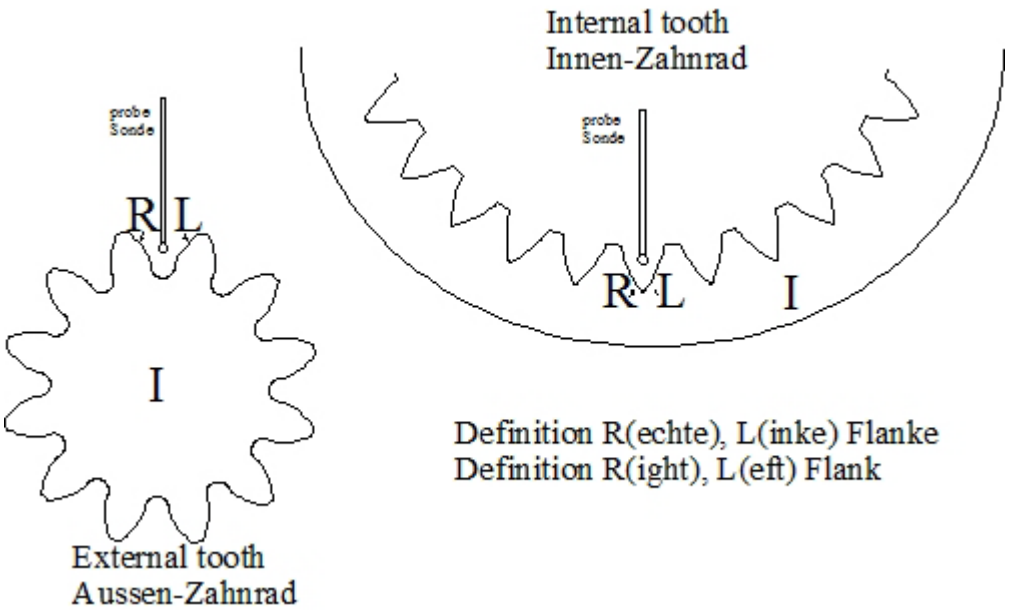


Figure 13.26: Tooth flank definition

13.6.3 Underlying principles of calculation

The geometry of straight or helical cylindrical gears is calculated in accordance with ISO 21771 or DIN 3960. Many manuals and standards use very similar methods to calculate this geometry. In addition to calculating the geometry, it is very useful to have information about how to check for defects (undercut, insufficient active profile, etc.). Technical documentation provided by tooling manufacturers or machine tool manufacturers may also contain information about this.

Measurements for tooth thickness and backlash can be selected in accordance with different standards, such as ISO1328 (1970 edition) or DIN 3967. Manufacturing tolerances can also be defined using standards such as ISO 1328, AGMA: 2000, AGMA: 2015, DIN 3961 or DIN 58405 to suit the particular situation.

Strength is calculated in accordance with, for example, ISO 6336 or DIN 3990, by verifying common defects (tooth root fracture, pitting, scuffing, micropitting). These standards include the most comprehensive and detailed calculation methods currently available. There are two methods that can be used to calculate safety against scuffing. The integral temperature method of calculating scoring resistance is mainly used in the automobile industry whereas the flash temperature method is

used in turbo gearbox manufacturing. It has not yet been established which of these two methods is the more reliable.

Micropitting is calculated in accordance with ISO 15144, Method B. This method is very reliable for gears without profile modifications. However, in the case of gears with profile modification, it has been specified that the tip relief Ca must correspond to the optimum tip relief Ceff (as proposed in the standard). If not, the verification must be performed without taking the modification into account. This is a significant disadvantage because corrections have a considerable effect on micropitting. In this case, you should use Method A (Safety against micropitting using method A).

In the USA, the AGMA 2001 standard must be applied when calculating resistance. This calculation method differs so much from the method specified in DIN 3990 that the results cannot be compared. In addition, numerous different methods are used to calculate the resistance of plastic gears.

One of the problems with applying DIN 3990 is the wide range of different calculation methods it contains. There are around 10 different calculation methods that can be applied between Method A (exact calculation involving measurements) and method D (the simplest, rough calculation). It is therefore no surprise that very different results can be obtained from applying calculations in accordance with DIN 3990 or ISO 6336 to the exact same gear wheel. Whenever possible, KISSsoft uses the most detailed formulae for dimensioning and analyses during this calculation procedure. This procedure corresponds to Method B. However, calculations performed using different programs may also give very different results. It also takes a lot of time and effort to investigate the precise reasons for this. It is therefore much more effective and efficient to use a reference program to perform the comparison. One such program is the ST+ cylindrical gear program package developed by the FVA (Forschungsverein Antriebstechnik, (Research Society for Transmission Techniques, Germany)), at the Technical University in Munich. For this reason, KISSsoft provides the `As in the FVA program (DIN 3990)` option, which supplies the same results as the calculation with the FVA code (see section "Calculation method" on page II-266). The differences between results obtained by KISSsoft and the FVA are negligible. They are due to the minor differences between the FVA program and the regular version of DIN 3990. If requested, we can provide you with a number of different documents to help you compare these methods.

Other interesting results are taken from Niemann's book [65]:

- Gear power loss with gear loss grade H_V in accordance with equation (21.11/4)
- Average friction factor μ_m in accordance with equation (21.11/6) with $1 \leq vt \leq 50m/s$

Gear power loss P_{VZ} in accordance with equation (21.11/3)

13.6.4 Profile modifications

profile modifications are actually variations of the involute and are known as height corrections. The following sections describe which profile modifications are included in the KISSsoft system.

Note: before you can define height corrections, you must first input the length factor L_{Ca*}. The length factor is the pitch length Ly (from the tip or root form diameter) divided by the normal module: $L_{Ca*} = L_y/mn$. The pitch length L_y is calculated in accordance with ISO21771, Equation 17, or DIN 3960, Equation 3.3.07.

13.6.4.1 Linear tip and root relief

Figure 13.26 illustrates `tip relief`. The constantly increasing amount of material removed in the transverse section, starting at d_{Ca}, up to the tip circle, refers to the theoretical involute. The same applies to the `root relief`.

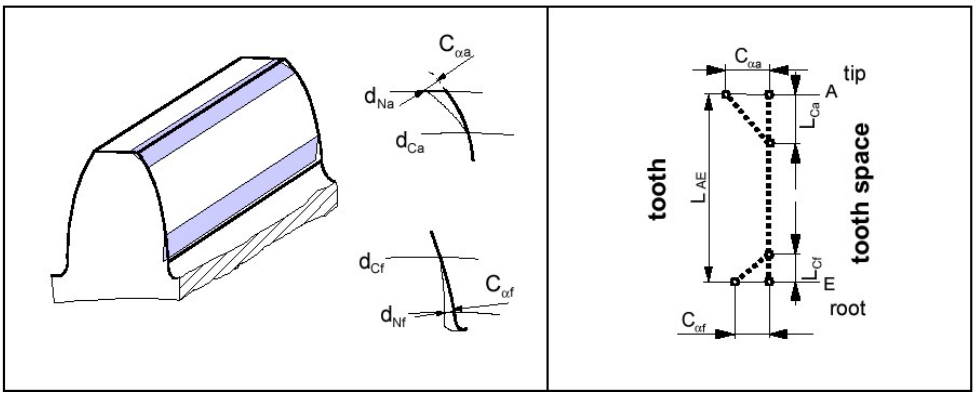


Figure 13.26: Linear tip and root relief

where

d_{Na}	Active tip diameter	d_{Nf}	Active root diameter
d_{Ca}	Modification end diameter (tip)	d_{Cf}	Modification end diameter (root)
L_{Ca}	Resulting tip relief length	L_{Cf}	Resulting root relief length
$C_{\alpha a}$	Tip relief	$C_{\alpha f}$	Root relief
A	Tip neighboring point	E	Root neighboring point

L_{AE} Resulting tooth height length[1)]

[1)] Corresponds to the meshing length g_α

In the KISSsoft system, input the size value $C_{\alpha a}$ in the `Value` input field, for tip relief. The `Coefficient 1` input field defines the quotient from the calculated tip relief length L_{Ca} and normal module m_n. Similarly, to represent root reliefs, input the values for $C_{\alpha f}$ and the quotient from L_{Cf} and m_n.

NOTE

In the "Modifications" tab you can specify that the modification starts at the root. The figure shows the situation when the modification starts at the active root diameter d_{Nf}.

13.6.4.2 Arc-like profile modification

The method used here is similar to the one used for a linear profile modification. The difference is that this method involves approximating an arc of circle which starts at the point where diameter d_{Ca} intersects with the unchanged tooth profile. The tangents of the arc of circle are identical to the tangent of the unchanged tooth profile at this point. The benefit of this modification is that the tangents do not change abruptly in the unchanged tooth form - circular pitch approximation transition point.

Figure 13.27: Arc-like profile modification

L_{Ca}	Resulting tip relief length	L_{Cf}	Resulting root relief length
$C_{\alpha a}$	Tip relief	$C_{\alpha f}$	Root relief

13.6.4.3 Progressive profile modification

The procedure used here is similar to the one used for a linear profile modification. The progressive profile modification is also detailed in the description of tooth form options (see Progressive profile modification (see page II-355))

Figure 13.28: Progressive profile modification

L_{Ca}	Resulting tip relief length	L_{Cf}	Resulting root relief length
$C_{\alpha a}$	Tip relief	$C_{\alpha f}$	Root relief

13.6.4.4 Linear tip and root relief with transition radii

Figure 13.29 illustrates `tip relief`. The constantly increasing amount of material removed in the transverse section, starting at d_{Ca} up to the tip circle, refers to the theoretical involute. The same applies for the `root relief`.

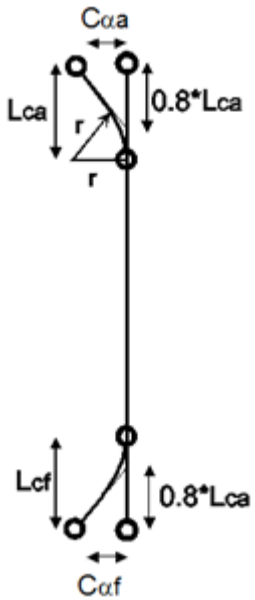


Figure 13.29: Linear tip and root relief with transition radii

L_{Ca}	Resulting tip relief length	L_{Cf}	Resulting root relief length
$C_{\alpha a}$	Tip relief	$C_{\alpha f}$	Root relief

This is why the KISSsoft in the KISSsoft system, input the value $C_{\alpha a}$ in the `Value input` field, in the case of tip relief. The `Coefficient 1` input field defines the quotient from the calculated tip relief length L_{Ca} and normal module m_n. Similarly, to represent root reliefs, input the values for $C_{\alpha f}$ and the quotient from L_{Cf} and m_n.

13.6.4.5 Profile crowning (barreling)

`Profile crowning (barreling)` is where a constantly increasing amount of material is removed in the transverse section in the direction of the tip and root circle, starting at the middle of the calculated tooth flank length. Points A, E and the value C_a define the arc-like progression.

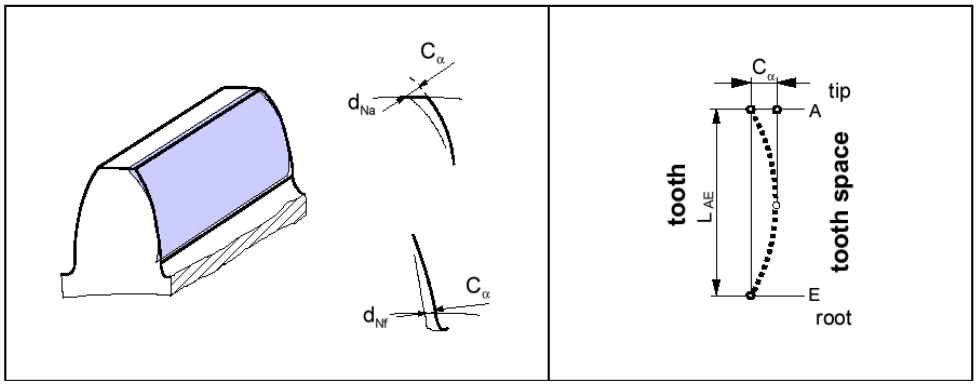


Figure 13.27: Profile crowning (barreling)

where

d_{Na}	Active tip diameter	d_{Nf}	Active root diameter
C_α	Profile crowning (barreling)	L_{AE}	Resulting tooth height length[1)]
A	Tip neighboring point	E	Root neighboring point

[1)] Corresponds to the meshing length g_α

This is why the KISSsoft enter the value `Ca` in the ς α λ υ ε input field.

13.6.4.6 Eccentric profile crowning

In the "Modifications" tab you can add eccentric profile crowning to the tooth profile.

The definition of eccentric profile crowning corresponds to the definition of eccentric crowning, where coefficient 1 is determined from the difference between the starting diameter of the modification at the tip. Here, you should note that the modification is defined by the diameter, not by the length of path of contact. Therefore, if you input a value of 0.5 for coefficient 1, this does not correspond to the profile crowning, because this should run symmetrically to the center point of the path of contact (dSm).

13.6.4.7 Pressure angle modification

You define the `pressure angle modification` in a similar way to tip/root relief (see section "Linear tip and root relief" on page II-331). However, the difference here is that the value C_H applies over the entire tooth depth (see Figure 13.28).

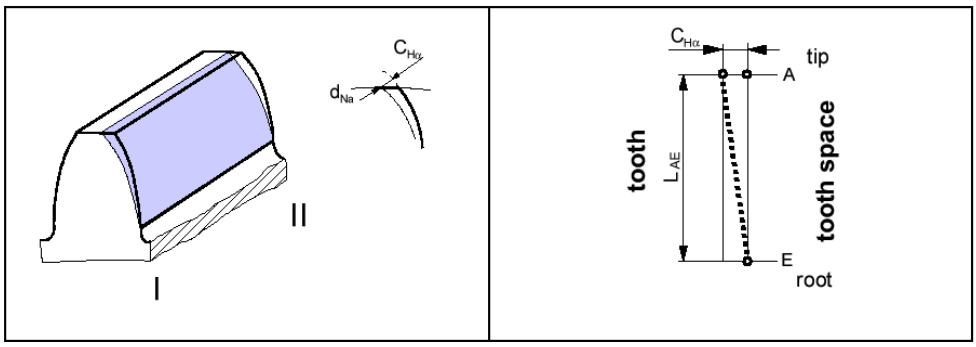


Figure 13.28: Pressure angle modification

where

d_{Na}	Active tip diameter	$C_{H\alpha}$	Pressure angle modification
A	Tip neighboring point	B	Root neighboring point
L_{AE}	Resulting tooth height length[1)]		

In KISSsoft enter the value $C_{H\alpha}$ in the `Value` input field.

13.6.5 Tooth trace modifications

Tooth trace modifications are variations across the facewidth. The following sections describe which tooth trace modifications are implemented in the KISSsoft system.

13.6.5.1 Linear end relief I and II

A `linear end relief` is the constantly increasing removal of material from the tooth trace, starting from particular points, in the direction of the front and rear face surface. In this case, the numbers for I and II relate to both face surfaces (see Figure 13.29).

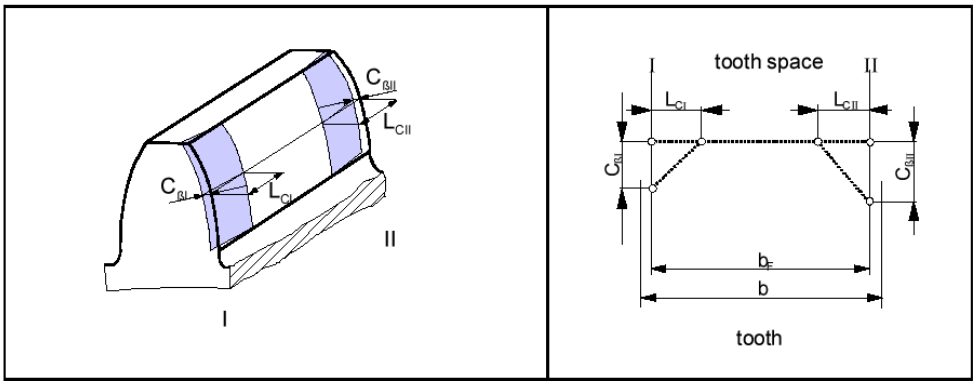


Figure 13.29: Linear end relief I and II

where

	Face I		Face II
L_{CI}	End relief length	L_{CII}	End relief length
$C_{\beta I}$	End relief	$C_{\beta II}$	End relief

This is why the KISSsoft system, go to the `Value` input field and enter the value $C_{\beta I(II)}$, in the `Coefficient 1` input field, enter the quotient $L_{CI(II)}$ / b_F where b_F is the facewidth minus chamfer.

13.6.5.2 Arc-like end relief I and II

An `arc-like end relief` is the constantly increasing removal of material from the tooth trace, starting from particular points, in the direction of the front and rear face surface. In this case, the numbers for I and II relate to both face surfaces (see Figure 13.30).

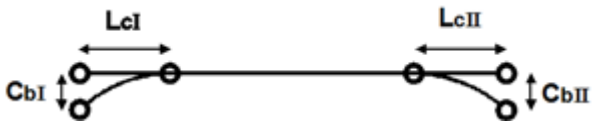


Figure 13.30: Arc-like end relief I and II

where

	Face I		Face II
L_{CI}	End relief length	L_{CII}	End relief length
$C_{\beta I}$	End relief	$C_{\beta II}$	End relief

This is why the KISSsoft system, go to the `Value` input field and enter the value $C_{\beta I(II)}$, in the `Coefficient 1` input field, enter the quotient $L_{CI(II)}$ / b_F where b_F is the facewidth minus chamfer.

13.6.5.3 Helix angle modification

You define the `helix angle correction` in a similar way as end relief (see section "Linear end relief I and II" on page II-337). However, the difference here is that the mass L_{CI} applies over the entire facewidth (see Figure 13.30).

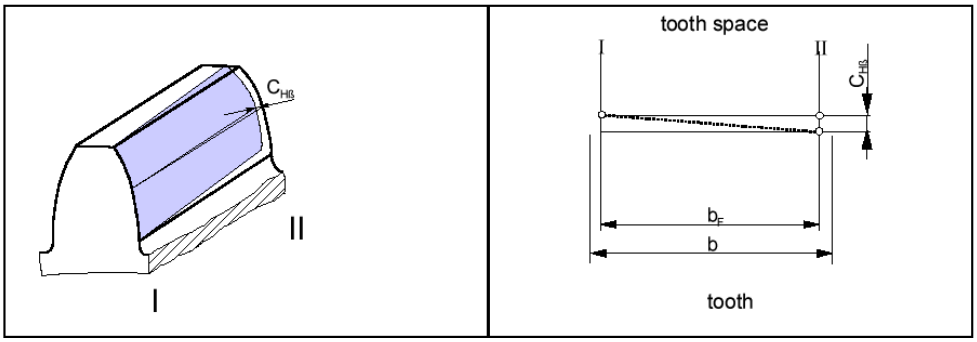


Figure 13.31: Helix angle modification

where

b	Facewidth	b_F	Usable facewidth
$C_{H\beta}$	Helix angle modification		

This is why the KISSsoft enter the value $C_{H\beta}$ in the `Value` input field.

13.6.5.4 Crowning

`Crowning` is where material is removed constantly and symmetrically in the direction of the face surfaces, starting from a common point and where the tooth trace remains constant. The material is removed in an arc-like progression with the maximum at the point$_F$/2.

NOTE

Offset crowning, with its maximum to the right of the point b_F/2, is often used in practice. You can define this modification by inputting centrical crowning with an additional helix angle modification (on page II-338).

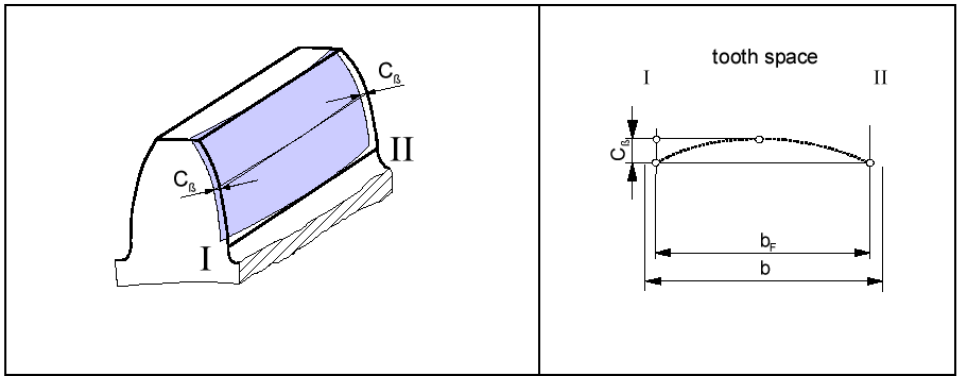


Figure 13.32:Crowning

where

b	Facewidth	b_F	Usable facewidth
C_β	Crowning		

In KISSsoft, go to the `Value` input field and enter the value C_β.

13.6.5.5 Eccentric crowning

In the "Modifications" tab you can add eccentric crowning to the facewidth.

For eccentric crowning, the Value defines the amount of modification and the Factor 1 defines the modification position from the side I divided by the facewidth. The modification is defined as a part of arc that the center is located along the vertical line defined by the Factor 1. The radii are shown in the Information field according to your input. If you define the Factor 1 as 0.5, the modification is equivalent with general Crowning.

13.6.5.6 Triangular end relief I and II

The corners are broken.

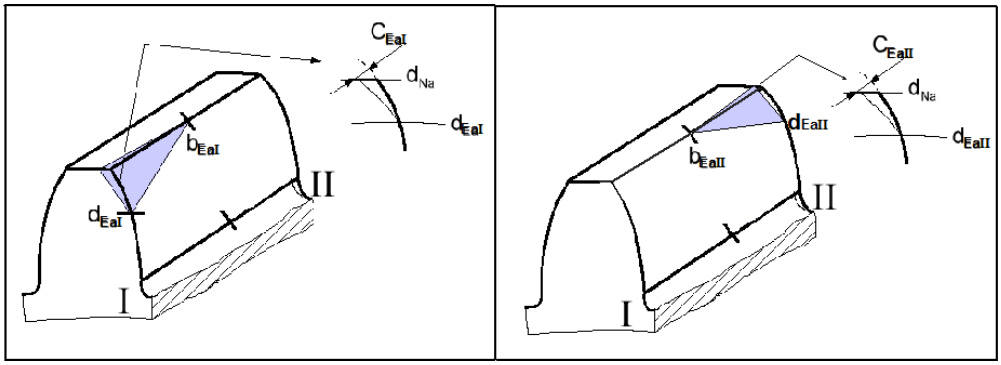


Figure 13.33: Triangular end relief I (left) and II (right)

where

C_{Ea}	Tip relief	d_{Ea}	Modification end diameter
L_{Ea}	Resulting triangular end relief length	b_{Ea}	Triangular end relief length
d_{Ef}	Modification end diameter	b_F	Usable facewidth

This is why the KISSsoft enter the value C_{Ea} in the input field. Then go to the `Coefficient 1` input field and enter the quotient of L_{Ea} / m_n. Then go to the `Coefficient 2` input field and enter the quotient of b_{Ea} and facewidth b.

13.6.5.7 Twist

`Twist` is the torsion of the transverse section profile along a helix. Usually, the angle increases in a linear progression from the start of the effective flank to its end. A positive directional torsion moves clockwise away from the observer. See also Figure 13.34. Modification C can be input as either a positive or negative value.

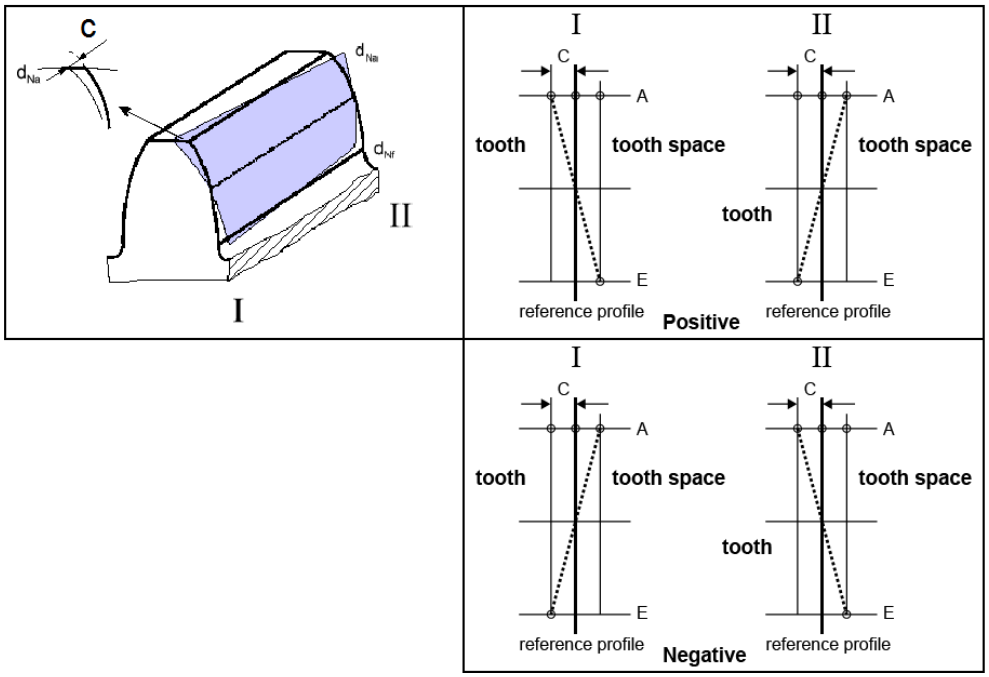


Figure 13.34: Twist

where

C	Relief on dNa at I		
dNa	Active tip diameter	dNf	Active root diameter

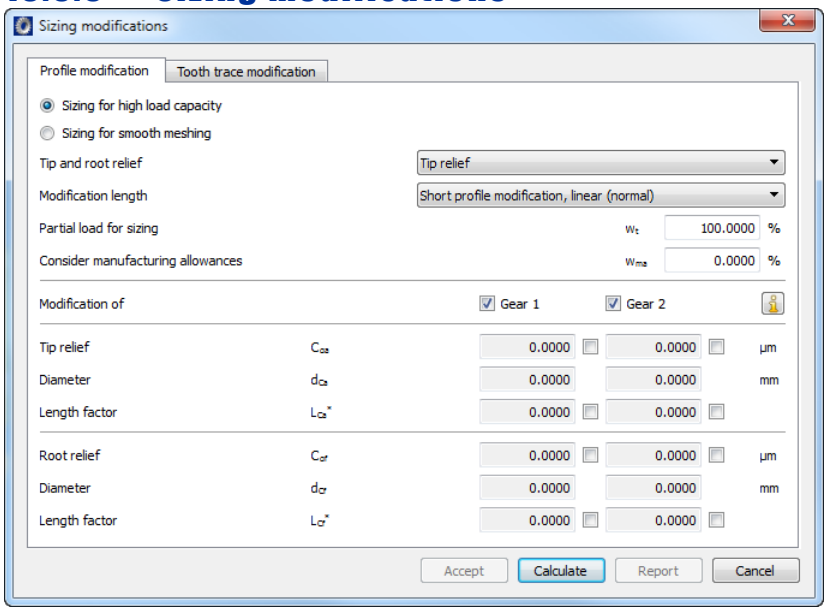
The notation used here is also shown in sections 13.8.4.2 (see section "Helix angle modification" on page II-338) and 13.8.3.4 (see section "Pressure angle modification" on page II-336).

13.6.5.8 Topological modification

The Topological modification option allows you to define any type of modification. The actual modification is described in the file that is to be imported. You will find an example of this type of entry in the "topological_template.dat" in

the dat directory. The file is self-explanatory. You can define coefficients in any slice and for any rolling depth. When the file is imported, these coefficients are multiplied by the value entered under Ca. You can display and check the modification by clicking `Graphics > 3D Geometry > Modifications`.

13.6.6 Sizing modifications



Click the button, as shown in Figure 13.23 on page II-326, to open the `Sizing modifications` dialog. The next two sections describe the basic method for performing profile and tooth trace modifications.

13.6.6.1 Profile modification

a) Tip relief on the driven gear reduces the entry impact, whereas tip relief on the driving gear reduces the exit impact. Tip relief is therefore usually applied to both gears. It is only applied to the driven gear alone in exceptional circumstances.

b) When calculating the profile modification, you must always specify the tip chamfer. If not, the active involute will not be included in the calculation.

c) Tooth contact stiffness is always calculated in accordance with the selected calculation method. Alternatively, you can derive the contact stiffness from the tooth form (see page II-277).

d) The points along the length of path of contact are labeled in accordance with ISO 21771. In a situation involving a driving pinion, a tip correction must

be applied on the pinion from H -DE to E (or D to E) and on a gear, from A to H -AB (or from A to AB). For a driven pinion, the descriptions are swapped in accordance with ISO 21771 (A becomes E, E becomes A).

e) KISSsoft calculates the tip relief value for a nominal torque that was changed by a modification value. In the case of gears that do not always have the same operating torque, the modification value is assumed as approximately 50-75% of the maximum moment, evenly distributed across the pinion and the gear. The default value for tip relief C_α is defined using the mean value of the data as defined by Niemann. A (somewhat greater) value is set as the meshing start (C.I) at the tip of the driven gear. The value (C.II) is set as the value for the meshing end at the tip of the driving gear. When you select profile modification `For smooth meshing`, the value C.I is also set at the meshing end.
For deep toothing, where $\varepsilon_\alpha > 2$, the load-dependent portion of tip relief is reduced, depending on accuracy grade, to 12.5% (for quality level 8 and poorer) and up to 50% (for quality level 5 and better).

f) KISSsoft also calculates the modification length. The "long modification", as it is known, goes from point A to point B of the length of path of contact, but the "short modification" only goes to point H-AB (the midpoint between A and B). Usually the short modification is selected. However, the modification length (from A to AB) should not be too short. A minimum length (related to the tooth depth) of $0.2m_n$ should always be present. This value is checked during sizing. If the length from A to AB is too short, the program prompts you to use a minimum height of $0.2m_n$. However, the result of this is that the contact ratio in the unmodified part will be less than 1.0 (< 2.0 for deep toothing where $\varepsilon_\alpha > 2$). The program then displays an appropriate message.

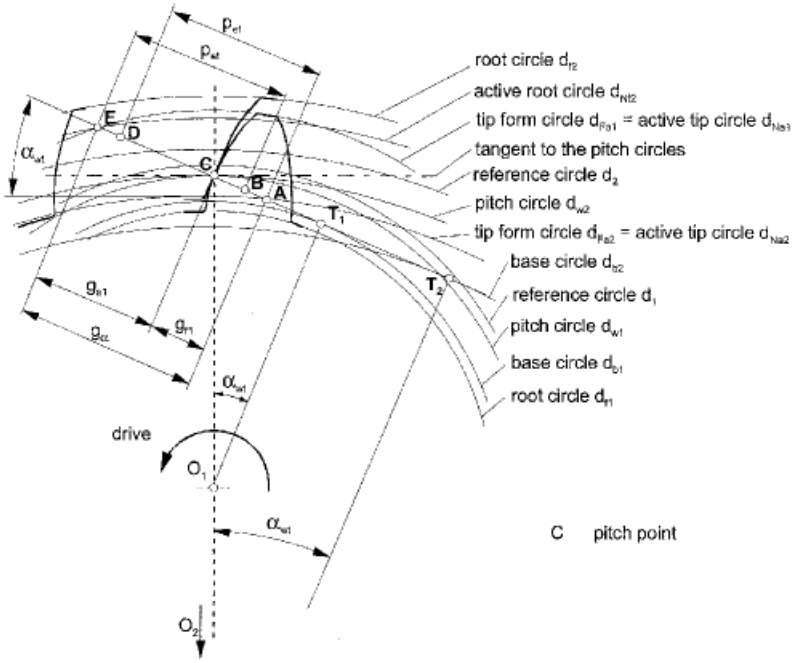


Figure 13.34: Length of path of contact for a cylindrical gear

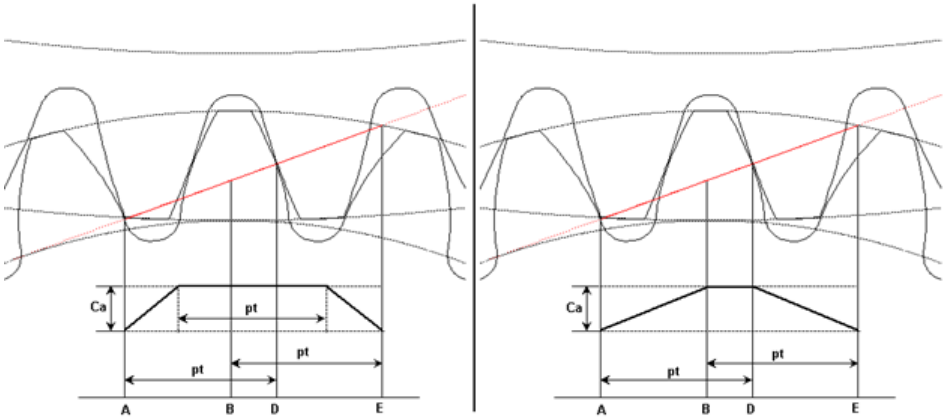


Figure 13.35: Short (left) and long profile modification

g) The type of `Profile modification` has an effect on how scuffing safety (see section "Relative structure coefficient (scuffing)" on page II-279) is calculated.
If you select`For high load capacity` in accordance with the suggestion stated in Niemann, the profile modification at the end of the contact (point E on the path of contact) is somewhat less than that at the beginning of the contact.
If you select `For smooth meshing`, the profile modification at the end of contact is set to the same values as that for the beginning of contact.

13.6.6.2 Tooth trace modification

The procedure you use to size a width modification, for example, an end relief (see section "Linear end relief I and II" on page II-337) or crowning (on page II-339), is performed as specified in ISO 6336, Part 1, Annex B.

If you are working with planet systems, the proposed tooth trace modification can be used to compensate for a misalignment of the planet and the sun. It can also take into account the effect of torsion on a particular gear. You will find more detailed information about the direction of torque and the axis alignment in the "Defining the misalignment of individual parts" section.

However, be aware that this sizing suggestion only applies to planets with a symmetrical misalignment because of the torsion that influences the beam.

The proposed modifications (KHβ = 1) are only then correct if the system has a single planet. If several planets are present, the program searches for the best compromise so that the proposed modification minimizes the maximum KHβ for all the planet contacts.

13.6.7 Notes about profile modification

If you select a `short profile modification` , the length of the modification at the tooth tip (or at the tooth root) for both gears is defined such that the contact ratio of the part of the tooth flank that has not been changed by the modification is still exactly 1.0 (for deep toothing with $\varepsilon_\alpha > 2$ is still exactly 2.0). This type of profile modification is the one most frequently used because it always ensures a sufficiently large transverse contact ratio (no matter what load is involved).
This short profile modification runs from point A of the path of contact up to the point H-AB (the midpoint between point A and point B), or from E to H-DE. However, the result of this is that the contact ratio in the unmodified part is 1.0.

However, if you want to design a gear unit that runs as quietly as possible, it is usually better to select the long profile modification because the transmission error is usually much lower in this case. To properly evaluate the effect of a profile modification, we recommend you calculate the meshing under load (see section "Contact Analysis" on page II-371).

13.7 Tooth form

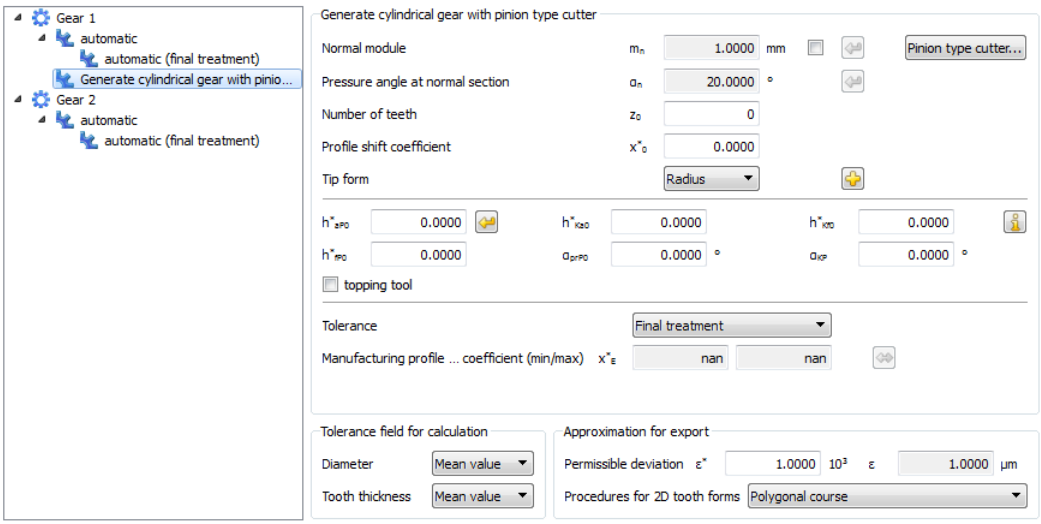


Figure 13.36: `Tooth form` input window

In addition to the actual calculation, the tooth form calculation offers a number of other options because it simulates the manufacturing process with a precisely defined cutter. These options include:

- tooth form modifications with profile modifications and root contour optimization
- taking into account several steps in the manufacturing with different tools
- calculating the cutter (pinion type cutter or hobbing cutter) required to manufacture the toothing (for example, for tooth forms that have been imported from a CAD program or for modified tooth forms)
- tooth form modifications for injection molds or for use in manufacturing pinion type cutters

NOTE

Special tutorials that specifically deal with tooth form modifications have been designed and provided for use. You can download these tutorials from our website, `http://www.kisssoft.ch`.

The `Tooth form` calculation module input window consists of two columns. The left-hand column shows which operations are to be performed on the gears. The right-hand column consists of the `Tolerance field for calculation`

and `Approximation for export` groups and the corresponding operations group.

13.7.1 Context menu

Click the right-hand mouse button in the operation directory structure group to open a context menu. This menu refers to the active element (shown with a blue background) in the directory.

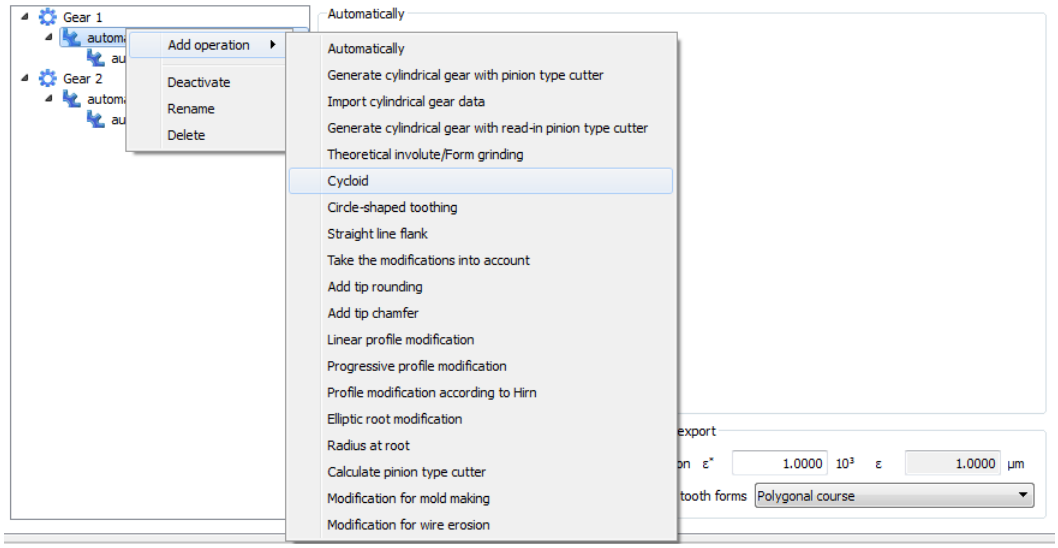


Figure 13.37: Context menu in the tooth form calculation

The context menu gives you these selection options:

- **Add operation** Select this menu item to open a sub-menu that lists the operations (see page II-349) that can be performed on a particular gear.
- **Choose as result** This result is usually displayed in the graphic and used in the strength calculations. The default setting is for the results of the last operation to be displayed here, unless the modification involves mold making, wire erosion, or a pinion type cutter.
- **Activate/Deactivate** Use this option to remove an operation that has been assigned to a gear from the list without deleting it. The icon is then marked with a red cross. The `Activate` menu item returns a deactivated operation to the list of active operations. The red cross then disappears.

- **Rename** Changes the name of an operation. Note that if you change the name of an operation this does not change the area name in the right-hand sub-window.
- **Delete** Permanently removes an operation entry along with all its associated parameters.

13.7.2 Operations

You can calculate the tooth form by using a combination of various different operations. You can apply one processing step after another, for example, using a hobbing cutter or a pinion type cutter and applying modifications such as rounding or profile modifications. You can label each operation to make it easy to identify at a later point in time.

13.7.2.1 Automatically

The default operation for the tooth form calculation is `Automatically`. The tooth form (with all its preliminary and final treatments) is then generated using the data entered in the Standard tabs (see page I-86). If you have defined modifications, these are taken into account when generating the tooth form. You can also disable this part of the operation in the context menu. The same applies to any tip chamfer or rounding you specify. If you select ZA as the flank shape, a ZA worm will be generated. Otherwise a ZI worm is created.

NOTE

If the `Automatically` operation has been disabled, none of the data input in the `Reference profile` or `Modifications` input windows will be taken into consideration.

13.7.2.2 Generate cylindrical gear with hobbing cutter

To generate a cylindrical gear with a hobbing cutter, input the gear reference profile. When you add this operation, the window is filled automatically using the values you defined in the `Reference profile` input window. If the tool is a non-topping tool, the addendum of the reference profile is determined automatically from the tip circle and not transferred from the values you input. For special applications (manufacturing a gear with a cutter with a different module) you can modify the module mn and the pressure angle αn. You can then use the sizing buttons. The sizing buttons ( ) calculate the correct value in each case for the specified base circle. Click the `Cutter...` button to open the `Define`

`cutter` (see page II-313) window which displays a list of tools. To define the tolerance field, you can either enter the generating profile shift coefficients directly (`Own inputs`) or use the pretreatment or final treatment tolerances.

The milling cutter data can also be input as factors or as absolute lengths (mm or inch). These selection options make your job much easier if the milling cutter data are the lengths (in mm or inches) given in a drawing.

When sizing haP0*, the system calculates the value which is then used to generate the involute up to the active root diameter. The proposed value shown here is the exactly calculated value, to which 0.05 is added (to obtain a small distance between the root diameter and the active root diameter).

If you use the sizing button to define the grinding wheel, the radius ρaP0 should be small (e.g. 0.1*mn), otherwise the grinding process may reach the root radius.

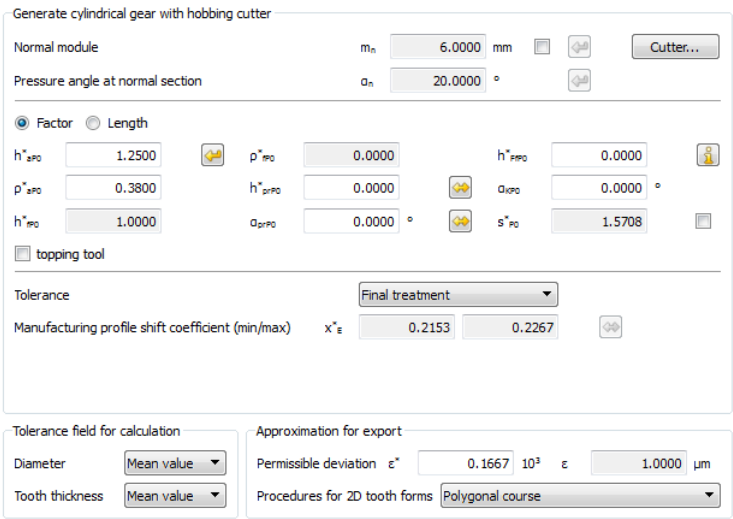


Figure 13.38: Operation: `generate cylindrical gear with hobbing cutter`

NOTE

The hobbing cutter information entered here is independent of the data specified in the `reference profile` input window. In other words, the tooth form calculation is based exclusively on the values defined in the `Tooth form` input window.

13.7.2.3 Generate a cylindrical gear with an imported hobbing cutter

You can import the cutter contour from the CAD system in either `dxf` or `vda` format. To do this, define 1/2 tooth from the tip at A to the root at E:

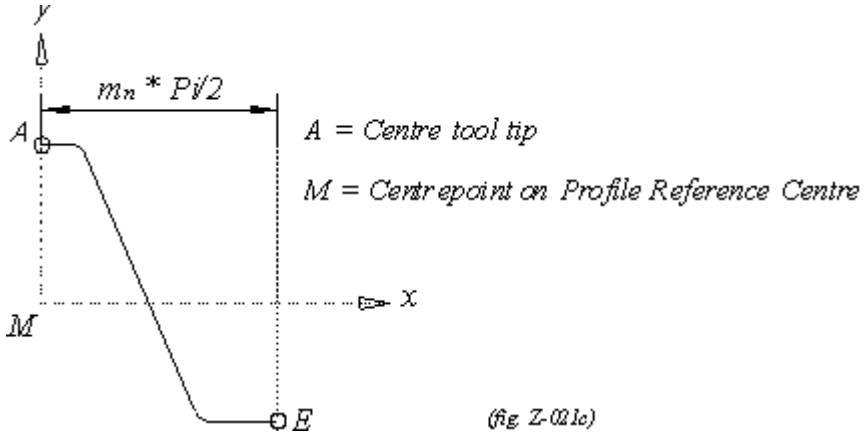


Figure 13.39: Tool profile

You can either specify the layer that includes the contour or select `ALL` for all the data. You can then decide whether to import the tool in transverse section or in normal section, and also change the module. The profile shift coefficient you enter here determines the tooth thickness.

Click on the "Cutter for displaced generation" option to select a normal module for the tool that differs from the cylindrical gear generated by the program.

Click on the "Input data as a reference" option to modify the module in the drawing. The cutter is then scaled to the normal module specified in the basic data.

13.7.2.4 Generate a cylindrical gear with a pinion type cutter

You must define the geometry of the pinion type cutter if you want to calculate the tooth form of gears manufactured using a shaping process.

Required input data:

- Reference profile of the pinion type cutter
  In the reference profile of the pinion type cutter, swap the values of the tip and root used in the reference profile of the work gear at $x_0 + x_E = 0$. In other cases, you need to input a displacement at x0.
- Z_0 Number of teeth on the pinion type cutter

- x_0 Profile shift for the pinion type cutter
  (if x_0 is not known, you can use the cylindrical gear calculation to define the profile shift from the tip diameter or the base tangent length → further info (see section "Profile shift coefficient" on page II-253))
- or determine the length of the chamfer on the pinion tooth tip s or the radius of the rounding r on the pinion tooth tip (see Figure 13.40)

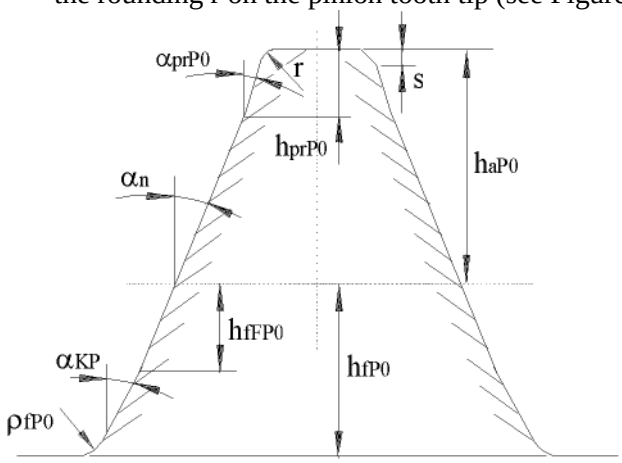


Figure 13.40: Tool profile

13.7.2.5 Generate a cylindrical gear with an imported pinion type cutter

You can import a pinion type cutter as a `*.dxf` or `*.vda` file. In this situation, the system imports a half tooth from the specified layer (select ALL for all layers), as shown below:

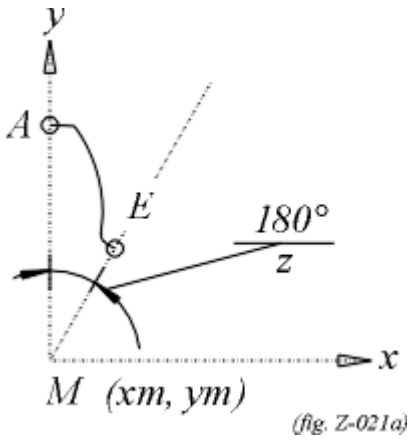


Figure 13.41: Pinion type cutter coordinates

A	:	Mid tooth tip: Start of contour
E	:	Middle tooth space: End of contour
M	:	Center point (x_m, y_m is a required entry)
z	:	Number of teeth

Click on the "Input data as a reference" option to modify the module in the drawing. The cutter is then scaled to the normal module specified in the basic data.

NOTE

The file (`dxf` or `vda`) must only have contours A to E in the layer you can specify for reading (importing). In this case, you must specify the number of teeth on the pinion type cutter and the manufacturing center distance.

13.7.2.6 Reading (importing) a cylindrical gear

You can import a cylindrical gear directly as a `*.dxf` or `*.vda` file. To do this, define a half tooth in the selected layer:

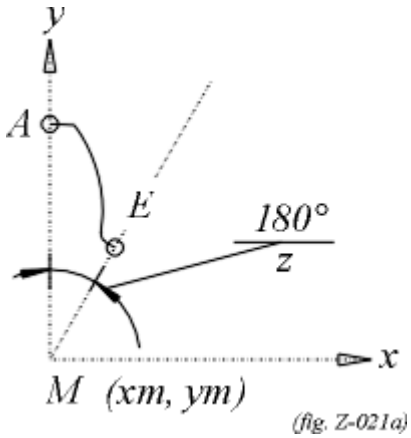


Figure 13.42: Coordinate system for the import

A	:	Mid tooth tip: Start of contour
E	:	Middle tooth space: End of contour
M	:	Center point (x_m, y_m is a required entry)
z	:	Number of teeth

Click on the "Input data as a reference" option to modify the module in the drawing. The cutter is then scaled to the normal module specified in the basic data.

NOTE

The file (`dxf` or `vda`) must only have contours A to E in the layer you can specify for reading (importing).

13.7.2.7 Adding tip rounding

You can add tip rounding as a tooth form modification. The rounding can be added either in the transverse or axial section.

13.7.2.8 Adding tip chamfer

You can add a tip chamfer as a tooth form modification. The chamfer can be added either in the transverse or axial section and is defined by the starting diameter and an angle.

13.7.2.9 Linear profile modification

In a linear profile modification, the tooth thickness is reduced in a linear progression from the starting diameter to the tip (relief C_a on each flank as a tooth thickness modification).

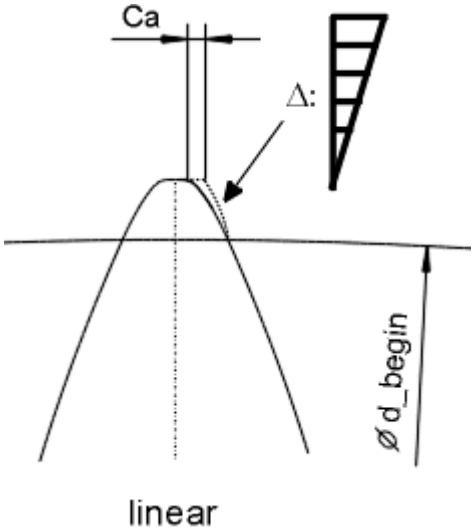


Figure 13.43: Linear profile modification

13.7.2.10 Progressive profile modification

In a progressive profile modification, the tooth thickness is reduced from a starting diameter to the tip (relief C_a on each flank as a tooth thickness modification) in accordance with

$$\Delta A_s = 2 \cdot C_a \left(\frac{d - d_{begin}}{d_a - d_{begin}} \right)^{factor/5} \tag{13.21}$$

. The coefficient controls the course of the relief. A coefficient of 5 represents a linear relief. For more information, see also Figure 13.44. If a coefficient greater than 5 is used, the progressive profile modification moves tangentially into the unmodified tooth flank. This is the preferred option if larger reliefs are to be achieved. We do not recommend you use a coefficient of less than 5 (some of these lower values are simply ignored by the program). Coefficients greater than 20 are also ignored. In this case, a coefficient of 20 is used.

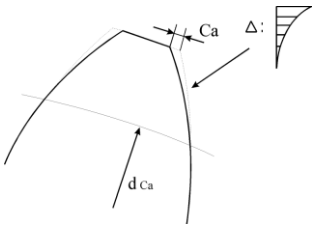


Figure 13.44: Progressive profile modification

13.7.2.11 Entry curve as specified by Hirn

An entry curve that passes into the involute tangentially is applied to the tooth tip starting from the specific diameter d_{begin}. This entry curve consists of three arcs of circle. The bend in the curve increases from arc to arc so that the final curve is tangential to the tip circle. This modified tooth form (also called a hybrid tooth) has significant benefits, because it permits extremely quiet running despite relatively imprecise production methods. For this reason the modification is applied for plastic products, for preference. See Figure 13.45.

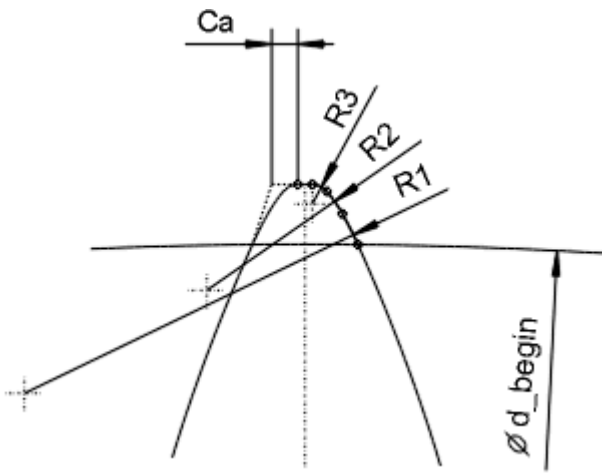


Figure 13.45: Profile modification as specified by Hirn

An entry curve is usually only applied to deep toothing with transverse contact ratios of greater than 2.1. In addition, KISSsoft can use its sizing function to suggest a suitable starting point (diameter) for the entry curve and the tip relief value. To do this, it uses the profile modification (see section "Modifications" on page II-326) calculation.

The start of the entry curve is defined as follows:

- For a transverse contact ratio of 2.0: The active involute is reduced until the transverse contact ratio is exactly 2.0.
- For a transverse contact ratio of less than 2.0: The diameter is calculated so that a medium tip relief is created, i.e. a transverse contact ratio of above 1.0 is reduced by approximately 50%.
  For example, from 1.8 to 1.8 - 0.5 **.** 0.8 = 1.4.

The exact definition is shown here:
For a transverse contact ratio > 2.0 : d_{start} = minimum (d_{PunktD}, $d_{PointE0.2}$)
For a transverse contact ratio < 2.0 : d_{start} = minimum (d_{PunktE}, $d_{PunktE0.2}$)

The relief C_a at the tip is defined as shown here:

- For normal crest widths less than 0.21 **.**m_n: 0.5 **.** Tooth thickness - 0.01 **.**m_n
- For normal crest widths greater than 0.21 **.**m_n: 0.10 **.**m_n to 0.12 **.**m_n

13.7.2.12 Elliptical root modification

The root fillet is replaced by an ellipse-shaped contour, which progresses tangentially in the flank and root circle. The aim is to achieve the greatest possible radius of curvature. The course of the contour can be influenced by the coefficient in the range 1 ÷ 20. Click the sizing button for the diameter to select an active root diameter as the starting point of the modification. The definable length on the root circle is then set to > 0 if you want an area of the tooth form to run on to the root circle. For example, this is a good idea if the root circle is to be measured with measuring pins.

The greater tooth thickness in the root area means that the generation process with the other gear in the pair must be checked.

13.7.2.13 Root radius

The root contour is replaced by an exact arc of circle with a specifically definable radius. After you make this modification, check the generation process using the other gear in the pair.

13.7.2.14 Theoretical involute/form grinding

The tooth form is construed mathematically. The involute is defined using the module and pressure angle along with the tip and root diameter. The tooth thickness is defined by the profile shift coefficients. You can also define a root radius (in the transverse section). This option is suitable for involute gears that cannot be manufactured by a gear generation process (e.g. internal gears with 4 teeth) or for a processing step involving form grinding.

13.7.2.15 Cycloid

You can select a cycloid as a special tooth form. The cycloid is defined with two rolling circles and the tip and root diameters. In the main calculation, the tooth thickness is defined by the allowances. Rolling circle 1 rolls on the inside on the reference circle and therefore cuts the dedendum flank. Rolling circle 2 rolls on the outside and generates the tip. Rolling circle 1 of the first gear should correspond to rolling circle 2 of the second gear. Sizing a cycloid toothing is made easier if you calculate the other gear in the pair using the data of the first gear during the optimization process.

Use the Stress curve and Kinematics analyses modules to analyze the strength and geometry properties of cycloid toothings.

13.7.2.16 Circular pitched teeth

The circular pitched teeth special toothing type can be defined using the tooth flank radius and the tooth thickness at the reference circle. An arc of circle is created in the root area.

A classic arrangement of circular pitched teeth, for example, as specified in NIHS 20-25 [67] consists of an arc of a circle with radius r starting from the reference circle, a straight line that progresses in the direction of the center of the gear below the reference circle, and a full root rounding.

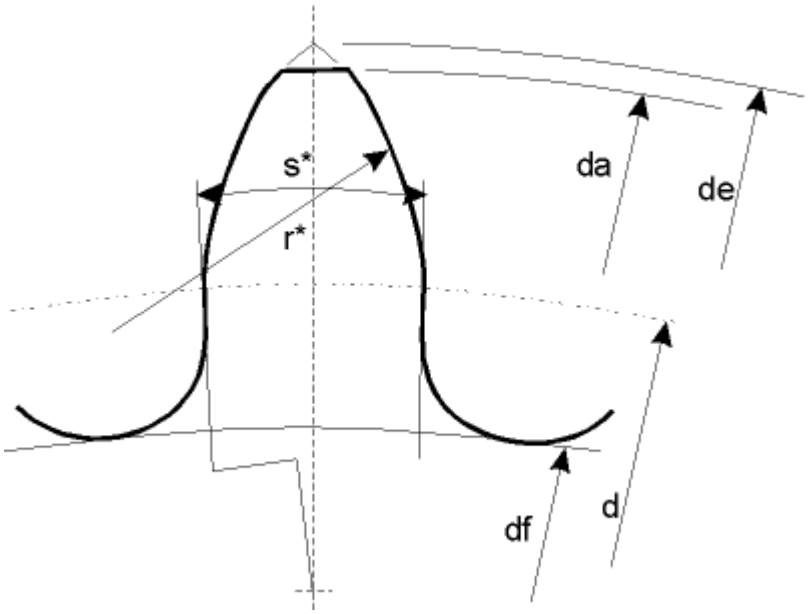


Figure 13.46: Arcs of circle on the tooth

13.7.2.17 Straight line flank

You can select a straight line flank as a special tooth form. The straight line flank is defined by the tooth thickness at the reference circle (theoretical toothing), the space width angle in transverse section, the tip and root diameter as well as the manufacturing profile shift coefficient (dependent on the tolerance). You can also predefine radii for tip and root rounding.

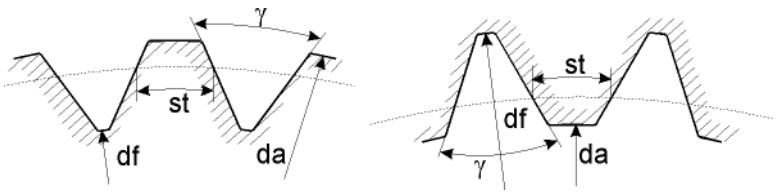


Figure 13.46b: Straight line flank

13.7.2.18 Generate with the other gear in the pair

You can use the other gear in the pair to calculate the tooth form on all the gears, except on gear 1 (gear number - 1). In this case, you can overwrite the manufacturing center distance and the tip circle. The clearance between the gears can be generated either by reducing the manufacturing center distance or by inputting the circumferential backlash. The tip clearance is achieved by increasing the tip circle of the tool.

13.7.2.19 Calculating the reference profile

You can calculate the reference profile of an existing tooth form. A hobbing cutter can then be used to manufacture it. The manufacturing center distance can be changed in this calculation. This has a significant effect on the practicability of creating a tooth form using the generation process. In contrast, the value you input for the profile shift has no effect on the profile. Instead this influences the null point.

The calculated reference profile is then used as a cutter to calculate the cylindrical gear again. By comparing the two tooth forms you can then evaluate the extent to which the tooth form can be manufactured using the generation process. Click `Cutter/Tool` to display the reference profile in the graphic.

13.7.2.20 Calculating a pinion type cutter

You can calculate a pinion type cutter for an existing tooth form. To do this, enter the number of teeth on the pinion type cutter and the manufacturing center distance. The center distance has a significant effect on the practicability of creating a tooth form using the generation process. Try out a number of different values to find the best one.

The calculated pinion type cutter is then used as a cutter to calculate the cylindrical gear again. By comparing the two tooth forms you can then evaluate the extent to which the tooth form can be manufactured using the generation process. Click `Cutter/Tool` to display the pinion type cutter.

13.7.2.21 Generating a face gear with a pinion type cutter

This operation is not yet available. To generate a face gear select the `automatic` option. Define the pinion type cutter in the `Reference profile` input window.

13.7.2.22 Generate a rack with a hobbing cutter

Once again, enter the rack's reference profile, as you do when generating a cylindrical gear using a milling cutter. In this case, the addendum is only relevant if you are using a topping tool. The profile shift is measured, starting from a reference line, which is defined by the rack height minus the reference profile addendum in the main screen.

The profile shift coefficients can be either input directly or defined by the premachining and final treatment tolerances.

13.7.2.23 Generate a rack with imported hobbing cutter data

You can define a hobbing cutter as a `*.dxf` or `*.vda` file. In this case, the contour must be output as follows so that the KISSsoft system can read the data correctly:

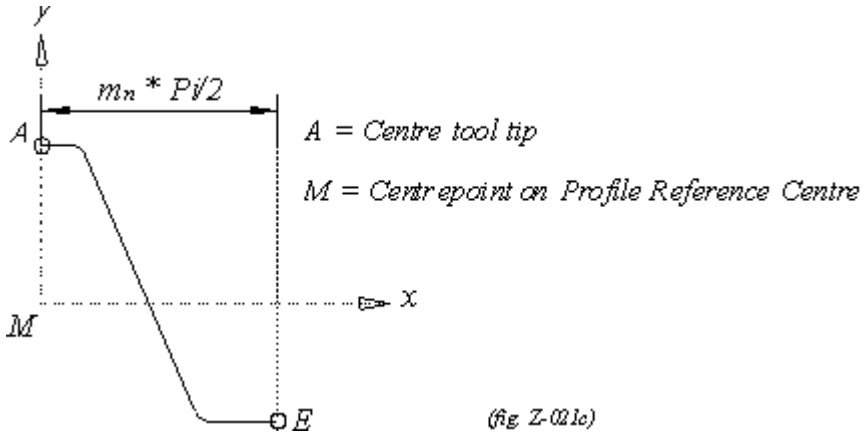


Figure 13.47: Tool profile

NOTE

The file (`dxf` or `vda`) may only have contours A to E in the layer you can specify for reading (importing).

In addition to the contour, you must also define the manufacturing center distance. In this case, the reference line for the center distance is defined using the rack height.

13.7.2.24 Generate rack with a pinion type cutter

Once again, enter the reference profile of the pinion type cutter, as you do when generating a cylindrical gear using a pinion type cutter. The profile shift is

measured, starting from a reference line, which is defined by the rack height minus the reference profile addendum in the main screen.

The profile shift coefficients can be either input directly or defined by the premachining and final treatment tolerances.

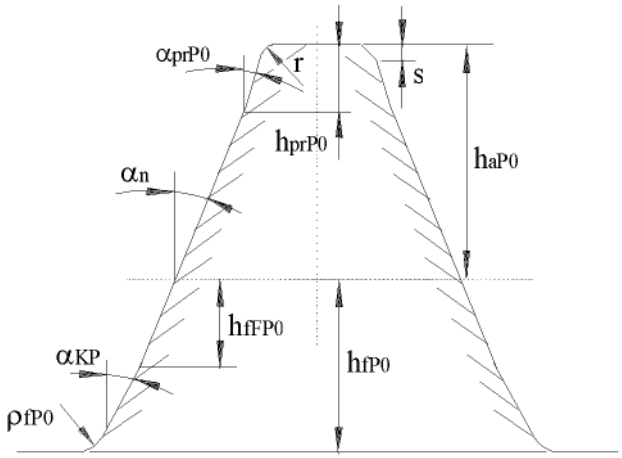


Figure 13.48: Cutter tooth geometry

13.7.2.25 Generate rack with imported pinion type cutter

You can generate a rack with an imported pinion type cutter. In this case, you must specify the number of teeth on the pinion type cutter and manufacturing center distance in addition to the pinion type cutter contour in `*.dxf` or `*.vda` format.

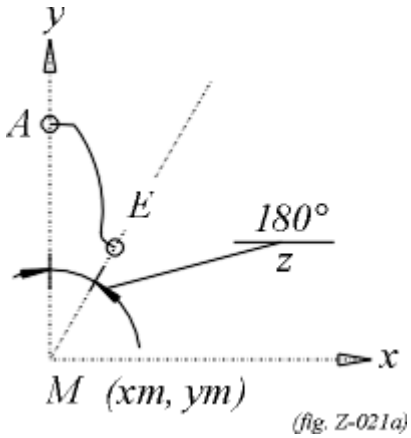


Figure 13.49: Coordinate system for the import

A	:	Mid tooth tip: Start of contour
E	:	Middle tooth space: End of contour
M	:	Center point (x_m, y_m this is a required entry)
z	:	Number of teeth

NOTE

The file (`dxf` or `vda`) may only have contours A to E in the layer you can specify for the import.

13.7.2.26 Reading (importing) the rack

You can import a rack gear directly as a `*.dxf` or `*.vda` file in the following format:

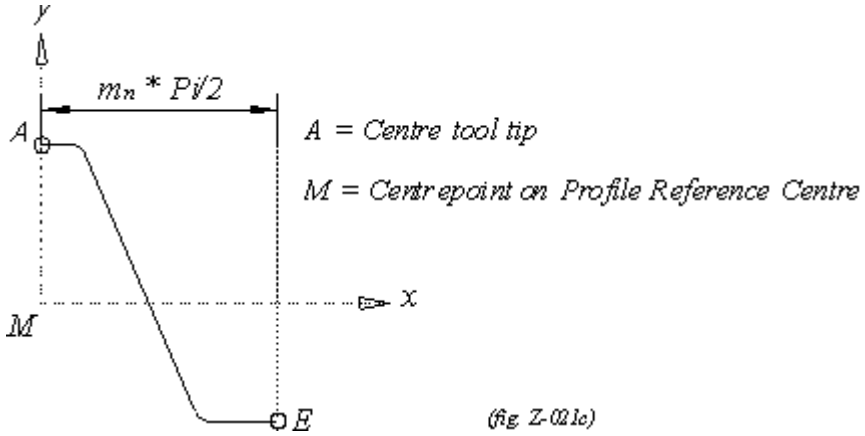


Figure 13.50: Tool profile

NOTE

The file (`dxf` or `vda`) may only have contours A to E in the layer you can specify for reading (importing).

13.7.2.27 Generate a SA worm

This function is currently only available as the `automatic` option.

13.7.2.28 Reading (importing) a worm into the axial section

You can also import a worm in its axial section. In this case the contour is basically the same as the contour of the hobbing cutter, apart from the null point which forms the axis of the worm.

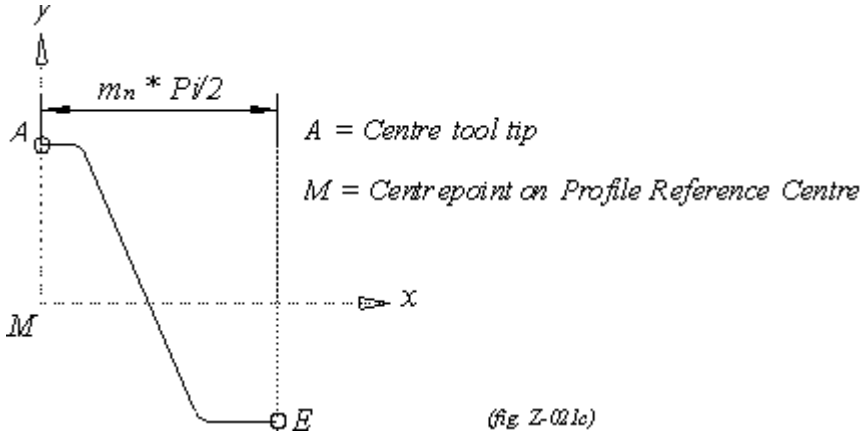


Figure 13.51: Tool profile

NOTE

The file (`dxf` or `vda`) may only have contours A to E in the layer you can specify for the import.

13.7.2.29 Modification for mold making

When plastic gears are manufactured using the injection molding process, the material shrinks as it cools. To counter this effect, and manufacture precise tooth forms, the size of the cutter must be increased by the shrinkage amount. Depending on what type of material is involved, shrinkage may occur either radially or tangentially. If you enter the same values in the radial and tangential directions, the strain will be uniform in all directions

If the gear is injection molded around an inlay body, you must also input the external diameter of this body. The radial strains will then calculated using the "outside diameter of inlay body".

The modifications only affect the transverse section of the tooth form. No strain in the axial direction is present when a 3D volume model is generated. If you want to create an expanded 3D model of a helical toothed gear (if the strain is to be the same in all three axes), you can achieve this by scaling the module (m_n), the center distance and the facewidth.

EXAMPLE

In the main screen, increase the module, center distance and facewidths by the required strain coefficient.
Coefficient 1.02

$$m_n' = 1.02 \cdot m_n$$

$$a' = 1.02 \cdot a$$

$$b_1' = 1.02 \cdot b_1$$

$$b_2' = 1.02 \cdot b_2$$

Then, do not input a value for strain in the tooth form calculation.
This modification also increases the lead p_z by the same coefficient. However the angle of rotation of the spirals across the facewidth remains the same.

Usual values are:

- Radial shrinkage approx. 2%
- Tangential shrinkage approx. 2%

13.7.2.30 Modification for wire erosion

In the erosion process, the electrodes must maintain a specific distance from the required shape, because additional material is removed due to the spark gap. This is usually taken into account by the machines involved in the wire erosion process.

When sink eroding an injection mold the eroding wire must therefore be thinner than the required shape by the amount of the spark gap. If a gear shaped electrode is used, the tooth will be correspondingly thinner. To achieve this, enter a negative value for the spark gap. Usual values for the spark gap are 0.03 to 0.07 mm.

After this modification you can also calculate the reference profile in the next step to determine the shape of a hobbing cutter for the electrodes.

NOTE

You can also use the wire erosion modification to check the practicability of using the wire erosion method. If the aim is to erode external teeth, enter one modification with a positive wire radius and then the second with a negative radius. If the aim is to erode an injection mold for external teeth, first input a negative radius and then run a modification with a positive radius. By comparing the tooth forms you can then see whether the form can be manufactured, or whether a practical form can be created using these two steps.

13.7.2.31 Modification for pinion type cutter

The effective cutting angle and the draft angle of the pinion type cutter cause a tooth form deformation in the projection of the pinion type cutter in the horizontal plane.
The conversion performed here deforms the tooth form in the horizontal plane so that the projection once again shows the exact tooth form once the pinion type cutter has been manufactured.

By grinding with angle ϕ(effective cutting angle) Q moves to P (see Figure 13.52). If the projection P' is to agree (exact contour in the horizontal plane), P must = Q must in the H plane.

$$PP' = (r_a - r_p) \cdot \tan(\phi) \tag{12.22}$$

$$P'Q = (PP') \cdot \tan(\xi) \tag{12.23}$$

$$somit: \; P'Q = (r_a - r_p) \cdot \tan(\phi) \cdot \tan(\xi) \tag{12.24}$$

where

ϕ	Effective cutting angle
ξ	Tip draft angle in axial section
M	Pinion type cutter axis
r_a	Pinion type cutter tip circle radius
r_p	Coordinate of the point P

Conversion of the tooth form:

Given:	Exact tooth form in polar coordinates P = r (Angle)
Searched for:	Tooth form in H-plane P' = r' (Angle)
Solution:	$r' = r + \tan(\phi) \cdot \tan(\zeta)(r_{a-r})$

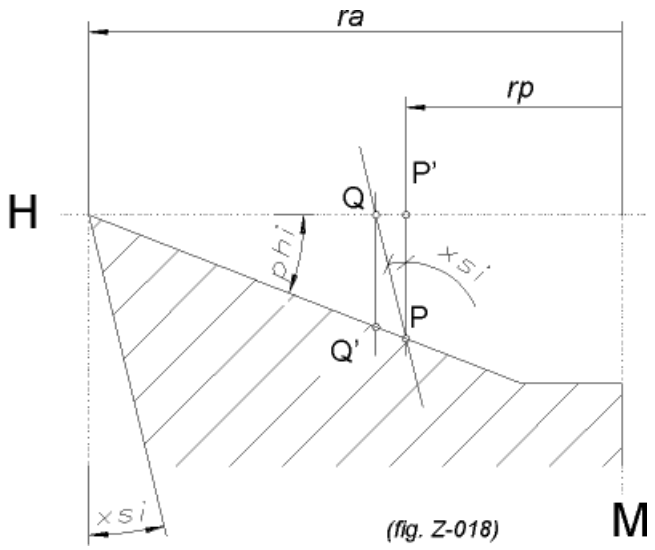


Figure 13.52: Pinion type cutter profile

13.7.2.32 Elliptical deformation

Applicable on the external gear (Gear1) of an internal-external cylindrical gear pair. This allows you to display the elliptical deformation of the race in a special gear box in 2D. Typically z1+z2 = -2 applies here;

The contour of the race is stretched vertically by the lengthening factor and compressed horizontally so that the root circumference of the ellipse matches the root circle circumference of the undeformed gear. In a 2 D display, it is important you check:

- that the gear can be generated without collision over a pitch.

- that opposing sides mesh correctly.

If you need to make a correction, select a different lengthening factor or a different number of teeth (if the total number of teeth is an even number). Values between 0 and 5 % can be used as the lengthening factor.

Note: You cannot create a 3D output for this variant.

13.8 Flank breaking

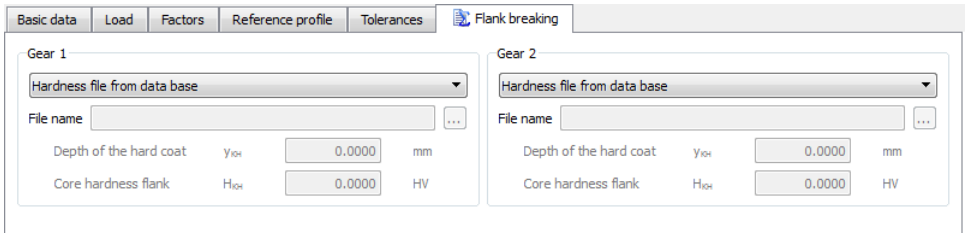


Figure 13.44: "Flank breaking" input window

Flank breaking appears in the area of the active tooth flank instead of in the area of the highest bending stress at the 30° tangent.

This calculation tab is designed to calculate the safety against flank breaking in accordance with Dr. R. Annast [89]. The original calculation procedure from Dr. R. Annast requires detailed measurements of the gear hardness as a function of depth from the flank surface, to enable the depth of transition layer and the core hardness to be calculated.

There are three calculation options:

- Using a hardness file for the gear material, if this file already exists in the database
- Selecting an independent file with the hardness information, or
- Direct input of core hardness and transition depth

If a file is used (case one and two) and only one pair of data is found, then it is assumed (in accordance with case three) that these values are the core hardness and the transition depth.

```
-- index number     depth (mm)  hardness (HV)
-- these lines are comments
DATA
1   0.2 700
2   0.3 675
3   0.4 650
4   0.5 625
5   0.6 600
*   *   *
*   *   *
*   *   *
END
```

Figure 13.45: Structure of the Hardness file

When a file containing hardness data is used (case one and two), the original data are fed to the Annast algorithm. If the algorithm fails due to invalid data, the data is determined in accordance with the following regression formula (non-linear regression) $HV = a \cdot e^{-b \cdot y} + c$

. If the calculation with this data is also unsuccessful, a last attempt is performed (linearized regression) with the equation $\ln(HV) = \ln(a) + \ln(b) \cdot (-y)$

(as above, but without taking the constants into account).

13.9 Contact Analysis

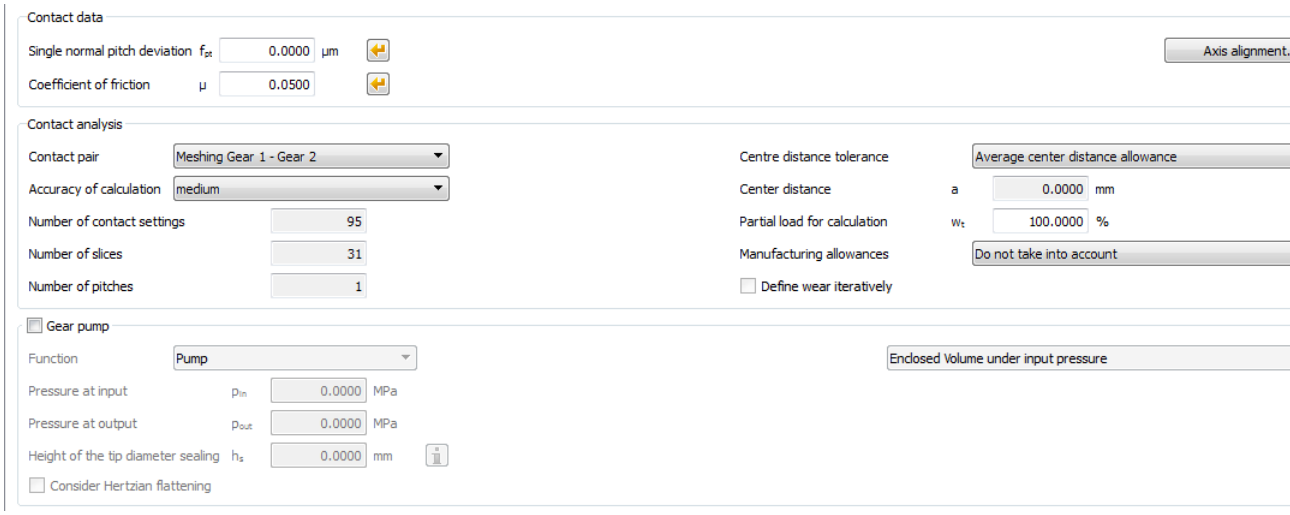


Figure 13.53: Contact analysis input window

The load is taken into account for calculating the path of contact. This also calculates the face load factor $K_{H\beta}$ using the more precise method defined in ISO 6336, Part 1, Annex E (see also Settings, Contact analysis (see section "Contact analysis/Face load factor" on page II-442)). In this case, the meshing stiffness can be calculated either according to Weber/Banaschek [69], ISO 6336-1 or "Own Input". The calculation of meshing stiffness according to Weber/Banaschek is based on the effective tooth form in normal section. For the load, the "Partial load for calculation Wt" coefficient can be entered, so that the torque can be defined. You can also predefine a meshing error. The proposed value for the pitch error is then

$$f_{pt} = 0.5 \cdot \sqrt{f_{pt1}^2 + f_{pt2}^2}$$

calculated. You can specify a value for the pitch error with both a positive and a negative prefix. The results are then displayed if the distance is too large or too small.

The coefficient of friction between the flanks is assumed to be a constant in the meshing. Click the sizing button to accept the coefficient of friction as defined in ISO TR 15144.

For helical gear teeth, non-parallel axes, or tooth trace modifications, the calculation is performed in several slices of spur gear teeth (discretized toothing model) that are linked by a coupling with stiffness. This therefore takes profile and tooth trace modifications into account.

The number of sections is set automatically according to the gear geometry and the "Accuracy of calculation" option. The number of automatically determined steps increases with higher overlap ratio and accuracy of calculation. You can also enter the number of sections, slices and pitches manually by setting the accuracy of calculation to "Own Input". If more than one pitch is calculated, the number of steps is distributed equally across all the pitches.

You can then view the calculation results in the report or in the Graphics > Contact analysis menu. The graphics showing the results are only displayed if a contact analysis has already been performed.

Coefficients K_A, K_V and K_γ are included in the calculation of Hertzian pressure and tooth root stresses.

The contact analysis can calculate either the transmission error as a length on the path of contact in µm or the angle of rotation error as an angle on the driven gear in °.

The effect of manufacturing errors (fma, fHβ) is taken into account with the "Manufacturing allowances" drop-down list in the "Contact analysis" tab. The manufacturing error increases the flank gap in the normal flank direction, see Figure. 1. A linear error distribution is assumed here, so that the manufacturing error on side I is 0, is at its maximum on side II, and increases in a linear progression along the facewidth. Manufacturing errors are taken into consideration in pairs, as either positive or negative values, (up to a total of 5 combinations, but if there is only one combination, the manufacturing error is ignored).

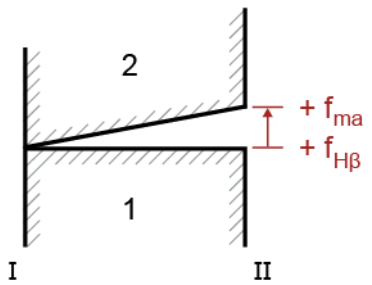


Figure 1: Definition of the positive direction of manufacturing errors fma and fHβ

NOTE:

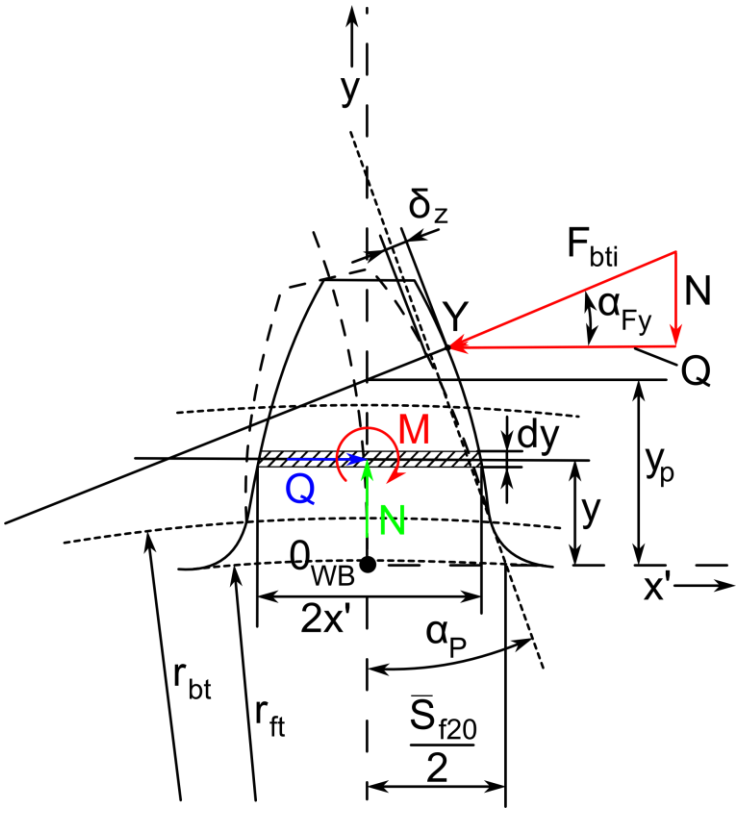
Numerical problems may arise if the selected pitch error is too large relative to the partial load. In this situation, we recommend you select either a smaller pitch error or a larger partial load.

13.9.1 Theory of Contact Analysis

As stated in Weber/Banaschek [69], the deformation of the meshing of gear pairs can be divided into three components:

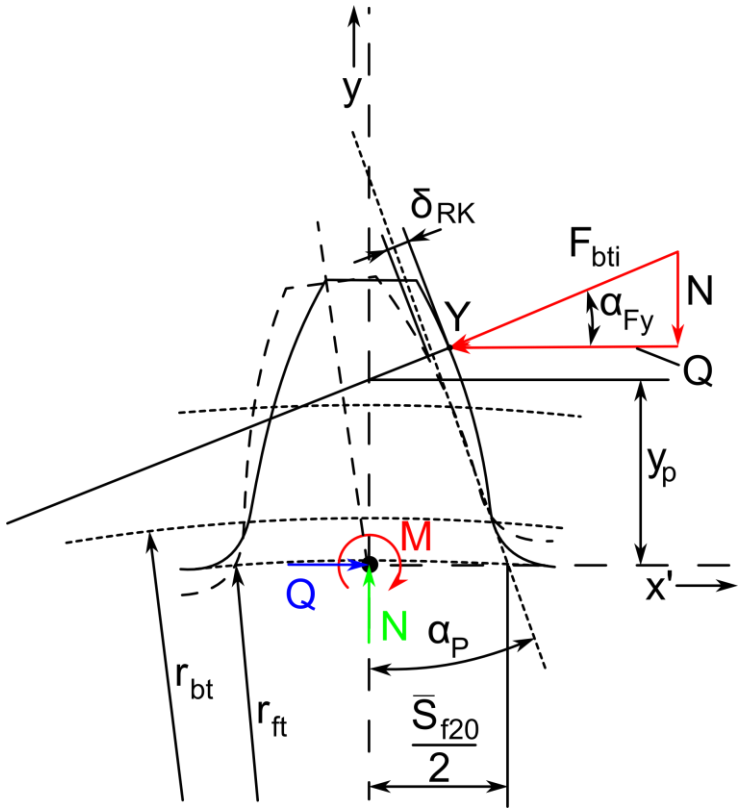
- Gear body deformation
- Bending
- Hertzian flattening

Bending:



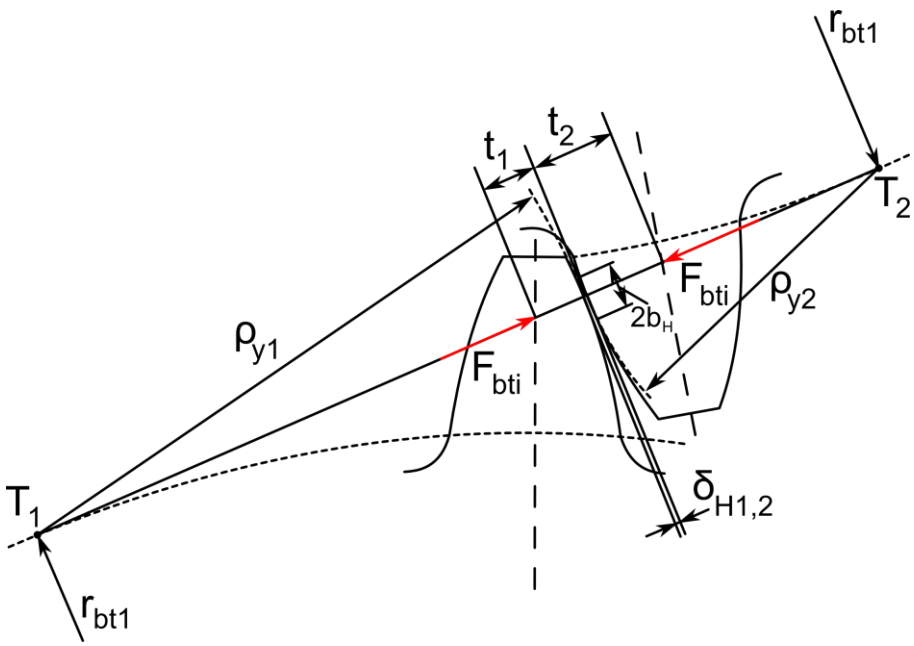
$$\delta_z = \frac{F_{bti}}{b}\cos^2\alpha_{Fy}\,\frac{1-\nu^2}{E}\left[12\int_0^{y_p}\frac{(y_p-y)^2}{(2x')^3}\,dy + \left(\frac{2.4}{1-\nu}+\tan^2\alpha_{Fy}\right)\int_0^{y_p}\frac{dy}{2x'}\right]$$

Gear body deformation:



$$\delta_{RK} = \frac{F_{bti}}{b}\cos^2\alpha_{Fy}\frac{1-\nu^2}{E}\left[\frac{18}{\pi}\frac{y_p^2}{\bar{s}_{f20}^2}+\frac{2(1-2\nu)}{1-\nu}\frac{y_p}{\bar{s}_{f20}}+\frac{4.8}{\pi}\left(1+\frac{1-\nu}{2.4}\tan^2\alpha_{Fy}\right)\right]$$

Hertzian flattening:



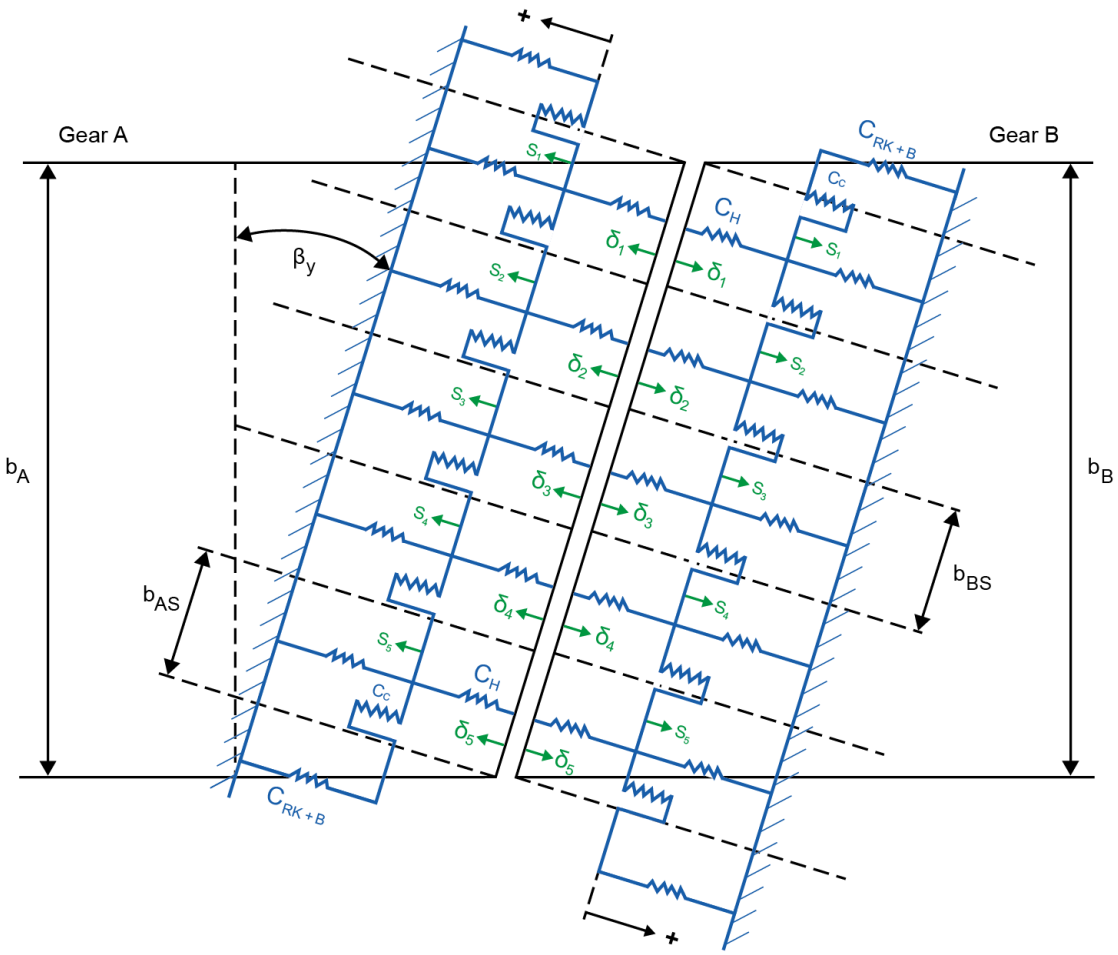
$$\delta_{H1,2} = \frac{F_{bti}}{\pi b_g}\left[\left|\frac{1-\nu_1^2}{E_1}\ln\left(\frac{b_H^2}{4t_1^2}\right)+\frac{\nu_1(1+\nu_1)}{E_1}\right|+\left|\frac{1-\nu_2^2}{E_2}\ln\left(\frac{b_H^2}{4t_2^2}\right)+\frac{\nu_2(1+\nu_2)}{E_2}\right|\right]$$

Total deformation has the effect that the contact point is displaced along the path of contact and the theoretical path of contact is lengthened in relation to the actual path of contact. The transverse contact ratio under load is therefore greater than in the load-free state.

The spring equation F=d*C can be applied to calculate the components of the single contact stiffness from the individual deformation components and the normal force.

13.9.2 Discretized model

A discretized toothing model has been generated so that the deformation theory of meshing in gear pairs developed by Weber/Banaschek can be applied to three dimensional cylindrical gears with helical gear teeth.



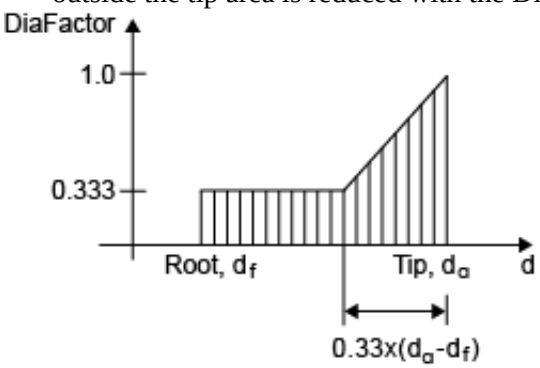
13.9.3 Smoothing the tooth form curvature to calculate Hertzian pressure in the contact analysis

The large variations in curvature that occur during contact cause local, high peak values in Hertzian pressure and are a well-known problem. These values mean that any calculations, such as micropitting in accordance with Method A, which involve Hertzian pressure, will be incorrect. To avoid this, the peak values are filtered out after the calculation so that results that match the actual situation can be achieved.

This problem usually occurs on the tooth tip (where the curvature radius is 0). Therefore we implement the following smoothing strategy to counter the curvature of the tooth form.

If the curvature radius ρ_y is less than 1.01 * m_n^*, then smoothing will be applied to ρ_y. (* current setting in the code)

- SmoothFactor = 0.8 (=0: no smoothing, = 1: full smoothing)
- $\rho_{y+\Delta d}$ and $\rho_{y-\Delta d}$ are calculated.
- Δd = 0.3 * mn : the corresponding diameters are then applied to the diameter, i.e. d+Δd and d-Δd.
- If ($\rho_{y+\Delta d} > \rho_{y-\Delta d}$) then $\rho_{corr} = \rho_{y+\Delta d}$, otherwise $\rho_{corr} = \rho_{y-\Delta d}$,
- As this process has been designed for the critical tip area, the smoothing outside the tip area is reduced with the DiaFactor factor.



- This results in the "smoothed" ρ_y : ρ_{yNew} = SmoothFactor * DiaFactor * ρ_{corr} + (1 - SmoothFactor * DiaFactor) * ρ_{yOld}

13.9.4 Reduced stiffness on the side edges

The bending stiffness of the tooth in helical gears is reduced at the edges.

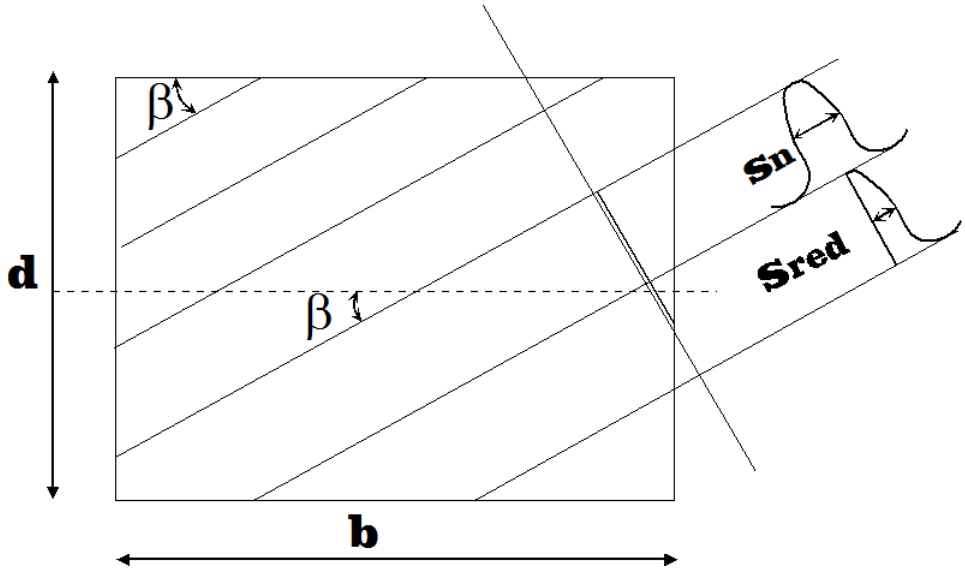


Figure 13.53.3: Illustration of two cuts for a helical gear

Cpet_border = Cpet*(sred/sn)^0.5

Exponent 0.5 was evaluated in comparative analyses with FEM and LVR. The reciprocal value of this exponent (border weakening factor (buttressing) can be changed by the user. It has a significant effect on the buttressing effect that occurs in helical gear teeth.

13.9.5 Linking the individual slices

The teeth are distributed in slices across the width and are linked together by torsional stiffness.

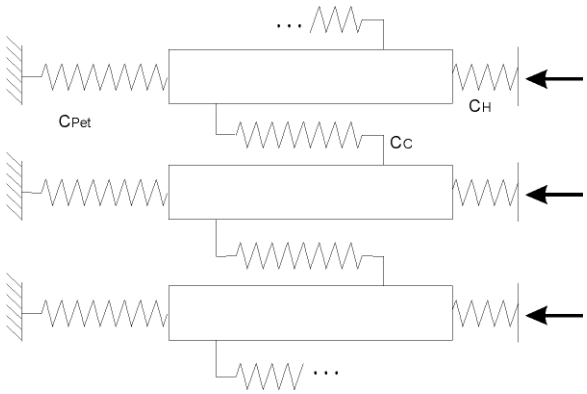


Figure 13.53.2: Linking the slices

Cpet = CZ + CRK = root stiffness as defined by Weber/Banaschek [69]

(CZ = bending stiffness and shear strength as defined by Weber/Banaschek)

(CRK = deformation stiffness due to rotation in the tooth blank)

CH = stiffness from Hertzian pressure as defined by Weber/Banaschek

CC = coupling with stiffness

CC = 0.04*(Asec)^2*Cpet

Asec: Number of slices

All C are in N/µ/mm.

0.04: Empirical coefficient, confirmed by comparative calculations with FEM. The user can change this coefficient (coefficient for linking individual slices) in the `module-specific settings`.

(Asec)^2 is used because different numbers of slices must return the same result over the total width.

13.9.6 Wear iteration

You can use the wear iteration function to define wear along the tooth flank in more detail, because it performs several steps of the contact analysis with the worn

tooth flank. However, this does significantly increase the time it takes this calculation to run. Click the "Define wear iteratively" checkbox to select this option.

You can input a coefficient for the maximum permitted wear per step. This coefficient is then multiplied by the normal module. In the contact analysis shown below, the service life after one iteration was reduced by only applying the maximum permissible wear. The next contact analysis was then calculated with the worn tooth flank. This process was repeated until the total service life was reached.

By clicking on the sizing button, you can set the maximum permissible wear to half the wear calculated for plastic [delWn] for the entire service life. This should be used to perform roughly ten iterations.

13.9.7 Calculation of contact analysis taking the shafts into account

13.9.7.1 Main settings

Instead of directly specifying the deviation and inclination of the axes (linear deformation model), you can use shaft calculation files to define the influence of bending and torsion on the shafts on which the gears are to be mounted more precisely.

The Factors tab is described below. This is where you determine the axis alignment by using the shaft calculation files. In the "File Shaft Gear 1/Gear 2" fields, enter the file name for the shafts to which the pinion (1) or the gear (2) belong. You must input the file name with its entire path (for example C:\MyCalculations\ContactAnalysis\pinion_shaft.W10). However, if the shaft files are stored in the same folder as the gear calculation file Z12, you only need to input the name of the shaft calculation file (as shown in the figure).

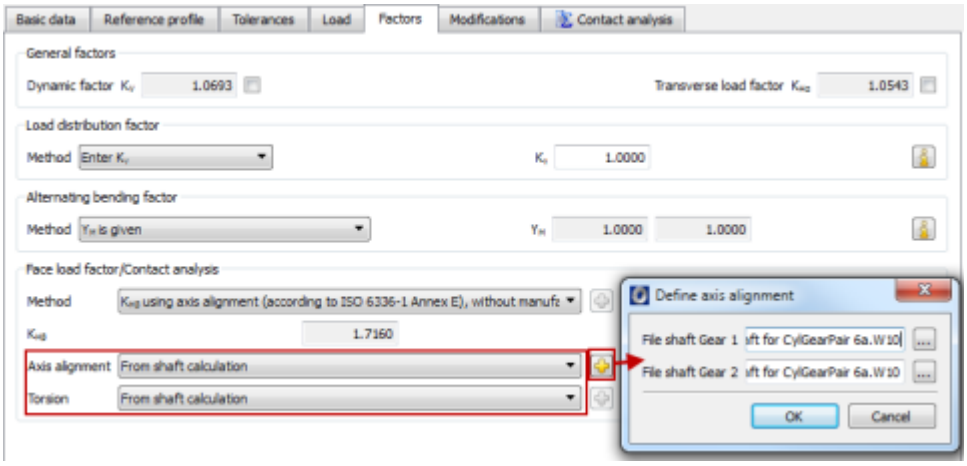


Figure 13.53.5: Factors tab

If several identical gears are modeled on the same position on the shaft (as may happen in a shaft model for a planet system), and KISSsoft cannot find suitable gears when it evaluates the shaft files, you can select the gears you want to use in the "Define gears on shafts" sub dialog.

13.9.7.2 Conditions for using shaft calculation files

If you are working with shaft fields, the sizing parameters in the Z12 gear module must match those in the selected W010 files. More specifically:

1. The pinion geometry (in accordance with Z12) must match the geometry defined for the pinion in shaft file 1. The selection is based on the operating pitch circle (with an error tolerance of 10%), the direction (driving/driven) and the contact flank. The same applies to the gear shaft.
2. The gear pair performance (in accordance with Z12) must (with an error tolerance of 5%) match the gear performance defined in the shaft files.
3. The sense of rotation for both the pinion and the gear (in accordance with shaft files W10) must be consistent. For example, if the pinion rotates in a clockwise direction, the gear must rotate counterclockwise. However, if the gear is an internal gear, both the pinion and gear must rotate clockwise in this example.

From these conditions you can also easily see whether the shaft files can be used for the contact analysis. If one of these conditions is not met, no calculation can be performed.

In addition to the conditions listed above, a number of other conditions (warnings) concerning the helix angle, the facewidth and the gear's working transverse pressure angle are also checked.

The software imports each shaft calculation file (gear 1 for the pinion, gear 2 for the gear) and uses the criteria described above to find the most suitable cylindrical gears. If a number of identical gears, all of which meet the criteria, are present, the program selects the highest gear listed in the Elements tree hierarchy of the corresponding shaft file.

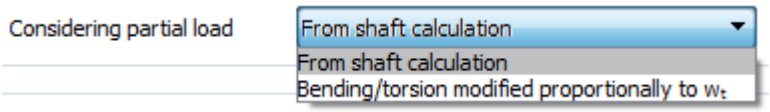
13.9.7.3 Effect of torsion on the body of the gear

You can take the effect of torsion on the body of the gear into account either by applying the results of the shaft calculation or by inputting your own data (the same applies to side I and II). Obviously, the results of the shaft calculation can only be referenced if shaft files have been used to define the axis alignment.

If you defined the gear's torsion in "Own Input", then the torsion moment of resistance is calculated from the root circle df and the inner diameter.

13.9.7.4 Effect of partial load

You can use the partial load coefficient wt (Contact analysis tab) to modify the performance of all the force elements defined in the shaft calculation files, as shown in the following setting. The diagram of bending are always modified by this setting. However, the effect of torsion can only be identified if the setting for torsion (previous section) has been made using the shaft calculation file.



For example, if all the force/power elements in a shaft file are 100N/100W, and the partial load coefficient is 85%, the force elements are calculated as 85N and the power elements as 85W.

13.9.7.5 Handling bending and torsion using the results for the shaft

If a gear pair has been found and the shaft calculations performed successfully, the bending and the effect of torsion are determined from the results for the shaft.

The results for bending in each shaft file are all transferred to a single coordinate framework, where pinion contact occurs at 0° and gear contact occurs at 180°. The torsional angle of each gear is assumed to be 0° on the side that is furthest to the left (side I, i.e. the side with the smallest Y-coordinate in the shaft file) and every torsional angle for this particular gear then refers to this side.

13.9.8 Contact analysis of a planet system

The contact analysis for a planet system is special because it uses a system approach. A set of individual contact analyses is performed for each meshing of the planet with the sun and the rim. For example, in a system containing 3 planets, 3 * 2 = 6 contact analyses are executed simultaneously. This is the reason why this calculation requires so much computing power.

For each angle of the rotating sun, an equilibrium is sought for the whole system. Equilibrium is found when the calculated transmitted power of the sun[6] equals the system power input, and the net torque on all planets is zero (free-floating planets). The initial assumption is that all gears make perfect contact with no load being transmitted.

The following variables are used in Figure 13.53.4:

- φ_s = the angle of the sun
- φ_c = the angle of the planet carrier
- φ_{pi} = the angle of planet i, where i = 1 to N (N is the total number of planets)
- $\Delta\varphi_{SPi}$ = angular offset between point of contact C (sun-planet) of planet i to point of contact C (sun-planet) of planet 1. This angle is measured on the sun. To transform it to the planet angle, use the system kinematics. After the calculation it is output on the planets in the report.
- ψ = the angular offset between point of contact C of the planets in their meshing with the sun, relative to point of contact C with the rim. This value is assumed to be equal for all planets.

The transmission error is measured on the planet carrier, and the value defined for it is such that the system is in equilibrium with the given sun angle φ_s. In addition

[6] *The transmitted power of the sun is the sum of the individual powers transmitted to all planets.*

to the synchronous rotation of all planets (due to system kinematics), each one experiences an additional rotation at a much smaller rate, in order to bring the system into power and torque equilibrium. The positive direction for all angles is in clockwise direction, as shown in Figure 13.53.4.

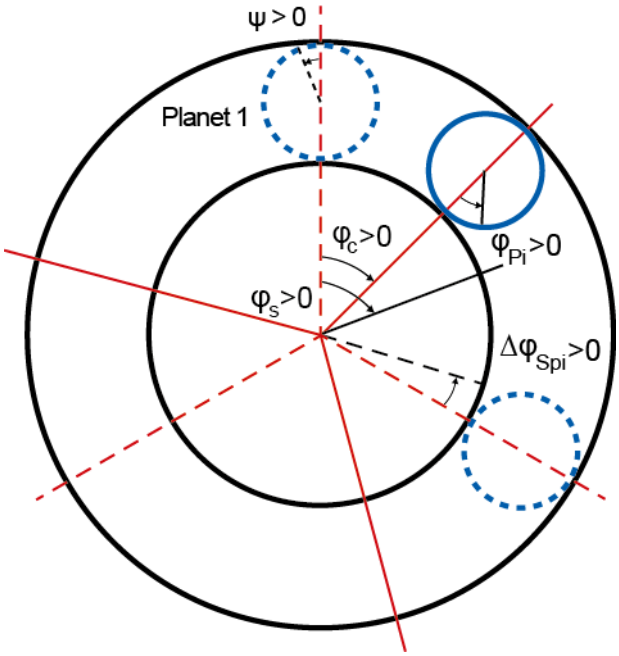


Figure 13.53.4: Model of the planet system

Figure 13.53.4: The variables used for a planetary contact analysis study. The red lines represent the planet carrier (solid line: current position, dashed line: initial position), the blue circles represent the planets, the inner black circle represents the sun, and the outer black circle represents the rim.

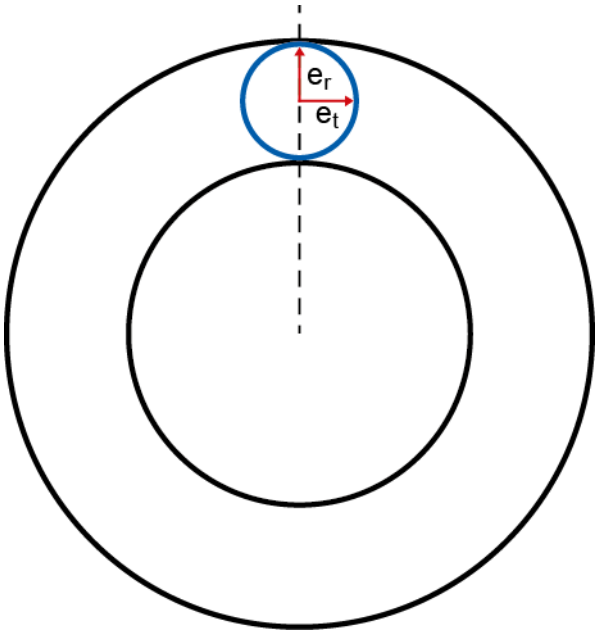


Figure 13.53.5: Definition of radial/axial error

Manufacturing errors of the planet center points can be considered in two directions, radial (e_r) and peripheral (e_t). For information on the definition of these errors see Figure 13.53.5. The radial error results in an effective change of the corresponding pair center distance, whereas errors in the peripheral direction result in an effective increase/decrease of the carried load on the given pair.

Each sun-planet and planet-rim meshing is treated independently. However, for documentation purposes only, the maximum value is documented for the following:

- For sun-planet and planet-rim meshing
  - amplitude of the transmission error, variation of contact stiffness, maximum flash temperature, variation of bearing load
- Gear wear
  - For each gear (sun, planet or rim) the documented value for wear is the maximum value that can occur in all the meshings of this gear. For example, the planet wear takes into account all meshings (with the sun and the rim), but the sun wear only takes into account the sun-planet meshings.

– The average tooth thickness reduction is treated similarly.

In contrast, the minimum value of all pair meshings is used for the safety against micropitting (method A), the minimum specific oil film thickness in the contact area, and the minimum local oil film thickness (related to all pair meshings). The corresponding mean value for all meshings is used for the pair average losses and the profile ratio.

13.9.8.1 Note about "Calculation variant"

When a contact analysis is performed on a planet system it takes into account all the meshings of the planets in the sun gear and the internal gear so that the load distribution coefficient (Ky) can be calculated accurately. However, in some cases the first step of a development project may involve calculating the load distribution along the gear's facewidth (KHβ, where an even distribution is required). The program provides a special input option for these situations. In the "Calculation variant" drop-down list the user can select Ky = 1 (i.e. the load is distributed evenly across all the planets), so that the load distribution across the facewidths can be calculated more quickly. The individual options are based on the following assumptions:

- "Load distribution on the facewidth including shaft misalignment, with Ky = 1 for all meshing" In this case, every planet has 1/Nth of the total transferred power and no iteration is performed for the system (fast calculation). However, the disadvantage of this is that the results are then not available at system level (transmission error on the carrier etc.). This option also ignores all the manufacturing errors.
- "Calculation of Kγ including shaft misalignment

This is the complete contact analysis calculation. Although it is slower than the procedure described above, it does calculate the load distribution between the individual planets and can also take manufacturing errors into account.

13.9.8.2 Define misalignment for individual elements

The following elements are assumed in a planet system:

Non-floating sun gear

- Planet carrier
- N planetary gears with the corresponding N pins
- Internal gear

The position of these elements in the gear unit and the appropriate misalignment can be defined in a dialog which you open by clicking on the "Axis alignment" button in the "Contact analysis" tab.

The alignment of the system elements is defined as follows in the "Axis alignment" tab:

- Alignment of the sun to the gear axis (see Figure 2)
- Alignment of the planet carrier to the gear axis (see Figure 3)
- Alignment of the planet pin to the planet carrier in circumferential direction dt and in offset direction dr (see Figure 4). To model a carrier deformation due to torsion, you must first define a value for dt.
- The planet gear is aligned relative to the planet pin. The positive misalignment (in circumferential direction dt and radial dr) is defined in accordance with the convention in Figure 4.
- Finally, the internal gear is also aligned relative to the gear axis (see Figure 2)

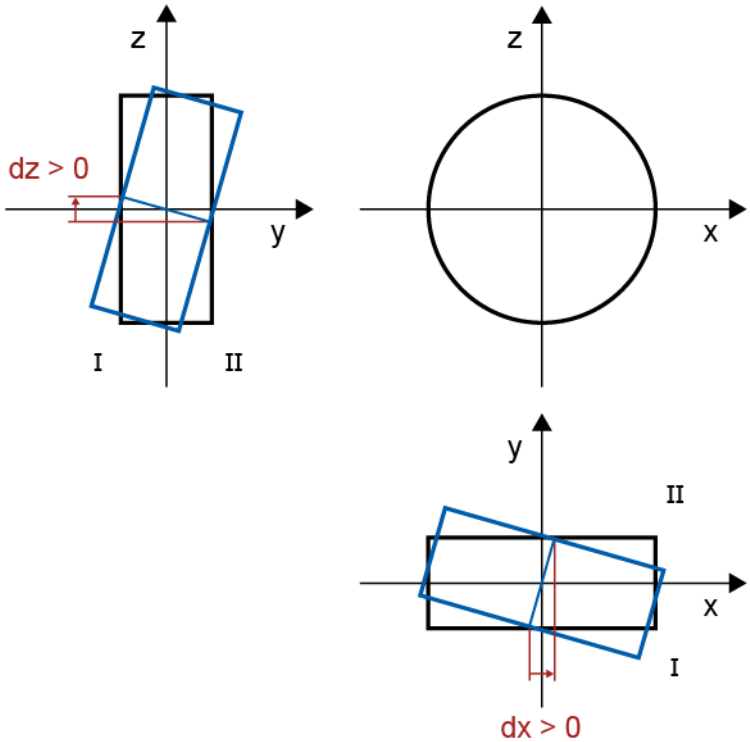


Figure 2: Misalignment of sun and internal gear to the gear axis

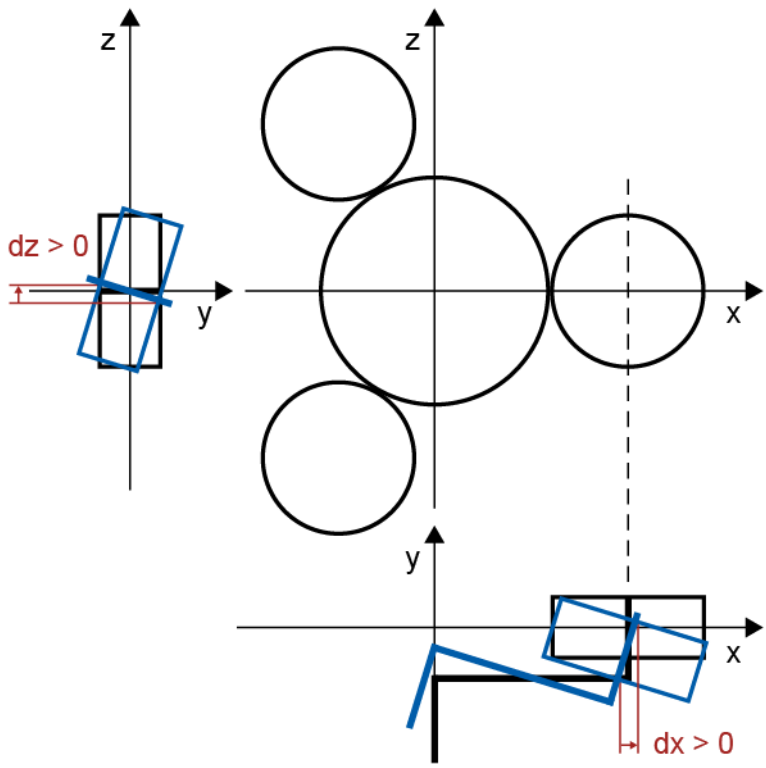


Figure 3: Misalignment of the planet carrier to the gear axis

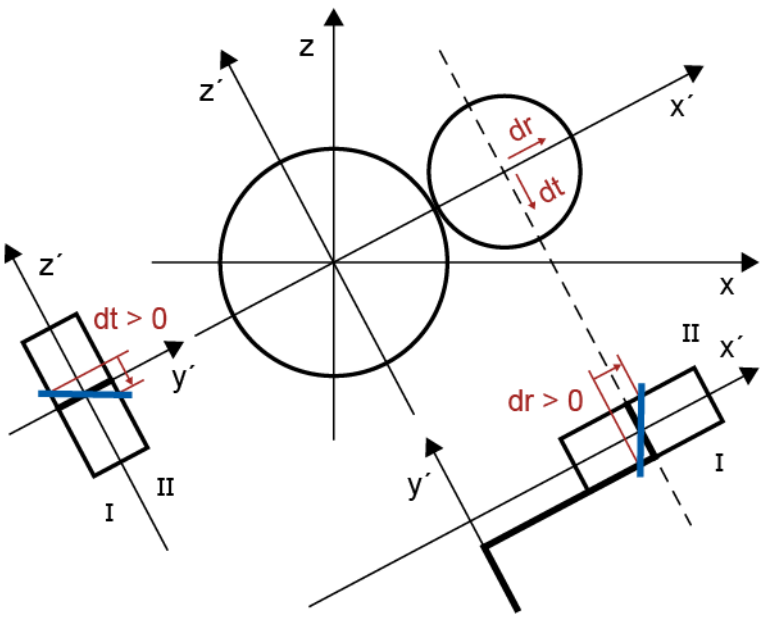


Figure 4: Misalignment of the offset direction to the planet carrier

You can also use shaft files to define the alignment of all the shafts, except the planet pin. The shaft files undergo the same checks as for a gear pair. For example, the value input for gear torque in the shaft calculation files must match the value entered for the gears in the calculation module. The beam shaft is characterized by its two couplings: one coupling transfers the torque to the sun gear and the other transfers the torque to the internal gear. The "effective diameter" for both couplings must be the same as the sun-planet-center distance. The "length of load application" must also be appropriate for the facewidth of the planet gear.

Angle Θ defines where the first planet gear must be located for a particular system definition. Every one of the subsequent planetary gears must have an angular offset of 2π / N to the previous gear.

In the "Torsion" tab, you define the side from which torque is introduced to the system or the side from which it is produced (depending on whether the element is a driving element or a driven element). You can select one of the following 3 options for inputting the direction of torque:

- Not taken into account

- Torque is applied/produced on side I
- Torque is applied/produced on side II

Each configuration is also displayed as a graphic so that the user can check their entries. In the following configuration, for example, the torque from the sun and the planet carrier is introduced on side I and the internal gear torque is introduced on side II.

If a shaft file is used to define the shaft alignment, the torque is calculated automatically from the results of the shaft calculation. However, the deformation is not calculated automatically from the direction of torque on the carrier because this value must be input directly in the "Axis alignment" tab. This value must therefore be checked to see if it is accurate.

13.10 Gear pump

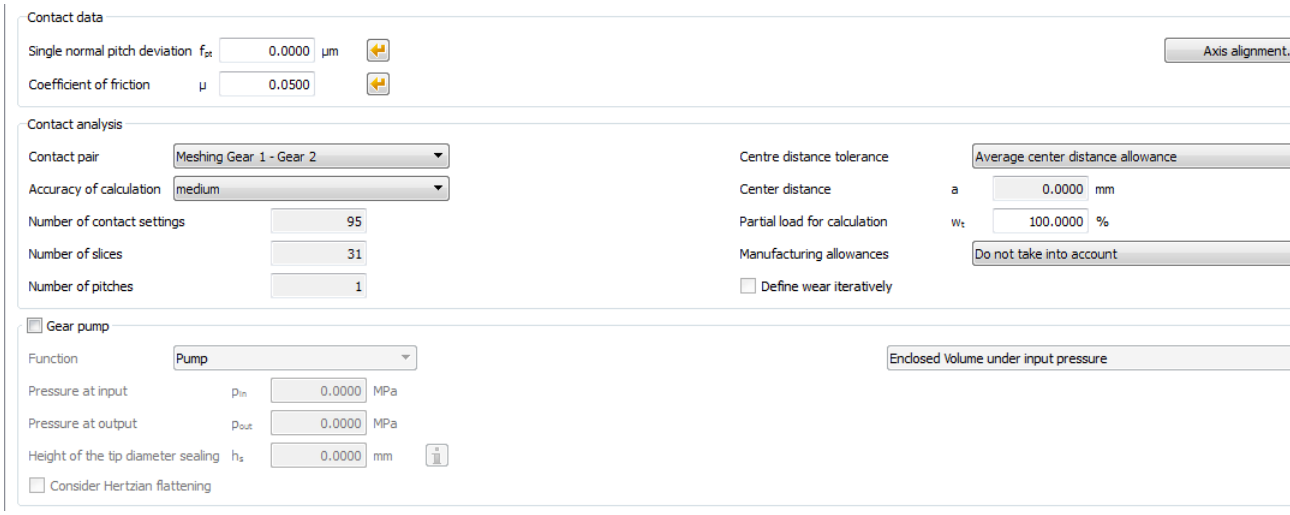


Figure 13.54: `Path of contact` input window for `Gear pump`

If you ignore the return volume, you can calculate the transport volume when you perform the normal calculation. You will find the parameter for this in the `Basic data input window`. (see section "`Basic data`" on page II-250) In this case, click the `Calculation of the displacement volume of gear wheel pumps` checkbox in the `Calculations` tab in the `Settings` window, which you open by clicking the `Calculation` menu.

In the lower part of the `Path of contact` input window you can then perform a detailed calculation for a gear pump.

The system calculates and displays the changes to the critical parameters of a pump that occur during meshing. These include geometric parameters such as the pinched volume (between two meshed tooth pairs, return volume), the volume with a critical inflow area (if possible, the flow of oil should be kept constant), the narrowest point (minimum distance between the first tooth pair without contact), inflow speed, oil inflow at the entry point (with Fourier analysis to evaluate the noise levels), volume under pressure at input. Other important information is the progression of torque on the two gears, the progression of the Hertzian pressure σ_H, the sliding velocity v_g and the wear coefficient $\sigma_H \cdot v_g$. Hertzian flattening can be included when calculating forces because this effect has a significant influence. The pinched volume depends on how the pump construction functions under pressure at input or output. This is defined by the appropriate input value and has a considerable effect on the torque curve. When the pinched volume is reduced, you see a significant momentary increase in compression in this volume. This produces strong pulsing forces on the support and therefore generates noise. A pressure

release groove must be installed to avoid this increase in pressure. For this reason, it is very useful to calculate and display the pressure flow in the pinched volume.

This calculation allows you to analyze any type of cylindrical gear with involute and non-involute teeth forms. At present, the only fundamental restriction is that this procedure is limited to spur gears.

Optimization strategies for gear pumps

The most important and critical problems regarding gear pumps are

- Noise
- Efficiency
- Size
- Wear

Here is some information that may help define the criteria according to which pumps can be evaluated.

- Noise:
  - Variations in flow through the pump generate noise in the pipes. For this reason, the flow (Q) should be as continuous as possible.
  - The enclosed volume (V1) should not be reduced during the generation process. A reduction in this volume would create a massive increase in compression in V1 and generate dynamic forces on both the bearing and the shafts. This effect can be reduced by the precise sizing of relief grooves.
  - The inlet speed of the oil through the narrowest point should be kept as low as possible
- Efficiency:
  - Return volume should be kept as low as possible
- Size:
  - The KISSsoft Fine Sizing functions provide a very efficient method of achieving the highest possible displacement volume for a specified size.
- Wear:
  - You must monitor the course of the wear values (sliding velocity and Hertzian pressure between the tooth flanks)

NOTE:

You will find more detailed information about gear pump analyses in KISSsoft-anl-035-E-GearPumpInstructions.doc [77] (available on request).

- The "Gear pump" report shows the input torque on gear 1 [T1] and the torque transferred from gear 1 to gear 2 [T1Contact].
- You should use the torque at the point of contact in the strength calculation and the contact analysis (calculated from P_{out} and P_{in}). Enter this data in the "Basic data" tab.
- You should use the torque at the point of contact in the strength calculation and the contact analysis (calculated from P_{out} and P_{in}). Enter this data in the "Basic data" tab. The total power [P] and the torque [T1] at the pump inlet are only documented in the "Gear pump" report and are not otherwise used. All the graphics shown under "Graphics" "> Gear pump" are based on compression. The torque curve used in the graphic is the input torque [T1].

13.11 Operating backlash

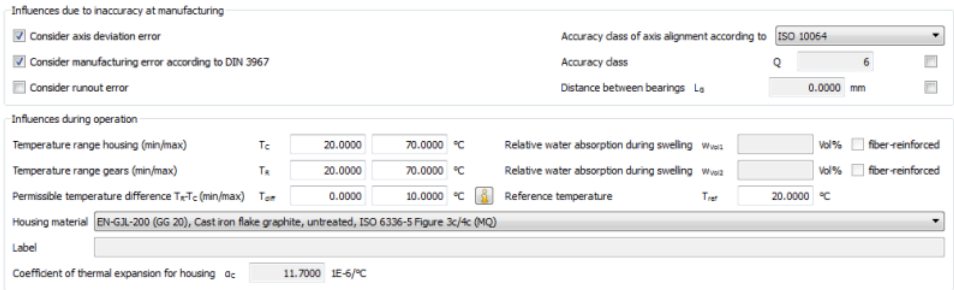


Figure 13.55: Operating backlash input window

In addition to calculating the theoretical backlash, the backlash after mounting can also be calculated as defined in DIN 3967 (this includes toothing deviations, deviation error of axis in accordance with ISO 10064 or DIN 3964 (see also Table 13.15). The operating backlash (including the temperature differences between the gears and the housing) is also calculated. To calculate them, the required input is a temperature range for the gears and the housing, and the maximum and minimum difference in temperature between them. Two cases are calculated simultaneously, one that produces the maximum operating backlash (with the given temperature inputs), and one that produces the minimum operating backlash.

If the module is < 1, the statistically evaluated circumferential backlash is also calculated in accordance with DIN 58405.

The reduction of the backlash due to individual teeth deviations is then calculated with tolerances Fb, Ff and fp in accordance with DIN 3961. These values as specified in DIN 3961 are not defined for module < 1. In this case, tolerances for module 1 are defined in accordance with DIN 3961 and then reduced in proportion to the module. According to formula: fp(mn) = fp(mn=1.0) * mn.

The reduction in clearance due to deviations in individual teeth is not taken into account for worm gears.

The effect of the runout error can also be taken into consideration. In this case the roller runout tolerance (determined using the approximation formula Fr = Fi" - fi") is used instead of the runout error Fr for module < 1.

Bearing center distance L_G (nominal length)	**Axis alignment accuracy class**											
	1	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**	**12**

in mm												
to 50	5	6	8	10	12	16	20	25	32	40	50	63
over 50 up to 125	6	8	10	112	16	20	25	32	40	50	63	80
over 125 up to 280	8	10	12	16	20	25	32	40	50	63	80	100
over 280 up to 560	10	12	16	20	25	32	40	50	63	80	100	125
over 560 up to 1000	12	16	20	25	32	40	50	63	80	100	125	160
over 1000 up to 1600	16	20	25	32	40	50	63	80	100	125	160	200
over 1600 up to 2500	20	25	32	40	50	63	80	100	125	160	200	250
over 2500 up to 3150	25	32	40	50	63	80	100	125	160	200	250	320

Table 13.15: Deviation error of axis in accordance with DIN 3964, values in [mm]

As shown in Table 13.15, the values in the Axis alignment accuracy and Distance between bearings input fields are used to calculate the axis deviation error in accordance with DIN 3964.

Backlashes are calculated as specified in DIN 3967.

Circumferential backlash calculation:

The circumferential backlash is calculated in accordance with DIN 3967 with the following formula on the reference circle:

$$j_t = (-A_s / \cos\beta) + 2 \cdot A_a \cdot \tan\alpha_t$$

In KISSsoft, the operating backslash is calculated using the more precise formula in the operating pitch circle:

$$j_t = (-A_s / \cos\beta \cdot \frac{\cos\alpha_t}{\cos\alpha_{wt}}) + 2 \cdot A_a \cdot \tan\alpha_{wt}$$

Planetary gear units are another special feature of the operating backslash calculation. Here, there are 2 operating pitch diameters for the planets (sun/planet and planet/internal gear). The change in operating pitch diameter due to thermal expansion is defined here for the operating pitch circle determined in this process.

In addition, the change in tip clearance due to thermal expansion (and water absorption for plastics) is also calculated.

Any elongations that occur in the body of the gear also change its pitch. A single pitch deviation occurs as soon as both gears show unequal strain. The increase or decrease in pitch caused by thermal expansion is defined as follows:

$$\Delta pt_1 = pt_1 \cdot \alpha_1 \cdot \left(\Theta_1 - \Theta_{ref}\right)$$
$$\Delta pt_2 = pt_2 \cdot \alpha_2 \cdot \left(\Theta_2 - \Theta_{ref}\right)$$
$$fpt = \Delta pt_1 - \Delta pt_2$$

pt	pitch
a	coefficient of thermal expansion
Q	temperatures
fpt	single pitch deviation

Plastics also undergo expansion due to water absorption.

13.11.1 Reference temperature

The `Reference temperature`T is the ambient temperature specified for manufacturing. The tooth thickness input here apply to this temperature.

The `Mass temperature` for the individual gears defines the thermal expansion of the individual gears, The Gear mass temperature of the scuffing calculation can be used as here as a starting point.

Taken together with the coefficient of thermal expansion, the `Temperature of housing` then defines the amount of thermal expansion that occurs for the housing.

13.11.2 Relative water absorption during swelling

You must input this value as a [%] of the volume. To calculate clearance, DIN 3967 specifies that: For plastics, the linear expansion due to water absorption detailed in DIN 3967 is approximately 1/3 of the amount of water absorbed. However, for fiber-reinforced plastics it is only around 1/12 of the volume of water absorbed. If you click this checkbox, this phenomenon is taken into consideration when calculating the change in volume.

13.11.3 Coefficient of thermal expansion for housing

If you select a material from the database, this field merely provides information about the coefficient of expansion of the selected housing material. In this case, you cannot change the value. However, if you have set the `Housing material` drop-down list to `Own Input`, you can enter your own value.

13.12 Master gear

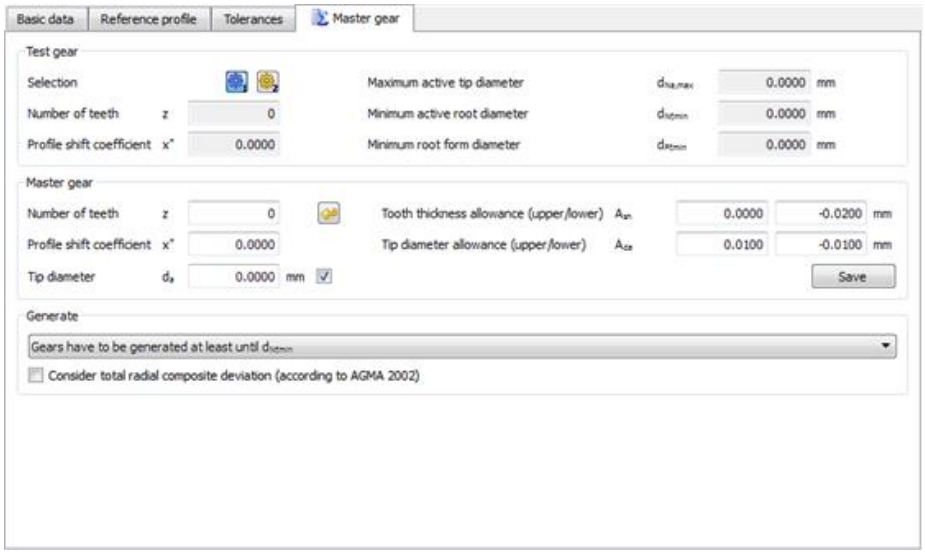


Figure 13.56: `Master gear` input window

This KISSsoft calculation module has been designed to enable you to size and check master gears.

To perform a test for double flank composite transmission, you require one master gear which is then rotated on a test device together with the gear you want to test. In the test run, the test gear and the master gear are pressed lightly together so that no backlash is generated. The deviation in center distances is then measured carefully. The difference between the minimum and maximum value calculated here is the tooth-to-tooth composite error. In order to achieve an accurate statement about the how the test gear behaves when running after it has been installed in the gear, the active involute of the test gear should be processed as completely as possible in the test run. However, it is essential that you prevent the master gear from meshing too deeply in the root area: If the value for the root form diameter of the test gear is not achieved, this will cause meshing interference which will in turn generate measurement results that are massively incorrect. You can call the master gear sizing function for each gear in a particular calculation. When you open the sizing window, the default values for a suitable standard master gear taken from DIN 3970 are displayed. The analysis functions check the maximum and minimum tolerance fields of the tooth thickness of the test gear whose involute is being processed. The report then show which area of the active involute has been tested, or not tested. If the value for the root form diameter is not achieved, the program issues a warning to prompt you to reduce the tip circle diameter of the master gear. This calculation is also available for cylindrical gears with a minimum number of

teeth greater than 4. Click the `Save` button to save the master gear data and the master gear-test gear pair as KISSsoft files.

Take into account total radial composite deviation (in accordance with AGMA 2002): When calculating the smallest test center distance [aMin], the theoretical center distance stated in AGMA2002 (equation 8.5) is further reduced by the total radial composite deviation (Vcq specified in AGMA 2000). If the manufacturing tolerances specified in ISO or DIN are being applied, Fi" is used for that purpose. If the tolerances specified in AGMA are applied, Vcq is used here:

13.13 AGMA 925

As specified in AGMA925, you can use this input window to define the probability of scuffing and wear as well as susceptibility to micropitting

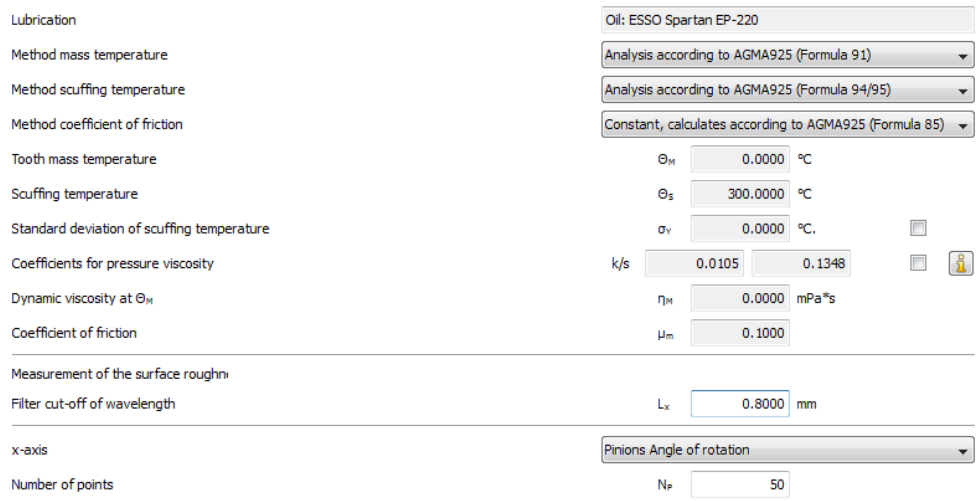


Figure 13.57: AGMA 925 input window

AGMA 925-A03 *Effect of Lubrication on Gear Surface Distress* calculates the conditions in the lubrication gap across the gear meshing. AGMA925 defines how to calculate the lubrication gap height whilst taking into account the flank deformation, lubricant properties, sliding velocity and the local Hertzian stress. The standard then uses this base data to calculate the probability of wear. The wear is caused by the metal surfaces contacting each other if the lubrication gap is too narrow. The probability of wear calculated by the standard is greater than the values that occur in practice.

The standard does not give any indications about safety against micropitting. However, data provided by the relevant technical literature and the results of research reveal that there is a direct correlation between the minimum lubrication gap-to-surface roughness ratio and the occurrence of micropitting. You can therefore use this calculation method to optimize gear toothing for micropitting. AGMA 925 also includes a definition of the probability of scuffing. This analysis uses the same base data (Blok's equations) as the calculation of scuffing according to the flash temperature criteria given in DIN3990, Part 4. However, defining the permitted scuffing temperature in accordance with AGMA925 presents more of a problem because of the lack of comprehensive or generally applicable information. In particular, there is no reference to a scuffing load capacity specification as given

in the FZG test. There is therefore a tendency to under evaluate oils that have effective EP additives.

Values for the compression viscosity coefficient α of typical gear oils vary between 0.00725mm^2/N and 0.029mm^2/N and are defined as follows in AGMA 925-A03:

$$\alpha = k \cdot \eta_M^5 \tag{13.25}$$

where

α	Compression viscosity coefficient	mm^2/N
k	see Table 2 in AGMA 925-A03	-
η_M	Dynamic viscosity for tooth temperature θ_M	mPa . s

In practice, calculating wear in accordance with Wellauer results in risk of wear values that are too high. For this reason, the analysis is performed as stated by Dowson (as in Annex E of AGMA 925). The report shows the results for both methods.

13.14 Rough sizing

Rough sizing provides suggestions for possible toothing configurations based on the data entered for the ratio and load. To use this function, go to the `Calculation` menu and select `Rough sizing` or click the corresponding icon in the tool bar.

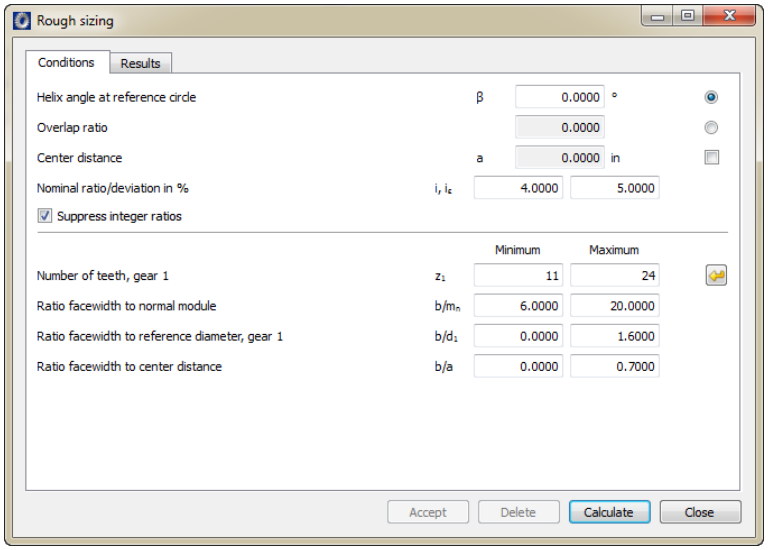


Figure 13.58: `Rough sizing` dialog window

At present you can apply this to inside and outside toothed cylindrical gear pairs and planetary gears. The target ratio is the most important input parameter. For an internal gear pair, the ratio must be entered as a negative value in the `Geometry` area. In planetary stages, the nominal ratio must be > 2.0.

The operating data (power, speed, etc.) is fetched from the KISSsoft main window (and can be changed there if required). You can also specify a helix angle or a required overlap ratio (e.g. ε_β= 1.0.).

Click on the `Calculate` button to open a list of suggestions that you can use to set the parameters for your gears. Click the right-hand mouse button on any entry in this list to open a context menu with a list of possible sizes (see Figure 13.59).

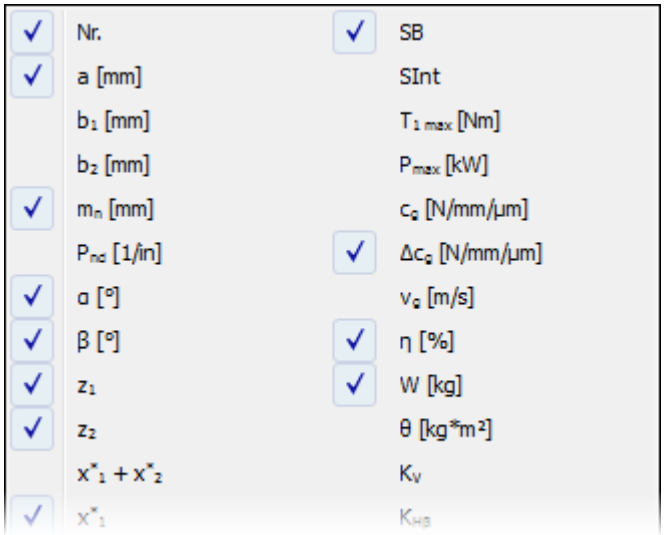


Figure 13.59: Section of possible parameters in the Context menu

The sizes that have a tick in their checkbox are displayed in a list. The other sizes are not displayed. Click a value to set/delete a cross. You will find a legend describing the parameters used here at the end this section.

Rough sizing automatically defines the most important tooth parameters (center distance, module, number of teeth, width) from the power that is to be used and the required ratio together with strength calculation in accordance with the selected calculation standard. Dimensioning is performed in accordance with minimum safeties (Required safeties (see page II-441)).

You can specify the intervals for the relationships b/m_n-, b/a-, b/d in the `Calculation` menu under `Settings > Sizings`. (Sizings (see page II-434))

The program displays a number of different solutions which you can use. You can then perform fine optimizing together with fine sizing. The window remains open, to allow you to use more solutions. You will find more detailed information about fine sizing in section 13.15.

The most important result of this sizing process is that it enables you to define the achievable center distance ranges and module ranges, as well as the facewidth. You can then decide how much space is required for the machinery itself.

You can predefine the center distance for special cases. However, in these cases, you must remember that the program's sizing options are not comprehensive, and fine sizing represents a better alternative.

Sizing of strength for a planetary gear

When performing rough sizing for planetary stages, it is assumed that the rim is static. If the rim rotates, you must change the speeds after sizing.

Different constraints for rough sizing

- **The system prompt suggests the number of teeth as defined by Niemann** Table of standard number of pinion teeth in accordance with Niemann [65], Table 22.1/8.

Ratio *u*	**1**	**2**	**4**	**8**
Through hardened or hardened				
counter-through hardened to 230 HB	32..60	29..55	25..50	22..45
over 300 HB	30..50	27..45	23..40	20..35
Cast iron	26..45	23..40	21..35	18..30
nitrided	24..40	21..35	19..31	16..26
case-hardened	21..32	19..29	16..25	14..22

  Click the Sizing button to transfer these values from the program automatically.

- Module ratio b/m_n, reference diameter ratio b/d_1, center distance ratio b/a (see page II-434)

Parameter	**Meaning**
No.	Sequential numbering
a	Center distance
$b_{1(2)}$	Facewidth
m_n	Normal module
P_{nd}	Normal diametral pitch
α	Pressure angle
β	Helix angle

$z_{1(2)}$	Number of teeth
$x^*_1 + x^*_2$	Total profile shift coefficients
$x^*_{1(2)}$	Profile shift coefficient
$h^*_{aP1(2)}$	Addendum coefficient
$h^*_{af1(2)}$	Dedendum coefficient
Cutter/Tool	ID number of the hobbing cutter[1)]
Reference profile gear 1(2)	Reference profile database ID
$d_{a1(2)}$	Tip diameter
$d_{f1(2)}$	Root diameter
ε_α	Transverse contact ratio
ε_β	Overlap ratio
ε_γ	Total contact ratio
$\zeta_{max(min)}$	Specific sliding
AC/AE	Ratio of contact length[2)]
i	Transmission ratio
i_e[%]	Deviation from nominal ratio
Hunting	z_1 and z_2 have - apart from 1 - no common parts
$d_{w1(2)}$	Operating pitch diameter
α_{wt}	Working transverse pressure angle
α_{wn}	Normal pressure angle
β_w	Helix angle at reference circle
b/d_1	Facewidth to reference diameter ratio
b/m_n	Facewidth to normal module ratio
b/a	Facewidth to center distance ratio
$SF_{1(2)}$	Root safety
SF_{min}	Minimum root safety
$SH_{1(2)}$	Flank safety
SH_{min}	Minimum flank safety
SB	Safety against scuffing for flash temperature
SInt	Safety against scuffing for integral temperature

T_{max}	Maximum torque
P_{max}	Maximum power
C_g	Tooth contact stiffness
Δc_g	Change in tooth contact stiffness
ν_g	Sliding velocity
η	Power loss
W	Total gear weight
Θ	Moment of inertia
K_v	Dynamic factor
$K_H\beta$	Face load factor
Summary	see Fine Sizing Results (see page II-417)
$H_{min,\ bending}$	Minimum service life, only include root
$H_{min,\ flank}$	Minimum service life, only include tooth flank
H_{min}	Minimum service life
$V5$	Displacement volume: as gear pump Note: To enable this calculation, set the flag for calculating the displacement volume under `Calculation > Settings >Calculations`.

1) according to the `List of cutters for reference profile` drop down list. Only for `Fine Sizing` (see page II-412).

2) Results (see page II-417), Point 5

13.15 Fine Sizing

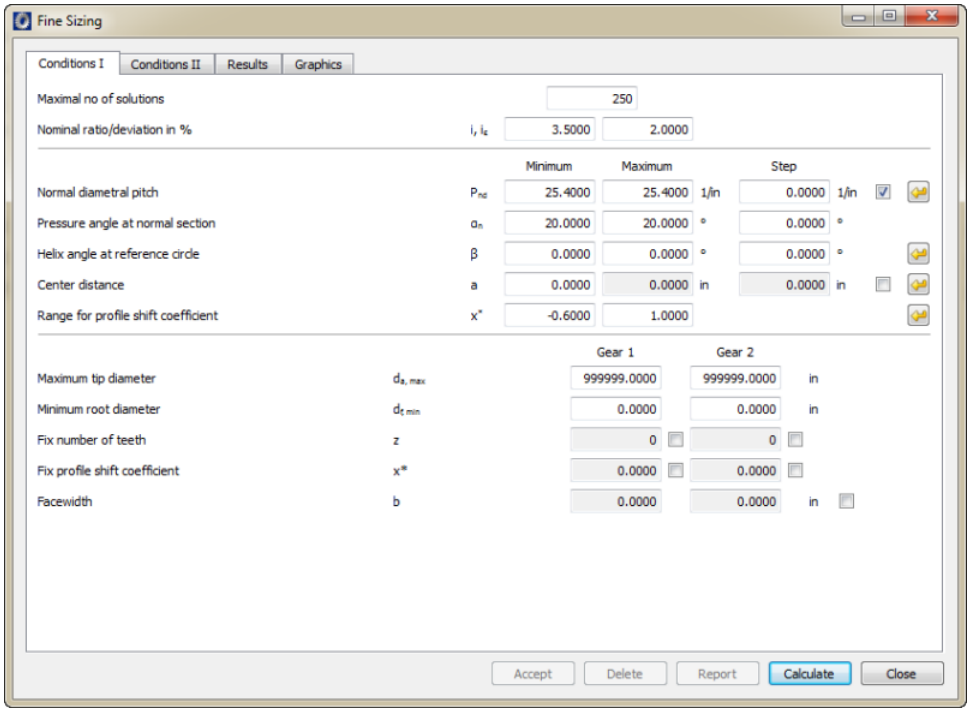


Figure 13.60: `Conditions I` tab in the `Fine Sizing` window

To start the `Fine Sizing` process, click the `Calculation` menu and select the `Fine Sizing` option or click the icon in the tool bar.

If you input a nominal ratio, a center distance and intervals for the module and helix angle as well as the pressure angle, the system calculates and displays all the possible suggestions for the number of teeth, module, helix angle and profile shift. It also shows the deviation from the nominal ratio, the specific sliding and the contact ratio. This module can also be used to size planetary stages and cylindrical gear stages with change gears.

All the variants found by this process can be evaluated by a wide range of different criteria (accuracy of ratio, weight, strength, tooth contact stiffness deviation etc.)

Depending on your requirements, limits can also be set on the most important parameters (tip circle, root circle, minimum number of teeth, tolerated undercut etc.). In addition to creating text reports detailing the solutions and the summary, the summary can also be displayed as a graphic.

For planetary gear units or cylindrical gears that have an idler gear, you can: perform the calculation either with the predefined center distance or with a predefined internal gear V circle d+2*x*mn (normal case).

In the case of cylindrical gear pairs, the center distance can either be fixed (normal case) or predefined in an interval. To do this, click the checkbox to the right of the `Center distance` input field.

The facewidth appears in the input screen, where you can modify it if required.

NOTE

You should check the center distance interval after you change the reference circle or select a variable center distance. You may then need to repeat the sizing process.

13.15.1 Necessary entries in the input window

Before you start the fine sizing process, you must enter the following data correctly in the `Basic data` or `Geometry` and `Strength` standard tabs to ensure the calculation returns the results you require.

Geometry:

- Reference profile
- Number of idler gears/planets (in a 3 gear configuration)

Strength:

- Materials
- Power/Speed
- Application factor
- Service life
- Lubrication

13.15.2 Conditions I

13.15.2.1 Limiting the tip diameter

Solutions whose tip circle exceeds the specified value are rejected. If you do not want to limit the tip value, you can input either 0 or 10^{10}.

However, the following problem prevents this option being used sensibly in practice: If a gear is to be installed in an existing housing, it is critical that it does not touch the walls of the housing.

13.15.2.2 Limiting the root diameter

Solutions whose root circle falls below the specified value are rejected. If you do not want to limit the root diameter, you can input 0.

However, the following problem prevents this option being used sensibly in practice: If a gear is mounted on roller bearings in a speed change gear unit, you must guarantee a minimum thickness of material between the bore and the root circle.

13.15.2.3 Maximal no of solutions

Proposal: 50 to 250

If the program finds more than the specified number of solutions, you see a warning message and an appropriate note is entered in the report.

NOTE

You should only perform a final evaluation after all the possible solutions have been displayed. Otherwise you run the risk of missing the optimum solution.

13.15.2.4 Limiting the number of teeth

You should not normally need to use this option, which is why the default setting is for it to be disabled. However, by clicking the individual checkboxes, you can still fix this parameter. A useful application for this option is when for sizing a planetary gear which has already been modified to fit inside a predefined internal gear. In this case, the module and the number of teeth for gear 3 are predefined.

13.15.3 Conditions II

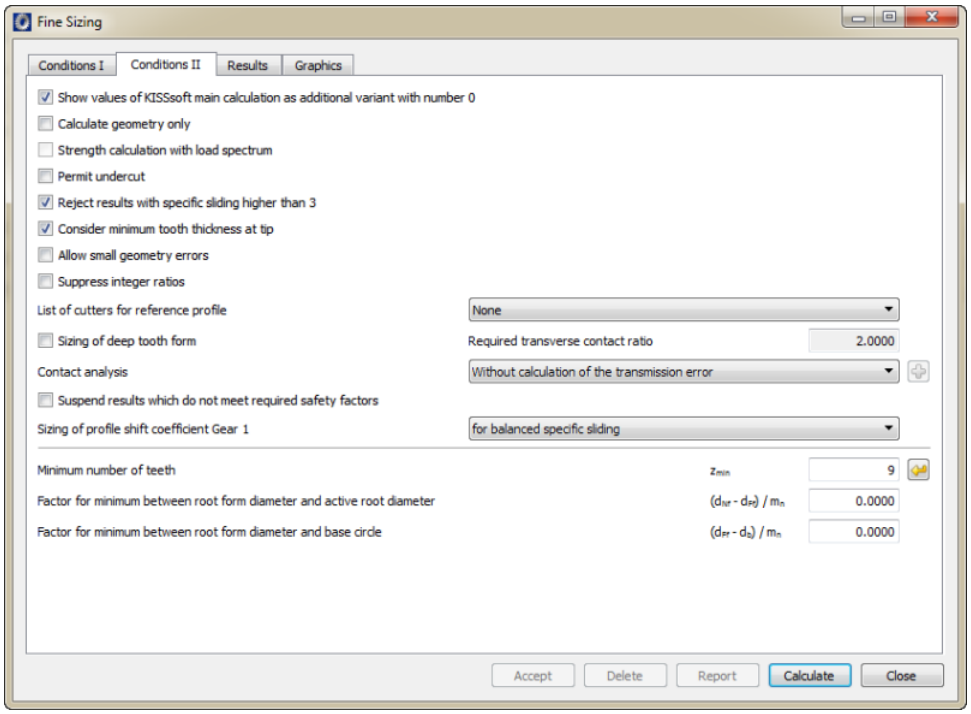


Figure 13.61: `Conditions II` tab in the `Fine Sizing` window

In the `Conditions II` tab you can specify other essential functions.

1. **Show values of KISSsoft Basic Tab as additional variant with number 0**
   The toothing data in the KISSsoft Basic tab can also be displayed as a variant with the number 0 (table and graphic). However, **the data at the start of the fine sizing process must be consistent before this can happen**. This option can either be enabled or disabled. When you enable this option, you must restart the fine sizing process so that the variant can also be displayed.

2. **Calculate geometry only**
   If you select this method, no strength calculation is performed.

3. **Strength calculation with load spectrum**

   Before you can perform calculations with a load spectrum you must specify a load spectrum in the KISSsoft main window before you start the fine sizing process and run the calculation (to ensure the data is consistent). In this case, when you start the fine sizing process, you are prompted to confirm that you want to perform the calculation with a load spectrum. The

flag in the window merely shows whether or not the calculation includes a load spectrum. You cannot reset this flag.

4. **Permit undercut**
   If this option is selected, solutions with undercut are not rejected.

5. **Reject results with specific sliding higher than 3**
   Usually specific sliding should not be greater than 3.

6. **Consider minimum tooth thickness**
   If this option is selected, solutions with tip tooth thickness that is less than the predefined minimum tooth thickness (see `Calculation > Settings > General`) will be rejected.

7. **Allow small geometry errors**
   Now minor meshing interference and similar geometry errors will be tolerated when the system is calculating variants! You can make separate settings to take into account the undercut and the minimum tooth thickness at the tip (see points 2 and 4).

   You must set this option if the program has to find solutions where the number of teeth is less than 7, or in other exceptional situations. We do not recommend you set this option in any other situation!

   Note:
   In these cases you must also change the minimum number of teeth (see point 11) accordingly.

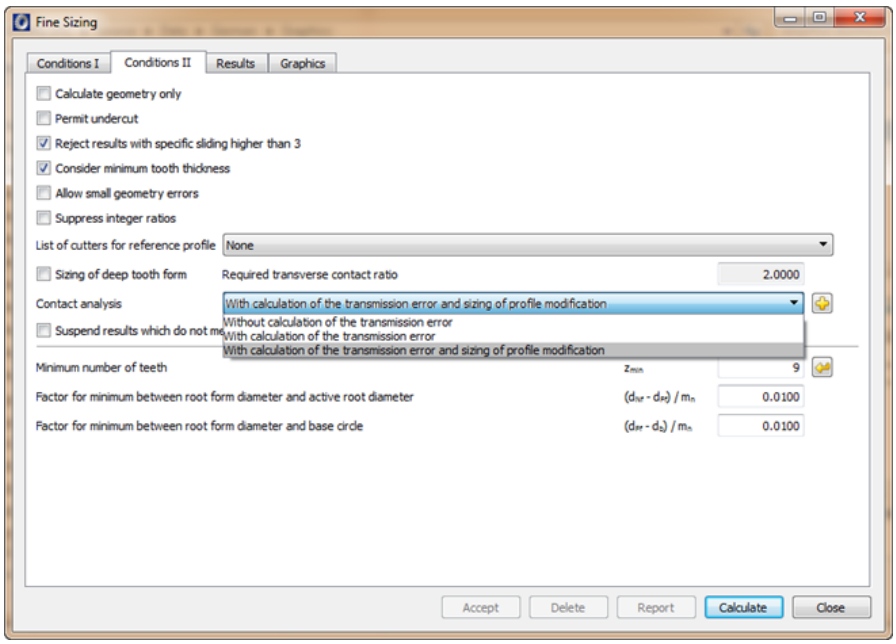
8. **Suppress integer ratios**
   If this option is selected, results with whole number gear ratios will be rejected.

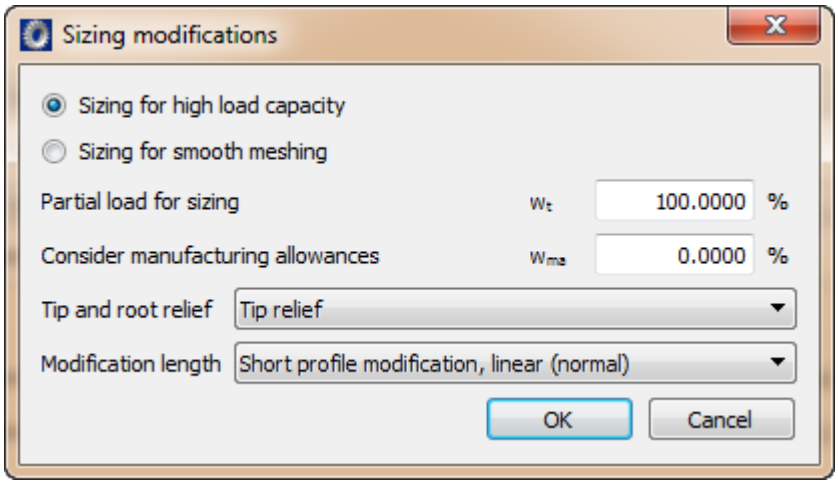
9. **List of cutters for reference profile**
   Instead of using the predefined reference profile, you can use a list of hobbing cutters for fine sizing. In this case, the calculation is performed for every cutter in the given module and pressure angle range and the tool is displayed in the results list.
   The same hobbing cutter is used for each gear. Internal teeth are not affected by this setting.

10. **Sizing of deep tooth forms**
    Special reference profiles with larger addendums and dedendums are used for deep toothing. This sizing function calculates the necessary transverse contact ratio on the basis of the required reference profile. If this function is active in fine sizing, the reference profile for every solution is calculated so that precisely the specified required transverse contact ratio is achieved.

As a result, only those solutions that have at least the required transverse contact ratio are displayed.

11. **Transmission error**
If the "Calculation of transmission error" option is selected, the contact analysis is performed for every variant. If the "Calculation of transmission error and profile modification" option is selected, the length and value of the profile modification is automatically determined according to the correction method settings. Click the button to open the profile modification settings window.





The correction method includes the objective (for high load capacity or

smooth meshing), tip and/or root relief, length (short or long), and the types (linear, arc, progressive, and linear with transition radius). It is important to note that the transmission error can be minimized only for one load, and the partial load for sizing should be set correctly according to the applied load level.

During the contact analysis for transmission error, the default settings are used to prevent the extraordinary behavior of the calculation except for the coefficient of friction and accuracy of calculation. Input the required values in the main program, in the "Contact analysis" tab. You can also specify the accuracy of the calculation, however, we strongly recommend you use "low" or "average" to reduce the processing time. Therefore, the transmission error in fine sizing may not be exactly the same as you get in the contact analysis, according to the settings.
The default settings are:

Calculation for: right flank
- Torque for gear A: not considered
- Torque for gear B: not considered
- Partial load for calculation: 100 %
- Center distance: Average center distance allowance
- Single pitch deviation: 0 mm
- Deviation error of axis: 0 mm
- Inclination error of axis: 0 mm

The results list shows:
- Transmission error (PPTE)
- Medium wear on the tooth flank (delwn1, delwn2
- Maximum flash temperature (theflamax)
- Variation in bearing force (VarL)

The calculation time increases significantly if the transmission error calculation option is used. We therefore recommend you limit the number of results before starting the calculation.

12. **Suspend results which do not meet required safety factors**
Variants which do not meet the predefined minimum safety levels (see `Calculation > Settings > Required safeties`) will be rejected.
Note:
Variants with insufficient safety against scuffing will not be rejected.

13. **Maximum x1 (x1max)**
    You can also derive the largest profile shift used (x1) from balanced specific sliding or from the minimum topland.

14. **Minimum number of teeth** z_{min}
    Practical values range for the minimum number of teeth:

    For helical gear teeth: 7 to 9
    For spur gear teeth: 10 to 12

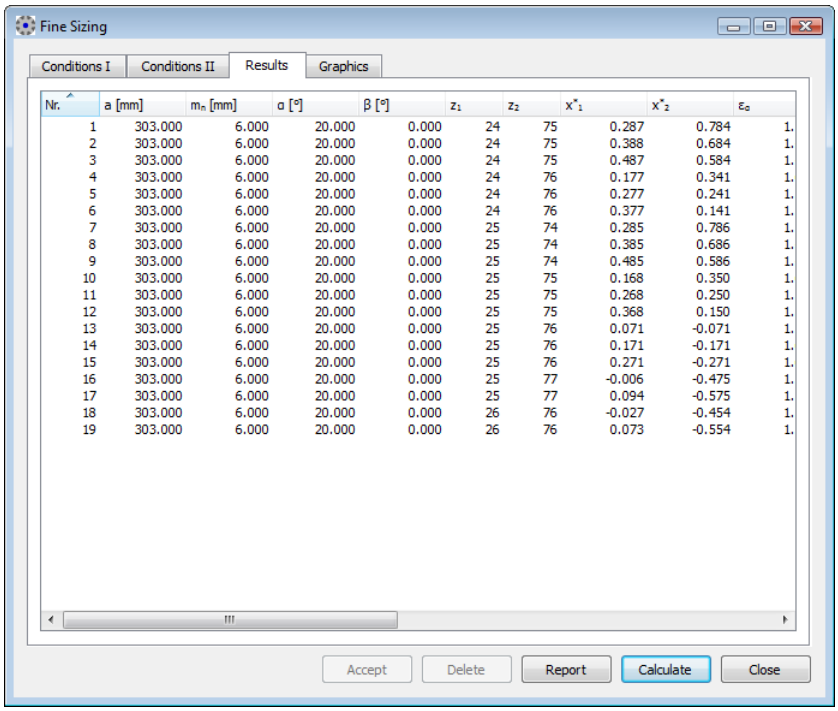
    Click the button to display a suggested value for the minimum number of teeth.

    Note:
    If you want to find solutions where the number of teeth is less than 7, you must first select the `Allow small geometry errors` option.

15. **Minimum distance between root form diameter and active root diameter** d_{Nf} - d_{Ff}
    Meshing interference occurs if the active root diameter is less than the root form diameter. Here you can specify a minimum value for the distance between the active root diameter and the root form diameter, i.e. between active and manufactured involutes. The input value is the minimum difference between the two diameters.

16. **Minimum between root form diameter and base circle** d_{Ff} - d_b
    If the start of the manufactured involute is closer to the base circle this will cause greater wear on a tool during the manufacturing process. Here you can specify a minimum value for the distance between the root form diameter and the base circle. The input value is the minimum difference between the two diameters.

13.15.4 Results



Nr.	a [mm]	m_n [mm]	α [°]	β [°]	z_1	z_2	x^*_1	x^*_2	ε_α
1	303.000	6.000	20.000	0.000	24	75	0.287	0.784	1.
2	303.000	6.000	20.000	0.000	24	75	0.388	0.684	1.
3	303.000	6.000	20.000	0.000	24	75	0.487	0.584	1.
4	303.000	6.000	20.000	0.000	24	76	0.177	0.341	1.
5	303.000	6.000	20.000	0.000	24	76	0.277	0.241	1.
6	303.000	6.000	20.000	0.000	24	76	0.377	0.141	1.
7	303.000	6.000	20.000	0.000	25	74	0.285	0.786	1.
8	303.000	6.000	20.000	0.000	25	74	0.385	0.686	1.
9	303.000	6.000	20.000	0.000	25	74	0.485	0.586	1.
10	303.000	6.000	20.000	0.000	25	75	0.168	0.350	1.
11	303.000	6.000	20.000	0.000	25	75	0.268	0.250	1.
12	303.000	6.000	20.000	0.000	25	75	0.368	0.150	1.
13	303.000	6.000	20.000	0.000	25	76	0.071	-0.071	1.
14	303.000	6.000	20.000	0.000	25	76	0.171	-0.171	1.
15	303.000	6.000	20.000	0.000	25	76	0.271	-0.271	1.
16	303.000	6.000	20.000	0.000	25	77	-0.006	-0.475	1.
17	303.000	6.000	20.000	0.000	25	77	0.094	-0.575	1.
18	303.000	6.000	20.000	0.000	26	76	-0.027	-0.454	1.
19	303.000	6.000	20.000	0.000	26	76	0.073	-0.554	1.

Figure 13.62: `Results` tab in the `Fine Sizing` window

Click the `Report` button to open the editor and display a list of the best results. A brief description of the criteria used to evaluate the best variants is given here. Please note that these criteria are not relevant to every case, and only need to be queried in particular applications!

1. Summarize variants for accuracy of gear ratio:
   The difference between the actual gear ratio and the required gear ratio is evaluated here.
2. Weight: this is an indicator for the manufacturing price
3. Specific sliding: maximum value
4. Sliding velocity: maximum value
5. Relationship AC/AE
   AC: length of path of contact from meshing point to pitch point
   AE: total length of path of contact
   "Pushing" sliding occurs in the AC area of contact (sliding velocity of the driving gear is greater than that of the driven gear). As this area is critical for unlubricated plastic gears, the AC/AE ratio should be as small as possible in this case.

6. Summarize variants for vibrations:
   The variation in the total contact stiffness is evaluated here (the lower the variation the better).
   The calculation is based on empirical formulae, unless the "Calculate mesh stiffness" option is set in "Conditions II.
7. Summarize variants for strength:
   This summarizes root and flank safety with reference to the required safety. Although safeties of less than the required safety are given a very negative evaluation, large safety margins above the required safety have very little influence.
8. Transmission error (PPTE)
   Transmission error is displayed if the corresponding option is set in "Conditions II".
9. `Summary` evaluation:
   The `Summary` evaluation weights each component to form a total evaluation coefficient. Set the weighting of individual components in `Calculation > Settings > Summary`. This weighting depends to a great extent on which solution you require, for example, whether you want a solution that is optimized for noise reduction or strength.

NOTE

The Rough sizing (on page II-404) section includes a complete list of all the available parameters. You will find information about noise optimization in [56].

13.15.5 Graphics

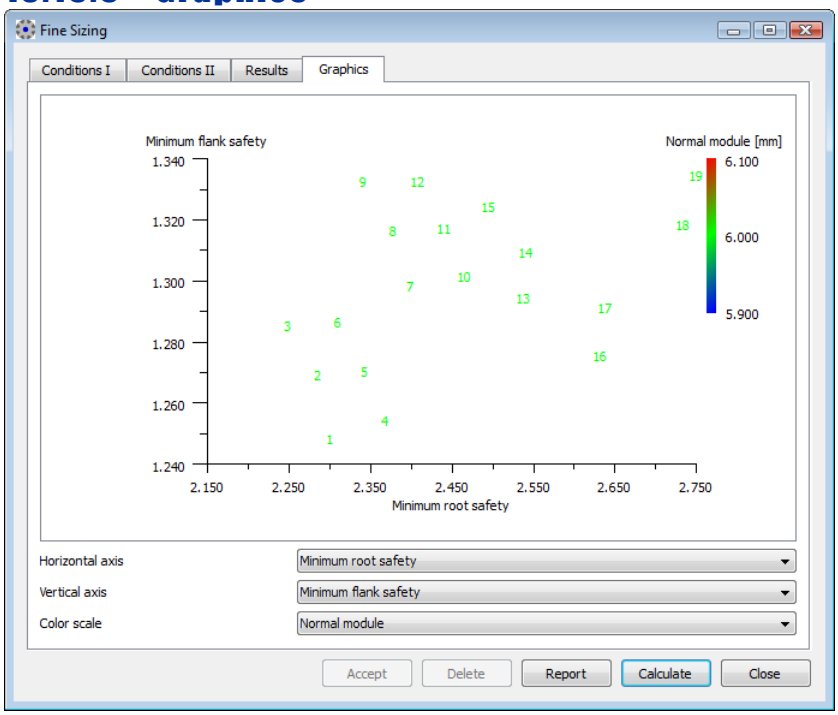
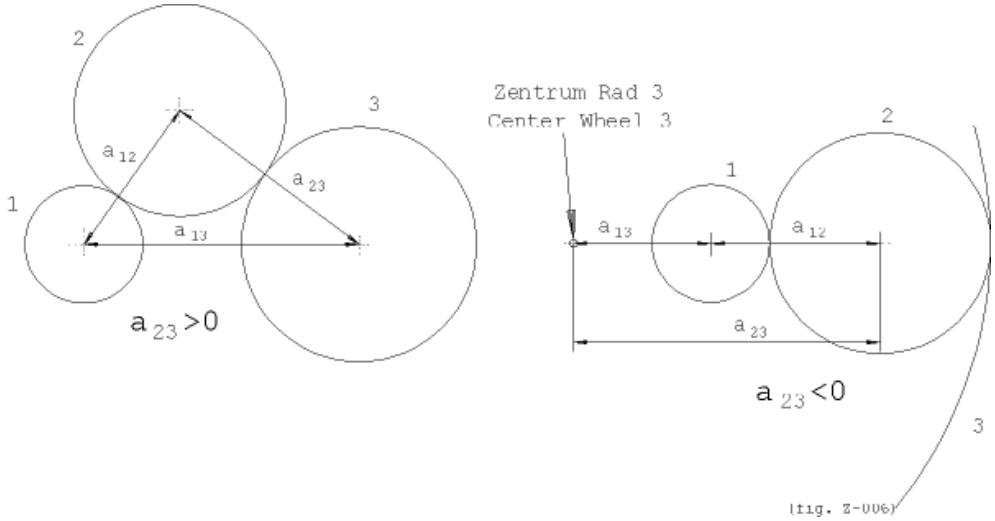


Figure 13.63: `Graphics` tab in the `Fine Sizing` window

The Graphics tab in the Fine Sizing window gives you a quick overview of the number of solutions. It can display three parameters simultaneously. You can change them in the selection lists. In addition to the two axes, the third parameter is displayed as a color.

13.15.6 Geometry-Fine Sizing for 3 gears

Definition of center distances:



13.15.7 Additional strength calculation of all variants

The KISSsoft system also calculates the strength (tooth root, flank and scuffing) of each variant of geometry and displays the data as a list. This option can be used for cylindrical gear pairs, planetary stages and cylindrical gear stages that have an idler gear. If you click on the `Calculate geometry only` checkbox in the `Conditions II` tab, the calculation does not include tooth safeties.

13.16 Measurement grid

A measurement grid report is available for cylindrical and bevel gears (Calculations > Measurement grid). This report is not available for face gears and globoid worm gears.

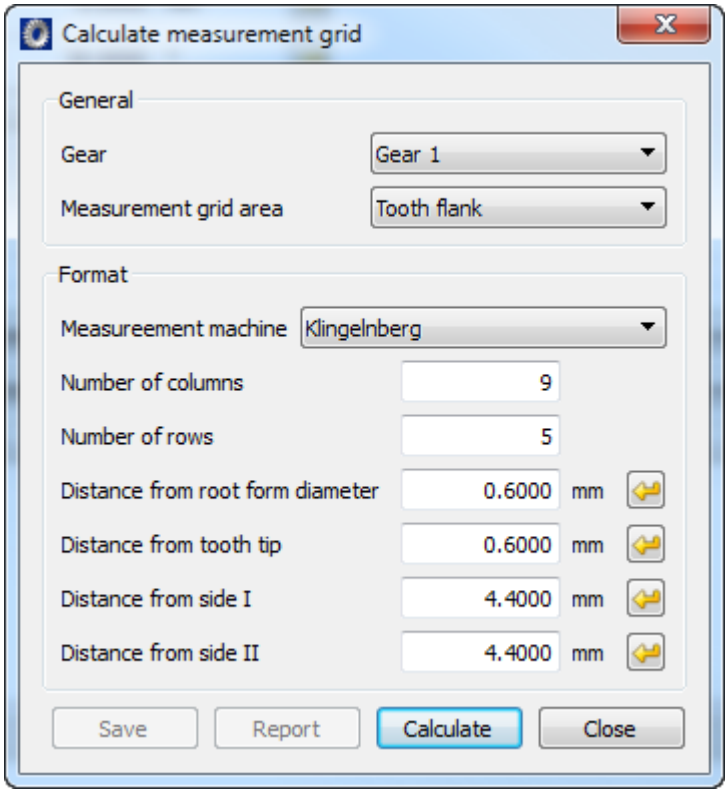


Figure 13.61: Calculating the measurement grid

Setting	Description
Gear	Setting the gear for calculating the measurement grid. If you select the "All" setting, the measurement grid will be calculated for every gear.
Measurement array	Setting the measurement array for the calculation. 0: Tooth flank 1: Root radius
Measurement machine	Setting the report format using a particular measurement machine 0: Klingelnberg

	1: Gleason
Number of columns	Setting the number of columns across the facewidth (>=3) Number of columns (number of sections – 2) for Parasolid settings, because the sections should not include both ends of a tooth.
Number of rows	Setting the number of rows across the tooth profile (>=3)
Distance from root form diameter	Distance from root form diameter Default value 0.1*normal module (middle).
Distance from tooth tip	Distance from tooth tip Default value 0.1*normal module (middle).
Distance from side I/toe	Distance from side I for cylindrical gears, distance from toe for bevel gears. Default value is (facewidth)/(number of columns + 1).
Distance from side II/heel	Distance from side II for cylindrical gears, distance from heel for bevel gears. Default value is (facewidth)/(number of columns + 1).

The report includes the co-ordinates and the normal vector of the grid points in the format [XP YP ZP XN YN ZN]. The reference point and the tooth thickness angle are displayed in the report header.

The reference coordinates of the data may differ according to which type of measurement machine is used. For example, the following convention applies to Klingelnberg machines.

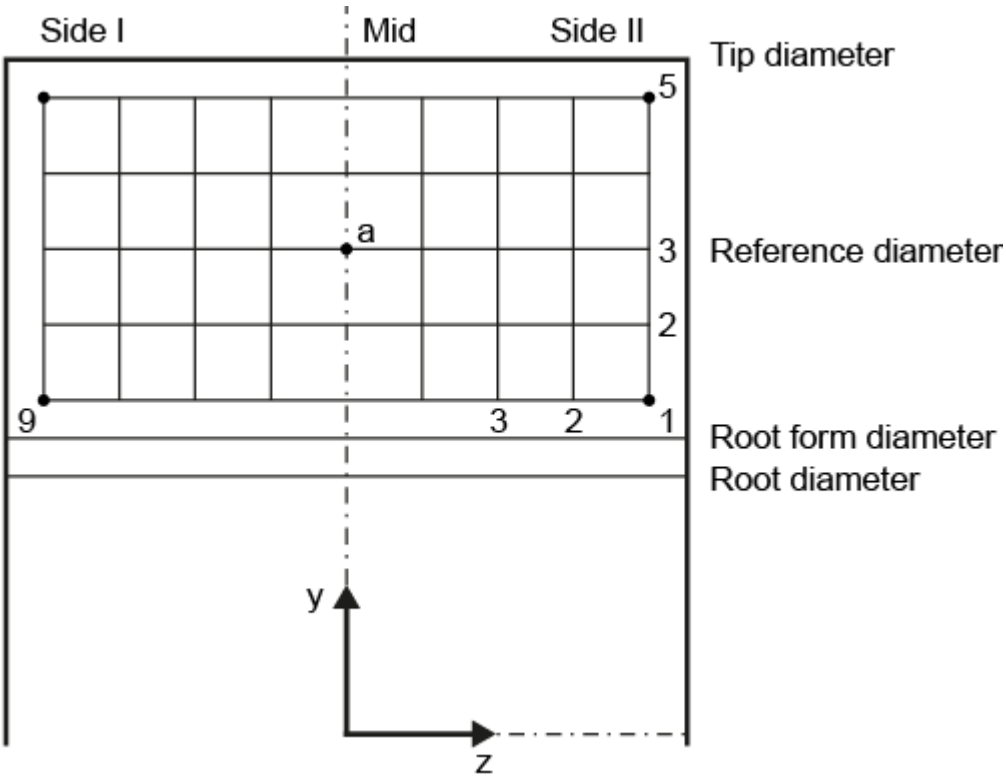


Figure 13.62: Measurement grid for bevel gears for Klingelnberg machines

The sequence of index numbers for points and sections is defined in accordance with ISO/TR 10064-6. In other words, the index for lines runs from bottom to top, and the index for columns runs from side II (heel) to side I (toe).

13.17 Profile modification optimization

To call the profile modification optimization function, click (tool bar icon), in the "Calculation" menu and then click either "Modifications optimization" or the "Optimized" button in the "Contact analysis" tab.

If you call the optimization process without first opening the "Contact analysis" tab, the default setting for this tab will be applied.

13.17.1 Conditions I/II

Conditions I

The Conditions I tab is where you define basic modifications that will not be changed and which are valid for every solution. Modifications already present in this tab are accepted automatically. You can then select the "Edit modifications from the Modifications tab" to change these values at a later date.

The "Cross-vary value and coefficient 1/2" option runs an additional variation of the coefficients with the modification value.

If the "Without contact analysis, only service life calculation with KHbeta according to ISO 6336-1, Annex E" option is enabled, the solution range is only performed using the service life and the calculation of KHbeta.

Conditions II

The Conditions II tab is where you define the modifications you want to vary. You can enter ten modifications for each gear, each one with a minimum value and a maximum value. You can also enter a maximum and minimum value for the load value wt%. By entering the number of steps per modification you can define the number of steps between the minimum value and the maximum value. If the "Synchronize with no." column contains a value that is not the one you input, the modification is synchronized with the modification you selected and all the variants are executed with the same number of steps.

13.17.2 Results

All the solutions are displayed as graphics in the "Results" tab. You can then select the solution that best suits your requirements. Click on Accept or double-click on the solution to transfer its data to the "Modifications" tab.

In the results overview, you see the following results for all the iteration steps:

- ID: ID of the solution. You can use this ID to search for more details about the results in the reports, e.g. 1:3:3
- Wt: Partial load of the calculated solution in % (depending on the number of iteration steps specified in the "Number of steps for partial load" field), e.g. 50% partial load with reference to the nominal load defined in the "Basic data" tab.
- Hmin: The minimal service life achieved by the gear pair in hours
- PPTE: Transmission error of the driven gear along the path of contact in [µm] or rotation error [°] of the driven gear.
- rel. PPTE: Relative transmission error/rotation error with reference to unmodified toothing.
- ε_a: Transverse contact ratio under load
- K_{Hb}: Face load factor (if the calculation is performed with load spectra, only the face load factor of the last load spectrum element is ever displayed)
- σ_{Hmax}: Maximum Hertzian pressure that occurs in the toothing
- Slam: Safety against micropitting as specified in ISO TR 15144
- η Efficiency
- $\Delta Wn_{A/B}$: Wear on gear A/B
- ΔT: Torque amplitude of the driven gear
- Modifications: You can display all the modifications via the Context menu (click the right-hand mouse button in the Results window).

13.17.3 Graphics

All the solutions are displayed as graphics in the "Graphics" tab. You can display a maximum of up to 10 graphics at the same time. Each graphic can process its own data record.

Select the required partial load from the partial load selection list (red is the largest partial load, blue is the smallest partial load).

13.17.4 Report

The results are documented in three different, detailed reports. We suggest you begin by looking at the summary report which gives a broad overview. The other two types of report are considerably longer, and also document intermediate results.

The main calculation performs a series of contact analysis calculations, each one having a different combination of modifications as defined in tables A to C, with all intermediate steps, and for each load level wt%. In addition, for each load level, a contact analysis without modifications is performed to provide a basis for comparison.

Transmission error (Delta)

w_t%	50%	75%	100%
-:-:-	0.1732 µm	0.2482 µm	0.3077 µm
1:1:1	2.3354 µm	2.6870 µm	3.2371 µm
1:1:2	0.9053 µm	1.0925 µm	1.5307 µm
1:1:3	2.2187 µm	1.5275 µm	1.0875 µm
1:2:1	2.5795 µm	2.9765 µm	3.5446 µm
1:2:2	1.1684 µm	1.2441 µm	1.7129 µm
1:2:3	2.4434 µm	1.6168 µm	1.2434 µm
1:3:1	2.6591 µm	3.3216 µm	3.7615 µm
1:3:2	1.2945 µm	1.4053 µm	1.8633 µm
1:3:3	2.5829 µm	1.8494 µm	1.2865 µm
2:1:1	1.5237 µm	1.9515 µm	2.5605 µm
2:1:2	1.5184 µm	1.2700 µm	1.2124 µm
2:1:3	2.6941 µm	1.9950 µm	1.5216 µm
2:2:1	1.7308 µm	2.1400 µm	2.8648 µm
2:2:2	1.7921 µm	1.4326 µm	1.4874 µm
2:2:3	2.9403 µm	2.2018 µm	1.9210 µm
2:3:1	1.8665 µm	2.4128 µm	3.0138 µm
2:3:2	2.0556 µm	1.5211 µm	1.5799 µm
2:3:3	3.2261 µm	2.4702 µm	2.1064 µm
3:1:1	1.9932 µm	1.6845 µm	1.2775 µm
3:1:2	2.9212 µm	2.5452 µm	2.0554 µm
3:1:3	3.8027 µm	3.1884 µm	2.7191 µm
3:2:1	2.0662 µm	1.7219 µm	1.3642 µm
3:2:2	3.2908 µm	2.7320 µm	2.3808 µm
3:2:3	4.3271 µm	3.6022 µm	3.0787 µm
3:3:1	2.1943 µm	1.7012 µm	1.3467 µm
3:3:2	3.5957 µm	2.8666 µm	2.4656 µm
3:3:3	4.6416 µm	3.9627 µm	3.3979 µm

Figure 13.62: Extract from the short form report

Figure 13.62 shows an extract from the documentation. The notation "1:3:3" is used to designate which modifications, from which group, have been used for the given calculation case. The first index ("**1**:3:3") corresponds to Group A, and means that the value for the first step (minimum value of Group A) has been used. A numerical value of 2 ("2:**3**:3") would mean that the modification of table A for the second step has been used (in this example the mean modification value). A 2 shows that values corresponding to step 2, etc. have been used. The second ("1:3:3") and third index ("1:3:**3**") have the same meaning for Group B and C, respectively.

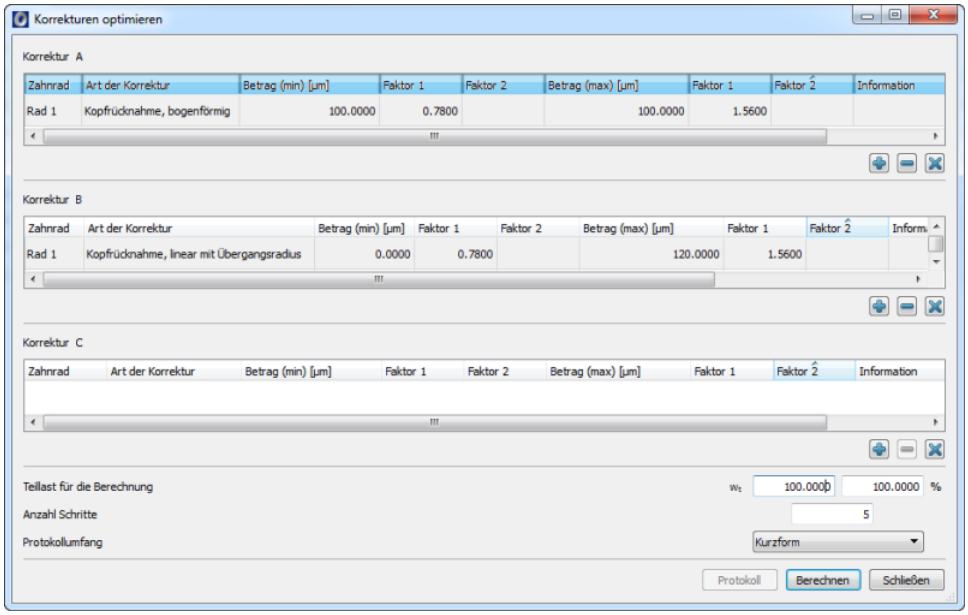
The more detailed reports use the same notation, and in addition the actual values of the modifications are documented.

A frequently asked question:

How can I use the "Optimize modifications" function to vary the length of the modification and the relief Ca independently of each other to find out which combination of length/value gives the best result?

Reply: For example, if you want to vary the tip relief Ca between 100 and 220 mm and vary the length factor between 0.78 and 1.56 to determine all the possible combinations of value - length.

Input the data like this:



13.18 Settings

You access the `Module specific settings` window by opening the `Calculation` menu and then clicking on `Settings`. A huge number of these settings are available for cylindrical gear calculations. You can therefore enable the widest variety of possible special functions. Normally there is no need to change the settings.

13.18.1 General

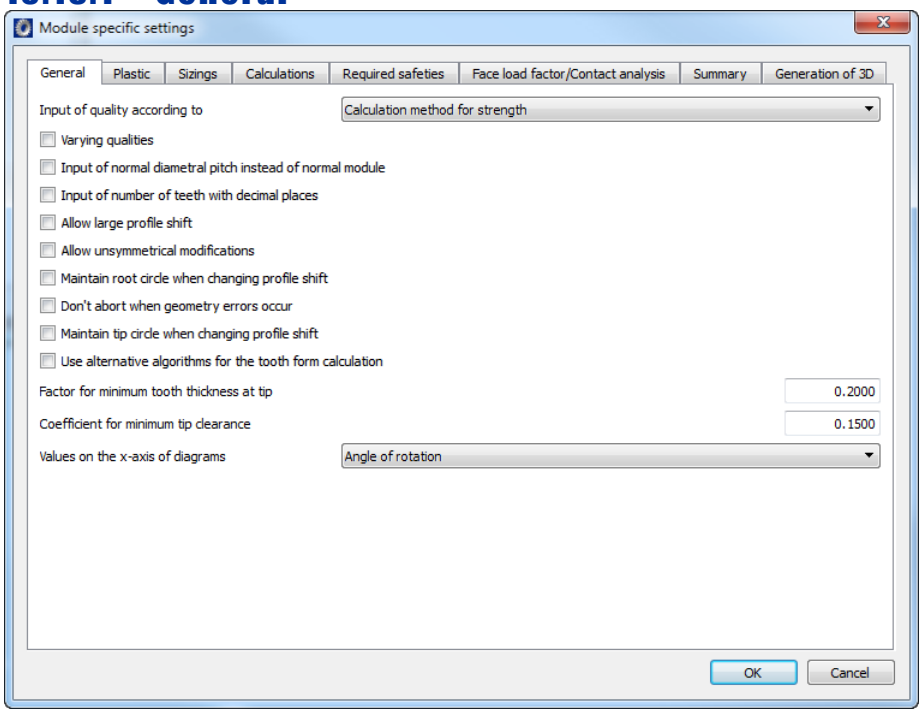


Figure 13.64: `General` tab in `Module specific settings`

13.18.1.1 Input quality

The manufacturing allowances that are output in the report and used for particular factors in the strength calculation procedure are defined either in the DIN 3961, ISO 1328 or AGMA 2015 standards. You can specify which standard is to be used. If you click the `Calculation method for strength` option, the system applies the standard that is best suited to the strength calculation method (for example ISO 1328 is used if you are using the ISO 6336 calculation method).

13.18.1.2 Varying qualities

If you select this option, the plus button next to the Quality field in the main screen appears. You can then use this to input specific tolerances manually.

You will find a more detailed description of this in Qualities (see page II-256).

13.18.1.3 Fp-Tolerance as specified in tables in DIN3962

The total cumulative pitch deviation Fp given in the tables in DIN3962 is, in some parts, very different from the Fp calculated in accordance with the formulae in DIN3961.

13.18.1.4 Input normal diametral pitch instead of normal module

If you select this option, the `Normal module` input field in the `Basic data` or `Geometry` input windows is replaced by the Diametral Pitch input field.

13.18.1.5 Input of number of teeth with decimal places

In KISSsoft, you can perform a calculation with a fractional number for the number of teeth. We recommend you use this option for arcs of circles or for nonsymmetrical teeth.

13.18.1.6 Allow large profile shift

Use this option to significantly increase the range of profile shifts that can be performed ($-1.2 \leq x^* \leq +1.5$). This is very useful for special cases. We recommend you use it for: cylindrical gears, bevel gears, worms, spiral-toothed gear wheels.

13.18.1.7 Do not cancel if geometry errors occur

If you select this option, the program will continue the calculation even if severe geometry errors, such as pointed teeth, meshing interference etc. occur. This options enables you to continue the calculation in critical cases, however, you should then use its results with extreme caution!

13.18.1.8 Retain tip circle when modifying profile shift

In the KISSsoft system, the reference profile is usually retained and the tip circle and root circle are modified to suit. If you select this option, the tip circle is

retained and the reference profile is modified to match it when the profile shift is changed. The tip circle value is retained as long as the number of teeth and the transverse module stay the same.

13.18.1.9 Maintain root circle when changing profile shift

In the KISSsoft system, the reference profile is usually maintained and the tip and root circle are modified to suit. If you select this option, the root circle is retained and the reference profile is modified to match it when the profile shift is changed. The root circle value is retained as long as the number of teeth and the transverse module stay the same.

13.18.1.10 Use alternative algorithms for the tooth form calculation

The tooth form calculation uses a very reliable algorithm for determining the points on a tooth form. However, in a few special cases this algorithm does not provide a good solution. In such situations, using an alternative algorithm may help.

13.18.1.11 Coefficient for minimum tooth thickness at tip

For manufacturing reasons, a specific minimum tooth thickness at the tooth tip must always be achieved. The minimum tooth thickness is: Module **.** Coefficient. As defined in DIN 3960, the coefficient is usually 0.2.

13.18.1.12 Coefficient for minimum tip clearance

The tip clearance is the distance between one gear's tip circle and the root circle of the other gear in the pair. You can specify a minimum tip clearance. If this clearance (taking into account the tip circle and root circle allowances) is not reached, the system displays a warning message.

13.18.1.13 Values on the X-axis of diagrams

You can select different values for the X-axis from a drop-down list.

Here you can select the rolling angle, the length (path of contact), the diameter of gear A and the angle of rotation.

You can also decide whether the X-axis (path of contact) and Y-axis (facewidth) are to appear as scales in the 3D diagrams or not at all.

NOTE

If you select the angle of rotation for the X-axis the gear axis is 0°.

13.18.2 Plastic

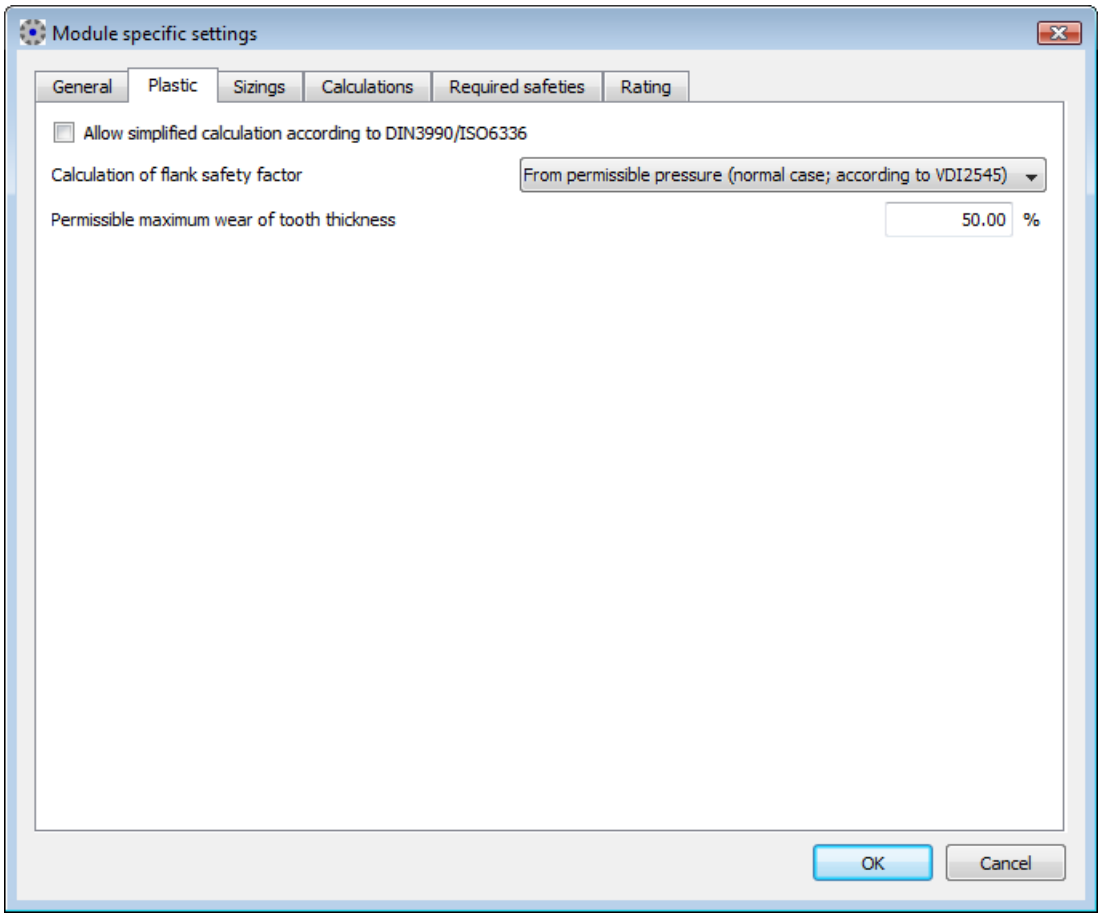


Figure 13.65: `Plastic` tab in `Module specific settings`

13.18.2.1 Allow simplified calculation in accordance with DIN 3990/ISO 6336

If you select this option, you can use the calculation methods for steel gears to calculate plastic gears. This calculation is performed in accordance with the endurance limit values listed in the materials database. The values for the plastics given in this database apply for oil lubrication, a temperature of 70° and a number of load cycles of 10^{8}. In contrast to the calculation in accordance with VDI 2545, the strength values do not depend on the temperature and lubrication type.

The calculation is performed in the same way as for heat treatable steel, with the corresponding Woehler line as defined in ISO 6336.

13.18.2.2 Calculation of flank safety

Flank safety for plastic gears is calculated as specified in VDI 2545 (in a similar way to the calculation for steel gears). In other words, flank safety is defined using Hertzian pressure with the permitted material value for pressure σ_{Hlim}. However, measurements taken from plastic gears have often shown that their tooth flanks wear in a similar way to the tooth flanks in worm gears. This is why the KISSsoft system also provides the calculation of safety against wear as an alternative. Flank safety is calculated using σ_{Hlim} if the material database (or the material file with extended data in that database) provides data for σ_{Hlim}.
Safety against wear is calculated if wear data is provided in the material database. If data for both calculations is present, the system performs both calculations. You can click the "Calculation of flank safety" option to specify which of the two safeties appears in the main window. If only data for one calculation is present, the corresponding safety is displayed automatically.

13.18.2.3 Permissible maximum wear of tooth thickness

When you calculate wear safety (see page II-432) you must specify a permitted wear threshold value. A usual value for plastic is 50% (tooth thickness wear in the reference circle). If no, or only very little, wear can be tolerated, we recommend you input a value between 5 and 10%.

13.18.3 Planets

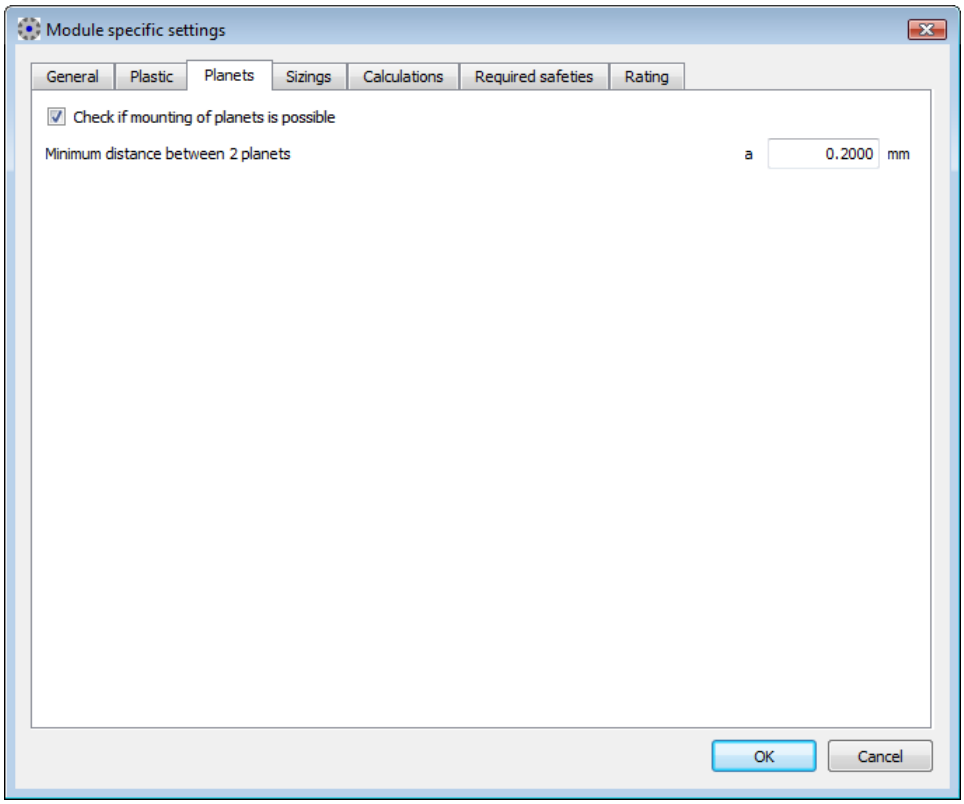


Figure 13.66: `Planets` tab in `Module specific settings`

13.18.3.1 Check if mounting of planets is possible

Planets are usually arranged with a constant pitch on the planet carrier (if 3 planets are involved, they will each be at 120 degrees etc.). In these situations, the number of teeth must fulfill certain conditions to ensure the planets can be mounted correctly. The KISSsoft performs these calculations when you click this checkbox.

13.18.3.2 Minimum distance between 2 planets

You can specify a minimum distance between the tip circles of two planets in this input field. If this minimum distance is not reached, the system displays a warning message.

13.18.4 Sizings

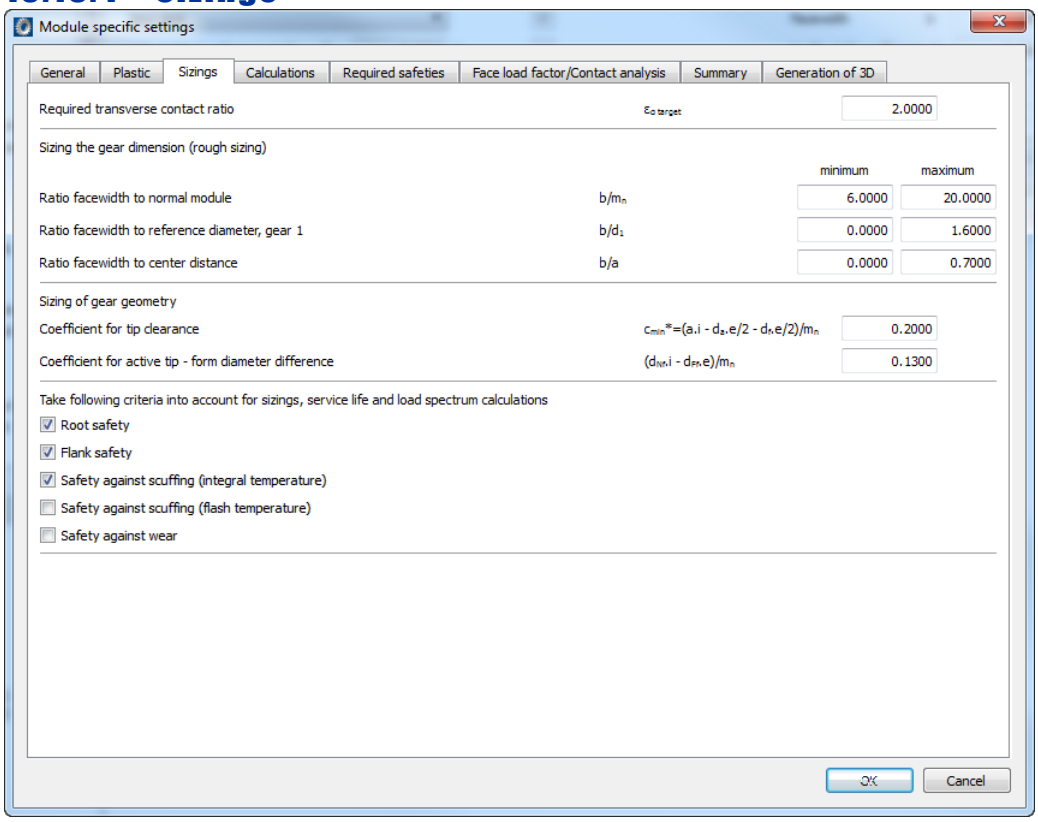


Figure 13.67: `Sizings` tab in `Module specific settings`

13.18.4.1 Required transverse contact ratio

You can specify the required transverse contact ratio for sizing deep tooth forms (see page II-612).

13.18.4.2 Ratio facewidth to normal module

The facewidth to normal module ratio is a characteristic value used to achieve reasonable dimensions for gear stages. If very narrow gears are involved, the axial stiffness of the teeth cannot be guaranteed, for this reason, b/m_n should be greater than 6 (see Niemann, Table 22.1/7 [65]).

If the gears are too wide, it is essential that the load is spread evenly across the entire facewidth, depending on the gear type and accuracy grade, b/m_n should therefore be less than 15..40 (see Niemann, Table 22.1/10 [65]).

13.18.4.3 Ratio facewidth to reference circle, gear 1

The facewidth to normal module ratio is a characteristic value used to achieve reasonable dimensions for gear stages. Depending on what type of heat treatment is involved, this ratio should be less than 0.8 to 1.6 (see Niemann, Table 22.1/5 [65]).

13.18.4.4 Ratio facewidth to center distance

The facewidth to center distance ratio is a characteristic value used to design standard gear units of modular construction. Depending on the stiffness of the housing, this value should be smaller than 0.3 to 0.5 (see Niemann, Table 22.1/6 [65]).

13.18.5 Calculations

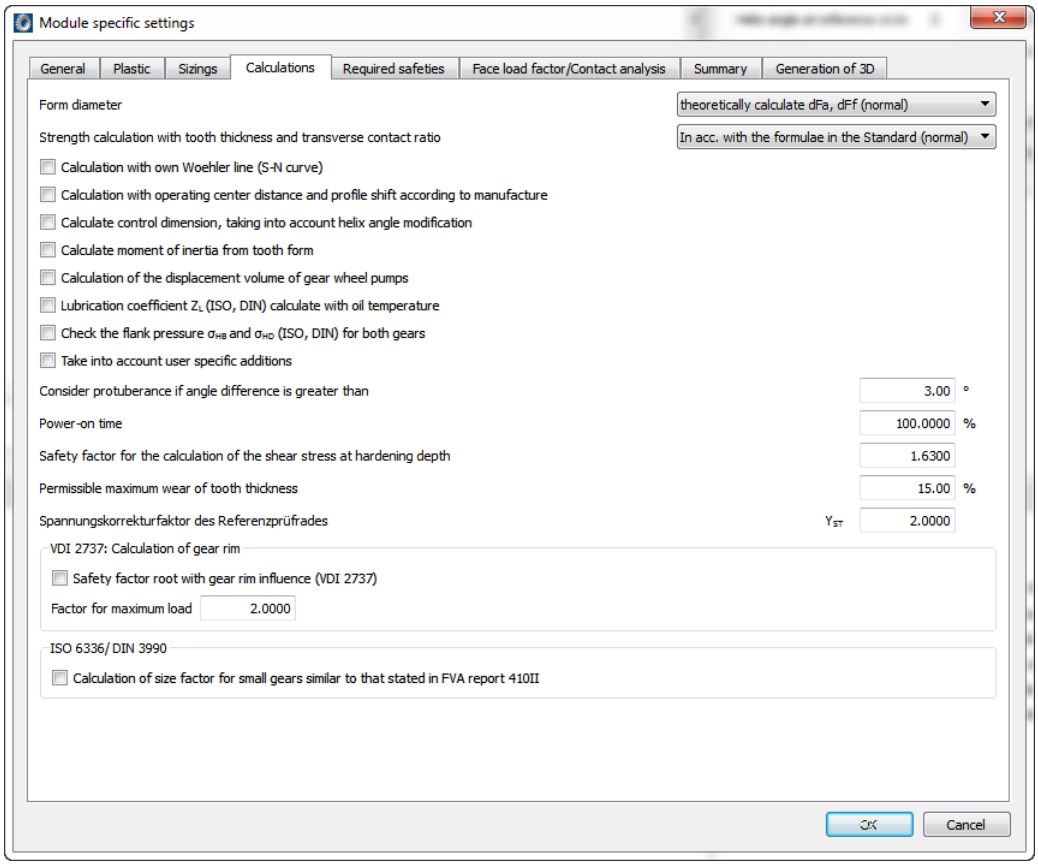


Figure 13.68: `Calculations` tab in `Module specific settings`

13.18.5.1 Calculate form diameter from tooth form

The tooth form calculation simulates the manufacturing process. In doing so it calculates the effective undercut in the tooth root. Use the `Calculate form diameter from tooth form` option to calculate the tooth form in every calculation run, define any undercut that is present and include it in the calculation. This is then used to calculate the transverse contact ratio and the root and tip form circles (generated diameters). If this option is not set, the root and active tip diameter are defined with the usual method for involutes without taking undercut into account. See, for example, DIN 3960. The warning that undercut may occur is also only derived from DIN 3960 formulae.

You can select whether the root form diameter, the tip form circle, or both these values, are to be deduced from the tooth form. Up to now, the form diameter for racks is not taken from the tooth form.

NOTE:

If this option is selected, and profile modifications have been predefined, the calculated form diameter will be at the beginning of the modification. This often results in very small transverse contact ratios $\varepsilon\alpha$.i and $\varepsilon\alpha$.e. This is correct because, at the start of the modification, the tooth form no longer exactly matches the involute. However, the message that appears to inform the user that the transverse contact ratio is too low is rather confusing. If the profile modification has been sized correctly so that meshing under load involves a whole tooth depth, this message can be ignored. This is because the transverse contact ratio under load corresponds to the theoretical transverse contact ratio $\varepsilon\alpha$. Generally speaking, we recommend you do NOT use this option for profile modifications.

13.18.5.2 Calculation using your own Woehler line

The Woehler line of metallic materials is usually defined by the endurance limit values sigFlim, sigHlim, entered in the database, and the finite life calculation values Y_{NT} (root) and Z_{NT} (flank) in accordance with ISO, AGMA or DIN. If this option is active and you input your own Woehler lines for material, the strength calculation is performed using your Woehler line.

If you use your own Woehler lines to calculate plastics, the `Calculation with own Woehler line` flag has no effect.

Notes about calculation methods using your own Woehler lines:

- Here you can use the calculation methods specified in ISO and DIN for metallic materials
- The Woehler curves are stored in a file (see under: Database). The sustainable stress (sigFadm for root and/or sigHadm for flank) of the material is defined in accordance with the number of cycles NL.
- The endurance limit values sigFlim and sigHlim, that are input directly in the database, are also required for documentation purposes and should be detailed in an appropriate interrelationship with the Woehler line data. We recommend you use the value of sigFadm/sigHadm if NL=10^7 for sigFlim/sigHlim.
- The service life factor, factor Y_{NT} and Z_{NT} is defined and reported as follows: Y_{NT} = sigFadm / sigFlim, Z_{NT} = sigHadm / sigHlim
- The other factors which influence the permitted material value, such as Ydrel, YRreIT, YX, ZL, ZV, ZR and ZW, are calculated and used in accordance with the selected calculation method (ISO or DIN). For this reason, the selected permitted material value sigFG or sigHG is not exactly equal to the value sigFadm/sigHadm from the Woehler line.

13.18.5.3 Calculation with operating center distance and profile shift according to manufacture

Cylindrical gear geometry in accordance with DIN 3960 is based on the calculation of the toothing (which is theoretically without clearance). This allows the total profile shifts for the individual gears over the center distance to be specified.

Using this option you can enter the profile shifts independently of the center distance. This is very useful as it provides a way to check the limits of a toothing (clearance, contact ratio etc.) if there are major variations in the center distance (e.g. in the case of center distance tolerance zones).

13.18.5.4 Calculate the internal temperature and the flash temperature

The calculation is performed for cylindrical gears and bevel gears. Here you can specify whether the scuffing is calculated in accordance with DIN or as specified in the selected strength calculation method as defined in ISO.

13.18.5.5 Calculate moment of inertia from tooth form

The toothing moment of inertia is calculated exactly from the tooth form in the tip to root diameter range. To achieve this, the KISSsoft tooth form calculation is run automatically for each calculation and defines the effective tooth form by the numerical integration of the moment of inertia. The result is output in the calculation report. The calculation is also performed in fine sizing and the results documented.

13.18.5.6 Calculating the displacement volume of gear pumps

This option calculates the transport volume without taking the return volume into consideration. If you activate this option, the tooth spaces are integrated numerically to calculate the transport volume and the result output in the report. In Fine sizing, the transport volume of each variant is also calculated and output. This enables you to identify, for example, the variant with the largest displacement volume.

13.18.5.7 12.17.5.6 Calculating the displacement volume of gear pumps

According to ISO 6336 (or DIN 3990), the Hertzian pressure at the single contact point B is only monitored for the driving gear, and in D the pressure is only monitored for the driven gear. This option can be used to check both gears at points B or D, depending on which one is subject to the greater Hertzian pressure.

13.18.5.8 Take into account user specific additions

If KISSsoft AG has added customer specific upgrades to the software, you can select them and view them here.

13.18.5.9 Calculate lubrication factor with oil temperature

Unlike in ISO 6336 and DIN 3990, where the calculation is always performed with an oil viscosity of J= 40oC, when you click this checkbox the lubrication coefficient is calculated with oil viscosity at operating temperature. If this option is selected, the material pairing factor ZW is also calculated with the viscosity present at operating temperature.

13.18.5.10 Strength calculation using mean position in tolerance field (of tooth form)

By default, values for the theoretical toothing (without allowances) are referenced for calculation. When you enable this checkbox, KISSsoft performs the calculation with the average allowances for the center distance, root circle and tooth thickness. This option is suitable for use where large tolerances are present.

This option has no influence on calculations performed in accordance with AGMA.

13.18.5.11 Take protuberance into account

If the angle difference (protuberance, or buckling root flank) to the pressure angle is greater than the maximum difference defined here, its influence on the tip and root form diameters as well as the transverse contact ratio are taken into account. The contact ratio then reduces accordingly.

13.18.5.12 Power-on time

The system also takes into account the power-on time when calculating the number of load cycles (multiplied by the service life).

The power-on time is also taken into account for plastic toothed gears when calculating the flank and root temperature. For worm gear units this time is also included when calculating the thermal safety.

13.18.5.13 Safety factor for the calculation of the shear stress at EHT

The safety factor is multiplied by the shear stress which is then used to calculate the hardness. The hardening depth is then defined using this value.

13.18.5.14 VDI 2737: Calculation of gear rim

The strength calculation of internal teeth is not very accurate. A significant improvement is needed. Gear rims are often subject to stresses that can affect their load capacity. At present, VDI 2737 is the only guideline that includes gear rim stress and the influences associated with this. The calculation is performed in two steps

1. Tooth root fracture safety (static and endurance) without taking the gear rim influence into account.
2. Tooth root fracture safety with gear rim influence. In this case, the maximum shear stress in the tooth root outside the meshing can in some

conditions be greater than the actual bending stress in the tooth that is under load.

The notch factor Y_S, as in ISO 6336:2006, is defined as the place at which the tangents on the flank and the tooth center line form an angle of 60°.

The results of the calculation specified in VDI 2737 are detailed in their own section in the normal report.

Coefficient for maximum load (VDI 2737)

To calculate static safety in accordance with VDI 2737, input a maximum load factor that is then multiplied with the nominal torque. To calculate the endurance limit, the nominal torque is, as usual, multiplied with the application factor K_A.

13.18.5.15 ISO 6336

If you select the `With changes (Technical Corrigendum 1 [2008]) for helix angle factor Zβ` checkbox, the helix angle factor Z_β is calculated using the corrected method

$$Z''_\beta = \sqrt{\frac{1}{\cos(\beta)}} \tag{13.26}$$

in contrast to the previous edition

$$Z_\beta = \sqrt{\cos(\beta)} \tag{13.27}$$

13.18.5.16 Calculation of size coefficients for small gears

Calculation of size coefficients for small gears similar to that stated in FVA report 410:

If mn < 1, the size coefficients (in accordance with DIN or ISO) are ZX and YX > 1.

The pre-defined required safeties should be checked.

13.18.6 Required safeties

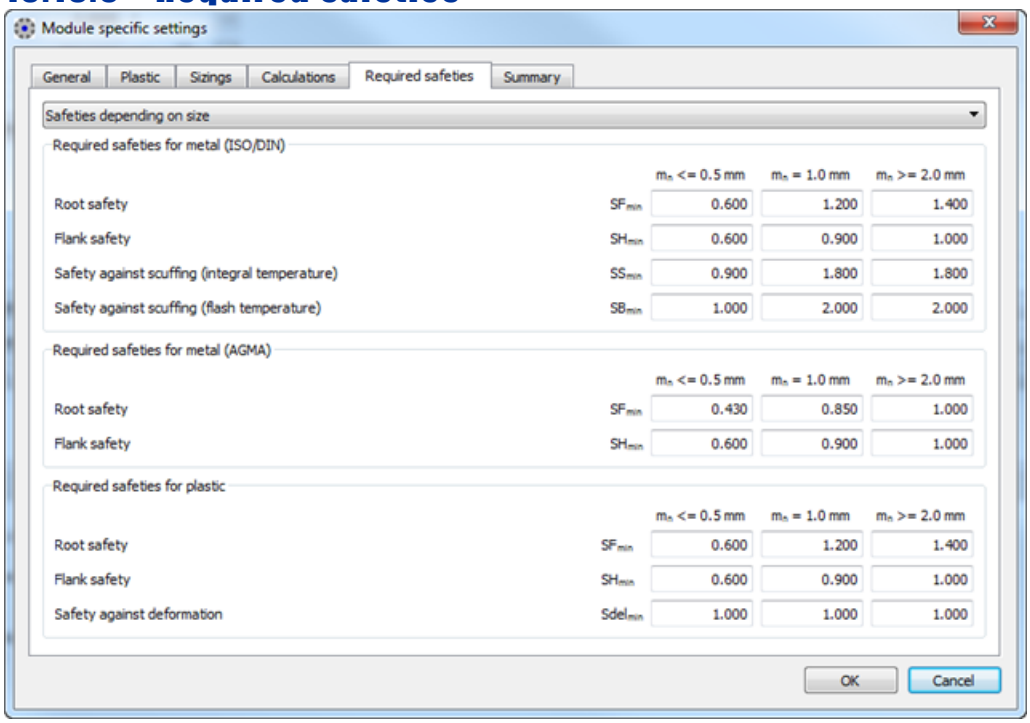


Figure 13.69: `Required Safeties` tab in the `Module-specific Settings` window

Required safeties must be predefined not only for every service life calculation but also for rough and fine sizing.

Safeties are not depending on size

Experience has shown that much lower minimum safeties can be used for smaller modules. Although the standards do not provide any information about this, this knowledge is based on experience with many different applications. However, if you do not require size-dependent safeties, you can still select the "Safeties are not depending on size" variant.

Minimum safety for calculation in accordance with AGMA

In the tooth strength calculation in accordance with AGMA 2001, the permitted tooth bending stress sat is a factor of 2 smaller than that in ISO 6336. Although its meaning is similar, the corresponding s_{at} value in the ISO guideline must be multiplied by a coefficient of 2, the reference gear's stress correction factor Y_{st}. Therefore, if the tooth strength is calculated in accordance with AGMA 2001, the resulting safety is approximately 50% smaller than that in the calculation using ISO

6336. As a consequence, the safety required for the calculation according to AGMA 2001 is smaller.

Service coefficient

Some applications of the AGMA calculation method require a predefined service coefficient. In actual fact this is merely a minimum safety. For this reason, if required, you can input Service coefficient C_{SF} for flank strength and K_{SF} for tooth bending strength.

13.18.7 Contact analysis/Face load factor

13.18.7.1 Contact analysis/Face load factor

Unit of transmission error: Here you can select either the length on the path of contact (transmission error) or the angle on the gear (angle of rotation error).

Calculation method contact stiffness: Here you can select either the calculation method defined by Weber/Banaschek [69] (dynamic stiffness analysis: default setting), the method defined in ISO 6336-1 method B and Own Input.

Single contact stiffness: Single contact stiffness calculated in accordance with ISO 6336-1 method B or Own Input in N/mm/mym.

Slices Linking factor: Linking factor used to link the individual slices of the discretized toothing model.

Border weakening factor: Border weakening factor for a weakening of stiffness on the edge of helical gear teeth.

Stiffness correction factor (in acc. with Winter): Correction factor for Hertzian flattening as described in the experiments performed by Winter/Podlesnik [95].

Draw data for path of contact: If this option is enabled the results of contact analysis are displayed quadratically in the 3D diagrams.

Tooth contact stiffness

This defines whether the tooth contact stiffness is calculated in accordance with ISO 6226 ($C_{\gamma\beta}$) (default) or whether, as specified in AGMA 927-01, it is constant with C_m = 11 N/mm/µm.

Axis alignment if load spectrum is applied

Here you can set the inclination and deviation error of the axis ($f_{\Sigma\delta}$, $f_{\Sigma\beta}$) and the tooth trace deviation ($f_{H\beta}$, f_{ma}) as a proportion of the partial load w_t of the current load spectrum element or set it as a constant.

Note: The calculation of $K_{H\beta}$ does not include triangular end relief and twist.

Calculate torsion moment of resistance

If the calculation of torsion due to deformation in the "Define axis alignment" dialog is set to "Own Input" the diameter specified here is used in the calculation.

13.18.8 Summary

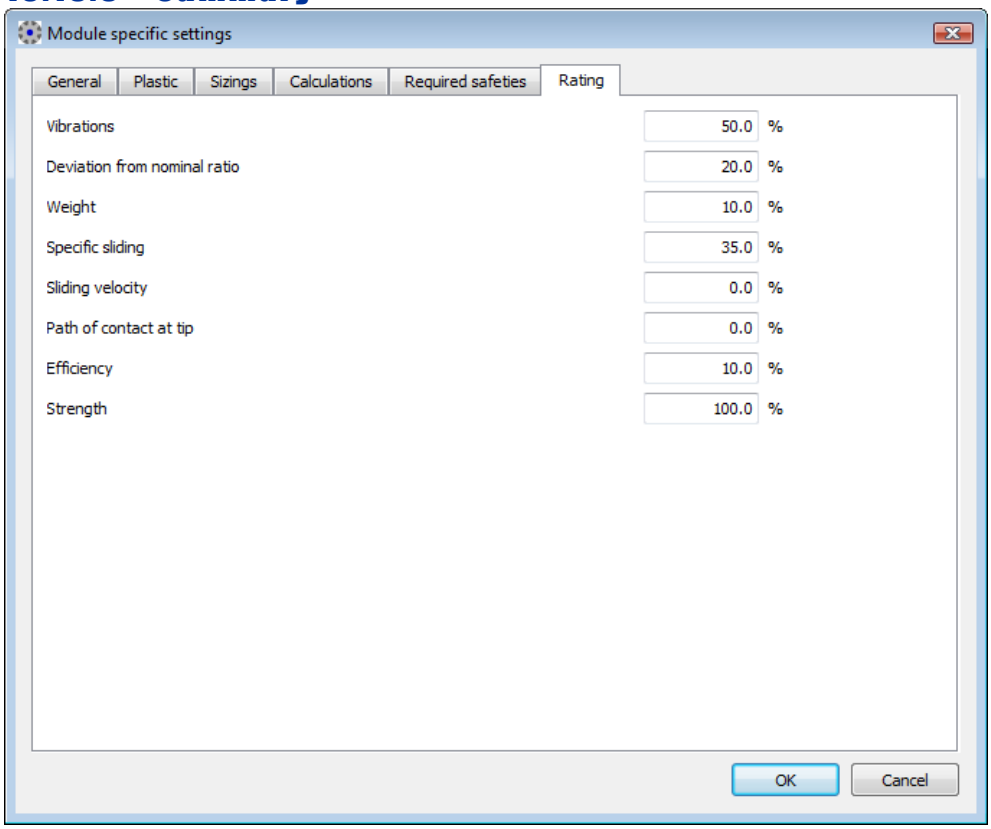


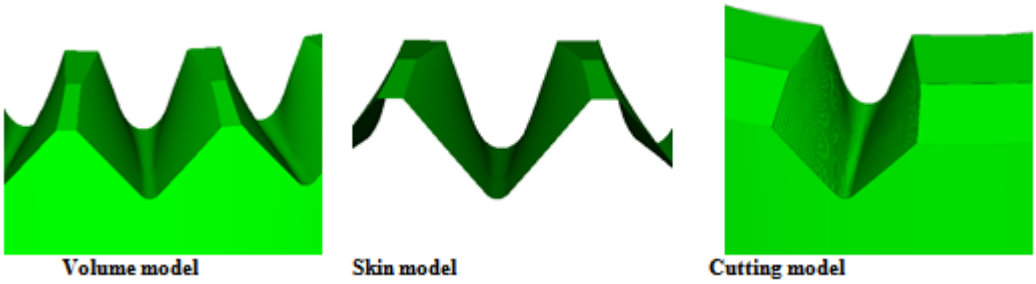
Figure 13.70: `Summary` tab in the `Module specific settings` window

Weighting the individual component to evaluate the `Summary` coefficients in Fine Sizing. (see section "Results" on page II-417)

13.18.9 Generation of 3D model

This is where you modify the parameters used to generate 3D models.

- Under **"Model"**, specify the type of model to be generated (volume model, skin model, cutting model). The volume model can be used for other purposes such as machining by CNC or finite element analysis. The skin model is most suitable for contact analysis. The cutting model is only suitable for the gear models that use cutting simulation, such as face gear and enveloping worm gear, and is used to view the actual cutting simulation.



Volume model Skin model Cutting model

- The **Number of cutting steps** sets the number of cuts per half pitch for the cutting process. The minimum value is 1, and the default value is 20. The quality of the final model can be increased by increasing the number of steps, but this also increases the probability of manufacturing errors. The "Scale factor" is used for solving the failure problem. If the operation fails, we recommend that you use a lower number of generation steps with a larger scale factor.
- The **Number of sections along facewidth** defines the number of sections along the facewidth for approximating the tooth flank form. The minimum value is 2, and the default value is 11. Normally, the quality of the final model can be improved by increasing this value, but we do not recommend that you set a number that is excessively high, compared with the facewidth. The factor is used for the gear models using cutting simulation and gear models using multiple section curves such as spiral bevel gear and cylindrical gear with lead modification.
- The **Scaling factor for the cutting model** is used to zoom the model during the cutting simulation process. The minimum value is 1, and the default value is 10. Sometimes the cutting simulation can fail due to an internal operation error in the Parasolid kernel, especially when the model has a very small module and/or a large number of generation steps. In order to prevent this type of operating error, use the model zoomed by the scale factor in the cutting process. Consequently the cutting model can have different dimensions than the actual design. However, the volume and skin models are automatically zoomed back after the operation and therefore have the same dimensions as the entered gear.

- The **Modeling operation** sets the tolerance for the internal operations of the Parasolid kernel, such as the chordal approximation and clash detection in Boolean operations. The default value is 1 µm.
- The **Rendering quality** sets the resolution of the resulting graphics in the 3D geometry viewer. This is used only to improve the viewer display (usability) and does not affect the quality of the generated model. If the rotation operation in the viewer is slow, you can increase the quality value to speed up the operation. The default value is 5 µm.
- Click on "**Constant root radius along the facewidth**" to specify the method used to generate the root radius of a bevel gear. The root radius of the bevel gear changes by the factor of the normal module along the facewidth. If you activate this flag, we use the constant root radius defined by the normal module in the middle section. (Available for bevel gear)
- Click on "**Constant protuberance along the facewidth**" to define the protuberance of the bevel gear's reference profile. The protuberance of the reference profile of the bevel gear changes by the factor of the normal module along the facewidth. If you activate this flag, we use constant protuberance defined by the normal module in the middle section. (Available for bevel gear)
- If "**Display 2D geometry for outside and inside**" is selected, the tooth forms on the internal and external sides are represented as a 2D graphic. (Available for bevel gear)
- If "**Generate tooth system model in the saved position**" is selected, the system model is generated at the position you saved. This position is saved in the calculation file, and you will be able to restore the checking position of the contact pattern in future. (Available for bevel gear)
- Click on "**Number of points on the edge of cut for spline approximation**" to specify the number of modeling (intermediate) points on each edge that are used to approximate the spline curves for the root area or the tooth flank. The figure shows a diagram in which the points that are to be used are scanned. The end points (nodes) are removed since they are giving extreme waviness on the curve. We use only the intermediate points on the cutting edge supposing the parametric distance between the points is the same. We usually recommend that more points are used in the root area. However, this model will help the user determine the optimum value. (Available for enveloping worm gear)
- Click on "**Oversize factor for worm wheel cutter**" to define the coefficient used to increase the worm wheel cutter. There are different methods of implementing the oversize hob. Those methods include the axial pitch method, the base pitch method, the extra thread method, and the normal pitch design method. KISSsoft is using the normal pitch method since the method is practically regarded as the industry standard. These methods are based on the

principle, that the worm wheel cutter uses the same normal pitch and the same normal pressure angle in the normal section as the worm. The cutting distance between the hob and the gear will then be changed accordingly, to ensure a consistent result for the root and tip diameters on the gear. If you are using the oversize factor, the generated surface will not be conjugate with the worm surface and will not give the best contact pattern. Thus, especially if you are not using conventional cutting method but want to use theoretical surface geometry, you should not use the oversize factor. In practical case, the tooth thickness of the cutter will be increased to consider the tooth thickness tolerance of the worm wheel. In that case, it's recommended to use small oversize factor compensating the tolerance to get the best contact. (Available for enveloping worm gear)

- Click on "**Cutter shaft angle change**" to modify the shaft angle of the worm wheel cutter during the simulated milling run. The angle can be both positive and negative, and the positive angle is defined as shown below. (Available for enveloping worm gear)
- Click on "**Change in pressure angle of the worm wheel cutter in normal section**" to set the worm wheel cutter to a different pressure angle than the worm. (Available for enveloping worm gear)
- Click on "**Flank shape of worm wheel cutter**" to set a different tooth form for the worm wheel cutter than for the worm. There are many research works justifying the different combinations of the tooth form to get better contact pattern in worm gearing. This setting is used for this purpose. If this option is not selected, the same tooth form is used for both the worm wheel cutter and the worm. (Available for enveloping worm gear)

13.19 Tooth thickness

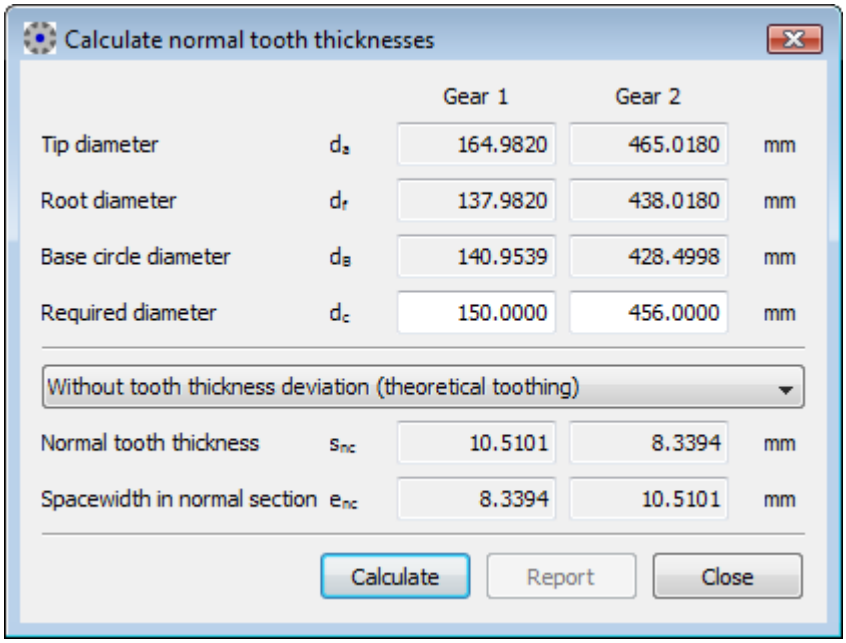


Figure 13.71: `Chordal tooth thicknesses` dialog window

Click the `Calculation > Tooth thickness` menu item to calculate the normal tooth thickness and the normal space width for any diameter.

The tooth thickness is output as an arc length and as a chordal length. To help measure the tooth thickness, the chordal height is output along with the tooth thickness allowances.

Chapter 14
Bevel and Hypoid gears

Calculation of the geometry and strength of straight, helical and spiral toothed bevel gears (gear axes intersect, offset (center distance) is 0) and hypoid gears (crossed gear axes, offset (center distance) not 0). Geometry as specified in ISO10300, ISO23509 and DIN3971, tolerances in accordance with ISO17485 and DIN 3975, strength calculation as specified in ISO10300 (replacement cylindrical gear toothing method), AGMA 2003, DIN 3991, or Klingelnberg in-house standard KN3030. The calculation only includes the geometry of bevel gears insofar as is necessary for the Strength calculation (see section "Methods used for strength calculation" on page II-470), no matter which manufacturing process is used.

14.1 Underlying principles of calculation

14.1.1 General

The geometry of bevel gears is calculated in accordance with ISO10300, ISO23509 or DIN 3971. The strength calculation is performed in two steps. A virtual cylindrical gear toothing is defined first. This is then used for the strength calculation in a similar way to cylindrical gears. The process is described in [24], [45] and [66].

Bevel gear machine tool manufacturers (such as Klingelnberg in Germany) also have their own methods that differ slightly from the processes mentioned above.

Hypoid bevel gears and bevel gears with offset (center distance) are primarily used in vehicle axle drives. Strength is calculated by defining a virtual cylindrical gear toothing. The tooth root, flank and scuffing safeties, which are important for hypoid gears, are calculated as specified in the Klingelnberg in-house standard KN3030.

14.1.2 Overview of the bevel gear manufacturing process and the terminology used in it

Various manufacturing processes are used to create bevel gears. Unlike cylindrical gears, the tooth length forms and tooth depth forms differ according to which manufacturing process is used. In particular, the process used to manufacture spiral teeth bevel gears uses a multitude of terms, the most important of which are described below.

The most important differences are shown in the tooth length form, which can be manufactured as arc of circle (face milling process), epicycloid or involute toothings (face hobbing process). Arc of circle teeth were developed by the company Gleason and are the result of the face milling principle. Here, every gap is milled separately and then the gear is rotated further by the width of that tooth space. Epicycloid toothing is used by Oerlikon and Klingelnberg. In this process the gear rotates constantly during the milling process. Only the palloid manufacturing process is used to create the involute tooth length form. Although nowadays, Klingelnberg and Gleason, the market leaders in machine manufacturing, can produce toothing using both the face milling and face hobbing processes, these companies are still associated with their traditional processes in the technical literature You will find more details in section 14.1.3 and 14.2.1.

Although alternative processes for spur gears are available, they are not listed here.

14.1.3 Calculation in accordance with Klingelnberg, Gleason and Oerlikon

The strength calculation defined in ISO 10300 DIN 3991 only includes the relationships (module, helix angle) in the middle of the facewidth in the replacement cylindrical gear toothing method calculation. The shape of the bevel and the process used to manufacture it are ignored. As a result, the KISSsoft strength calculation method can be applied no matter which process is being used, especially for Klingelnberg and Gleason This also reflects the experience that the capacity of spiral bevel gears is only slightly affected by the manufacturing process.

The geometry calculation procedure in KISSsoft defines the dimensions, such as diameter and tooth thickness, in the middle of the facewidth. It also calculates the diameter at the outside and inside end of the facewidth. These dimensions depend on the bevel type. However, the results may differ from the actual conditions because the processes are not described in sufficient detail. This is particularly true for the Gleason process.

- **Klingelnberg procedure:**
  The `Bevel gear (KN3028 and KN3030) and Hypoid gears (KN3029 and KN3030)` calculation methods enable to you calculate geometry and strength and check the manufacturing process in accordance with the Klingelnberg in-house standard. However, these methods do not calculate the machine settings for the selected Klingelnberg machine. When you input formula data from a Klingelnberg program, you must remember that the toothing data, such as module and helix angle, always applies to the middle of the facewidth (unless otherwise specified).

- **Gleason process:**
  Bevel gears are often designed by the Gleason company. Depending on which calculation program Gleason uses, toothing data such as the module and helix angle, is either predefined for the outside end of the facewidth or for the middle of the facewidth.
  The `Conversion from GLEASON data sheets` dialog window allows you to convert Gleason data from the outside end of the facewidth into data for the middle of the facewidth (see page II-452). Once this data has been converted, you can perform the strength calculation. Although the bevel dimensions (tip and root diameter) do not always exactly match the actual geometry they are close enough to enable you to check the assembly conditions (in a drive). This procedure does not check to see whether the part can be manufactured on Gleason machines.

- **Oerlikon process:**
  The Oerlikon process is roughly similar to the Klingelnberg process (select Klingelnberg bevel type).

14.2 Basic data

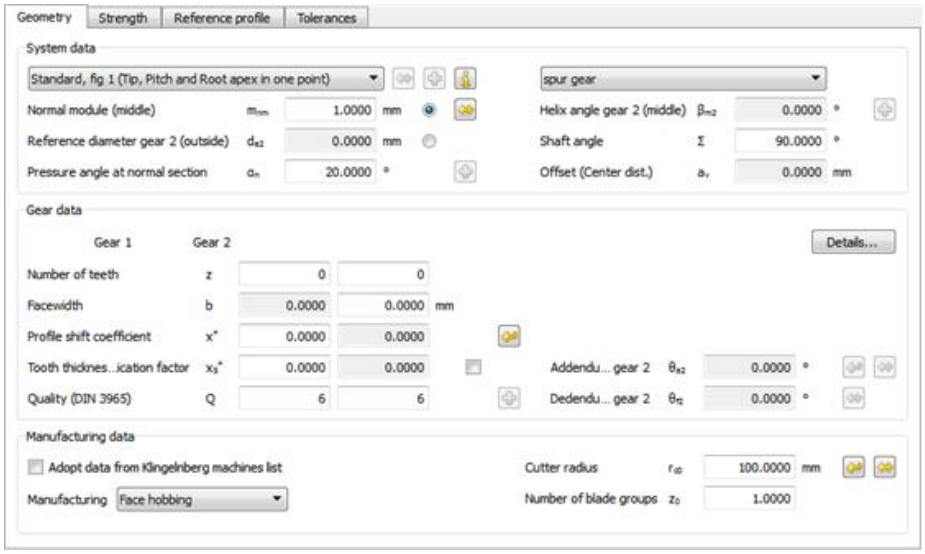


Figure 14.1: `Basic data` input window

14.2.1 Type

You will find a drop-down list for the type on the top left of the screen in the Geometry tab.

As you can see in Figure 14.2 the following bevel gear types are available for selection:

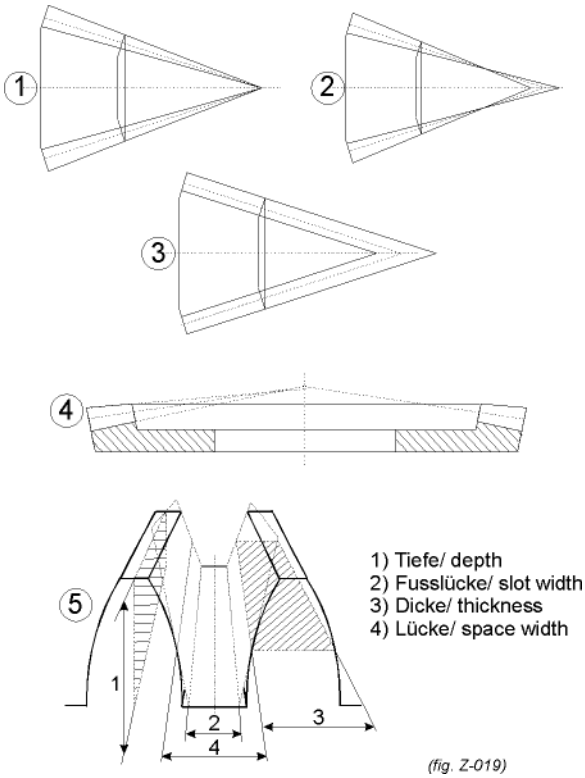


Figure 14.2: Basic types of bevel gears

- Standard, Figure 1 (tip, reference and root cone apex in one point)
  The geometry is calculated in accordance with ISO 23509. No offset (center distance) is taken into account. If you click the sizing button, the cone angle is calculated so that the crossing point of the gear axes meet each other (similar to the standard specified in ISO 23509, Annex C.5.2). In this case, the tip clearance is not constant. Typical applications include form-forged, injection molded, or sintered, bevel gears, such as differential bevel gears.
- Standard, Figure 4 (reference and root apex in one point)
  The geometry is calculated in accordance with ISO 23509. No offset (center distance) is taken into account. If you click the sizing button, the cone angle is calculated as specified in the standard (ISO23509, Annex C.5.2). The tip clearance is constant.

- Standard, Figure 2 (tip, reference and root cone apex NOT in one point)
  The geometry is calculated in accordance with ISO 23509. No offset (center distance) is taken into account. If you click the sizing button, the cone angle is calculated as specified in the standard (ISO 23509, Annex C.5.2). However you can also input values for the addendum angle and the dedendum angle manually. The cone angle of the counter gear is calculated by taking a constant tip clearance into account.
- Constant slot width, Figure 2 (Gleason)
  The geometry is calculated in accordance with ISO 23509. You can perform this calculation either with offset (method 1, hypoid gears) or without offset (method 0, spiral bevel gears). If you click the sizing button, the cone angle is calculated with a "constant slot width" (ISO 23509, Annex C.5.2). The tip clearance is constant. Gap 2 in Figure 5 does not change. A typical application of this is a ground bevel gear toothing in the Completing process (duplex), where the pinion and the bevel gear are each ground in one work step. This process requires machines that have an additional helical motion.
- Modified slot width, Figure 2 (Gleason)
  The geometry is calculated in accordance with ISO 23509. You can perform this calculation either with offset (method 1, hypoid gears) or without offset (method 0, spiral bevel gears). If you click the sizing button, the cone angle is calculated with a "modified slot width" (ISO 23509, Annex C.5.2). Gap 2 in Figure 5 changes. A typical application is the 5-section process, where the pinion is manufactured with 2 different machine settings and consequently a modified slot width is created. The bevel shape is often also referred to as a TRL (Tilted Root Line). The toothing can be either ground or lapped.
- Constant tooth depth, Figure 3 (Klingelnberg)
  The geometry is calculated in accordance with ISO 23509. You can perform this calculation either with offset (method 3, hypoid gears) or KN3028 and KN3029, or without offset (method 0, spiral bevel gears). The tip and root cone are parallel. Applications are the cyclo-palloid® process and the palloid process. After quench hardening, the cyclo-palloid® toothing can be either be hardened (HPG, HPG-S) or be lapped. Palloid toothing is characterized by an evolvent tooth length form with a constant normal module over the facewidth. After quench hardening, the toothing is usually lapped.
- Constant tooth depth, Figure 3 (Oerlikon)
  The geometry is calculated in accordance with ISO 23509. You can perform this calculation either with offset (method 2, hypoid gears) or without offset (method 0, spiral bevel gears). The tip and root cone are parallel. Applications are Oerlikon processes such as Spiroflex and Spirac. After quench hardening, the toothing is usually lapped.

14.2.1.1 Converting or inputting Gleason toothing data

The "System Data" group in the "Geometry" tab has a selection list (drop-down list) in its top left-hand corner. If the Gleason variant with "constant slot width" or "modified slot width" is selected here, the conversion and plus buttons are active. These two buttons allow you to input data in accordance with the Gleason definition.

- Select the conversion button if a Gleason data sheet is present. You can then input the data in the window as shown in Figure 14.3 and then click Calculate. Once the calculation is complete, the Report and Accept buttons become active. Click on the Report button to generate a short report. If a complete report is needed the user must click the button in the main window. Click the Accept button to transfer the data to the main window.

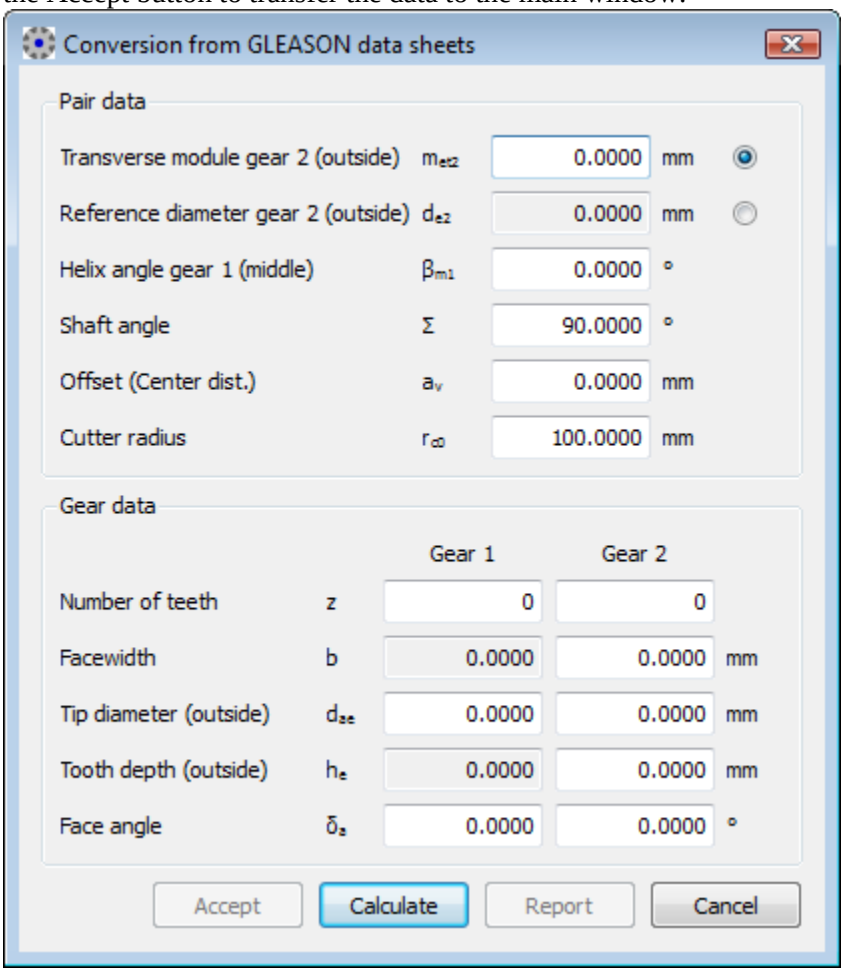


Figure 14.3: Conversion from Gleason data sheets

If you click the Plus button, the dialog window shown in Figure 14.4 appears. You can input bevel gear data directly here using the Gleason method Although the geometry results will not match the Gleason data sheet exactly, they are good enough for calculating strength in accordance with ISO 10300 (or AGMA or DIN).

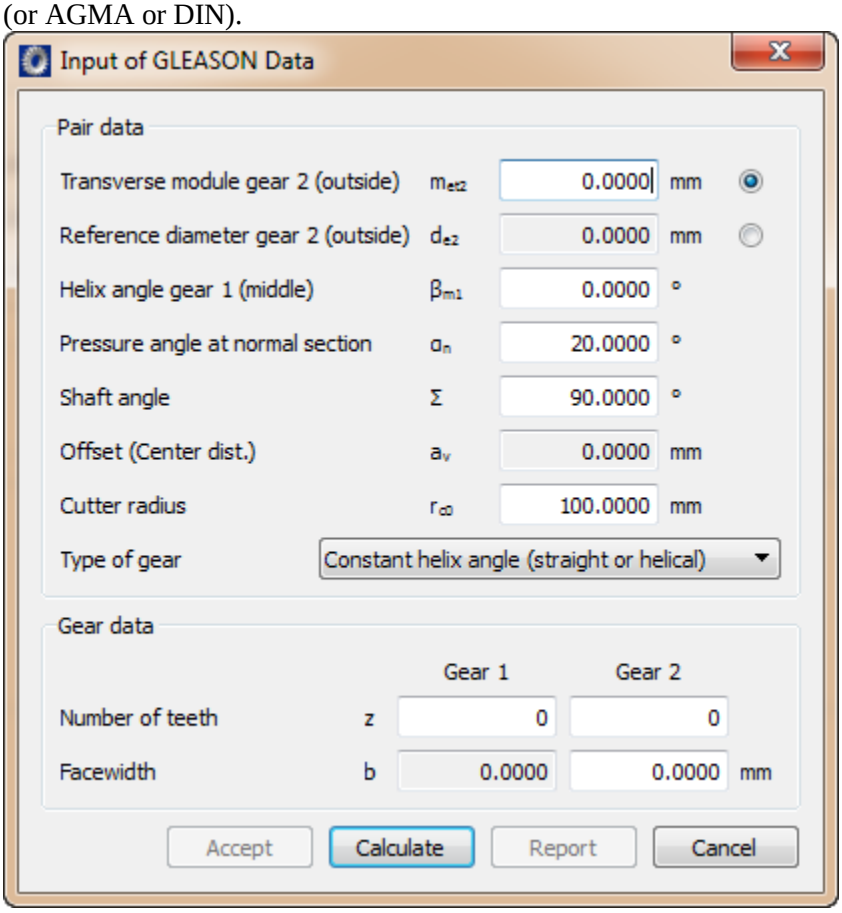


Figure 14.4: Inputting Gleason data

In the drop-down list on the right of gear type the user can choose between different types of Gleason design (by default straight gears are selected):

1. `Constant helix angle (straight or helical)`
   A constant helix angle represents a bevel gear with a constant helix angle. If necessary, you can modify the helix angle to compare the geometry data with the Zerol geometry data. If you click the Accept button to close the dialog, the calculation is usually performed with the selection "Default, Figure 4 (part and root apex in one point)".

2. `Duplex (constant slot width)`
   The term "duplex" refers to bevel and hypoid gears that are manufactured with a constant slot width across the entire tooth length of both gears. These gear types usually have a spiral angle of 35° in the middle of the facewidth with a continuously changing spiral angle across its width. If you selected Duplex (constant slot width) and then clicked the Accept button to close the dialog, the calculation is usually performed with a "Constant slot width".

3. `Spiral toothing, default (modified slot width)`
   These gear types usually have a spiral angle of 35° in the middle of the facewidth with a continuously changing spiral angle in the axial direction. This gear type is described as having a "modified slot width". If you select this gear type and then click on Accept, the calculation is usually performed with a "non-constant root gap". In this case the root gap of the gear pair is constant over the entire tooth length and any gap modifications are performed on the pinion.

4. `Zerol "Duplex taper"`
   This is a Zerol design (see Zerol), but a root angle variation is performed to achieve duplex dimensions. If you select Zerol duplex and then close the dialog by clicking the Accept button, the calculation is usually performed with the "Constant slot width" selection.

5. `Zerol "standard"`
   The Zerol standard is a gear pair with a spiral angle of less than 10° in the middle of the facewidth, with a continuously changing spiral angle in the axial direction. The inner spiral angle is usually negative. To ensure the program can take into account the change across the tooth length, a value of b=0.001 is assumed for the case b=0. If you close the dialog by clicking the Accept button, the calculation is usually performed with a "Modified slot width".

14.2.2 Normal module (middle)

The reference circle of the external end of the bevel of Gear 2 (de2) is usually specified for bevel and hypoid gears. In the basic "Oerlikon" type the reference circle of Gear 2 is predefined in the middle (dm2). Alternatively, you can specify the normal module in the middle of the facewidth. However, if you know the pitch, the transverse module, or the diametral pitch, instead of the normal module, click

the button to open a dialog window in which the conversion will be performed. If you want to transfer the diametral pitch instead of the normal module, you can select `Input normal diametral pitch instead of normal module` under `Calculation > Settings > General`.

14.2.3 Reference diameter gear 2

If there are changes to the sizes of the set of bevel gears, click the button to enter the new outer reference diameter of Gear 2. This is useful for designers since, very often, the amount of space available for installing the larger gear is predefined. The module is then recalculated (not optional).

14.2.4 Pressure angle at normal section

For standard toothings the pressure angle is $\alpha_n = 20^o$. You can use smaller pressure angles for a larger number of teeth to achieve higher contact ratios. Greater pressure angles increase the strength and allow a smaller number of teeth to be used without undercut. In this situation, the contact ratio decreases.

For hypoid gears, click the button to input the pressure angle for the driving flank and the driven flank independently of each other. The driving flank is the concave flank of the pinion and the convex flank of the gear. The driven flank is the convex flank of the pinion and the concave flank of the gear.

14.2.5 Pressure angle driving/driven flank: Hypoid gears

Bevel gears are usually better able to withstand stress when driven by the concave pinion flank, i.e. when the spiral on the pinion and its sense of rotation run in the same direction.

The concave flank of the pinion is usually called the driving flank (index D for "Drive"), and the convex flank is known as the driven flank (index C for "Coast"). In the gear, the concave flank is the driven flank (index C) and the convex flank is the driving flank (index D). Since the effective nominal pressure angle on the driving flank is greater by the amount of the edge pressure angle, and on the driven flank it is smaller than the pressure angle in a normal section, by the amount of the edge pressure angle, the nominal pressure angle driving flank and driven flank can be entered independently.

For hypoid gears, as specified in ISO 23509, you should input the nominal design pressure angle as α_{dD}, α_{dC}. This is used to calculate the generated pressure angle ("effective pressure angle") α_{nD}, α_{nC} and the effective pressure angle α_{eD}, α_{eC} respectively for the driving (index D for "Drive") and driven side (index C for "Coast").

The equations specified in ISO23509 are:

$\alpha_{nD} = \alpha_{dD} + f\alpha_{lim} * \alpha_{lim}$

$\alpha_{eD} = \alpha_{nD} - \alpha_{lim}$

If, as a result α_{nD} has been specified, α_{dD} can be calculated as follows:

$\alpha_{dD} = \alpha_{nD} - f\alpha_{lim} * \alpha_{lim}$

$\alpha_{dC} = \alpha_{nC} + f\alpha_{lim} * \alpha_{lim}$

or if α_{eD} has been specified, α_{dD} can be calculated like this:

$\alpha_{dD} = \alpha_{eD} + \alpha_{lim} * (1- f\alpha_{lim})$

$\alpha_{dC} = \alpha_{eC} - \alpha_{lim} * (1- f\alpha_{lim})$

The boundary pressure angle α_{lim} is calculated by KISSsoft and output in the report.

The influencing factor of the boundary pressure angle $f\alpha_{lim}$ has been introduced so that you do not always need to take the total value of the boundary pressure angle into consideration when calculating the flank angle on the tool. For forming tools (Klingelnberg process), $f\alpha_{lim} = 0$ is set. If you use the procedure with a constant slot width (Gleason) $f\alpha_{lim} = 0.5$ is set, otherwise $f\alpha_{lim} = 1.0$ is often used.

However, if precise data is not available, you can use the pressure angle in the normal section in the calculation (with $\alpha_{dD} = \alpha_{dC} = \alpha_n$ and $f\alpha_{lim} = 1.0$).

NOTE

These input fields are only available if you are calculating the strength of hypoid bevel gears (see section "Methods used for strength calculation" on page II-470).

14.2.6 Helix angle

The helix angle is transferred in the middle of the facewidth. In the case of helical bevel gears, the angle remains constant across the facewidth. However, in spiral bevel gears the spiral angle changes across the facewidth. As the same input screen is used for both straight flank and spiral toothed bevel gears the term "helix angle" can be selected for both types selected.

In hypoid gears, the spiral angle is specified in the middle of the facewidth for Gear 2. This value is then used to calculate the value for Gear 1 (pinion).

You can select any value as the helix angle in the middle of the facewidth. However, we recommend you use a larger angle of between 30 and 45° to ensure optimum running performance. You should only select a value that is less than this guide value if the bearing load has to be reduced.

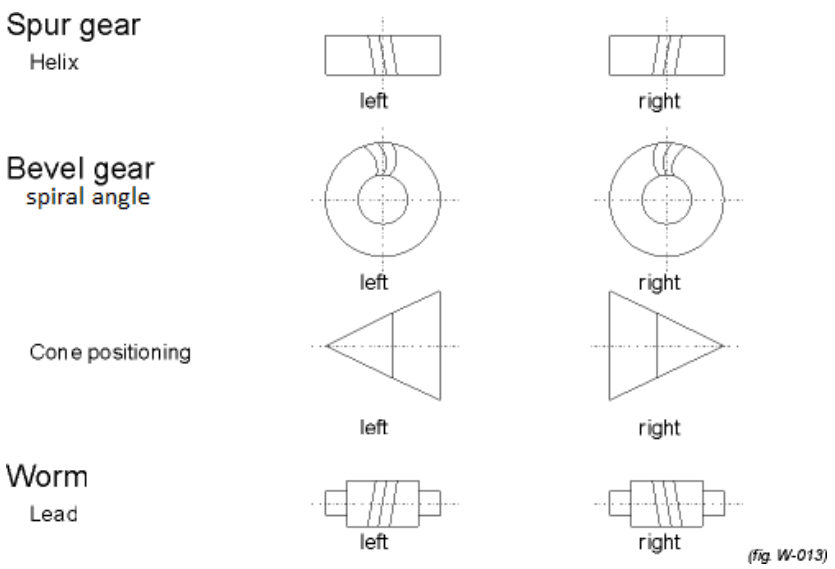


Figure 14.4: Helix angle

Click the button to the right of the helix angle input field to open the `Additional data for spiral teeth` window where you can input the internal and external helix angle for spiral bevel gears. Click the `with spiral teeth` checkbox to enable the input fields.

NOTE

In most cases, however, the internal and external spiral angle is calculated by the selected process, either arc of circle or continual toothing, and the cutter tip size [ISO 23509].

If no data has been input for the cutter tip you can usually input an external helix angle that is approximately 5° larger and an internal helix angle that is approximately 5° smaller than the helix angle in the middle for Gleason bevel gears.

14.2.7 Shaft angle

The shaft angle for bevel gears is usually 90°. However, you can perform the calculation for any shaft angle.

14.2.8 Offset (Center dist.)

In the case of bevel gears without offset (center distance) the axes of the bevel gears intersect at one point. In the case of bevel gears with offset (center distance) the axes do not intersect. This application allows you to achieve a higher contact ratio and greater strength at the tooth root. It is primarily used in the vehicle construction industry. This pairing is known as a hypoid bevel gear and is illustrated in Figure 14.5.

NOTE

A positive offset (center distance) is almost always applied to hypoid bevel gears, because this is the only way of achieving the improvements to the characteristics described above.

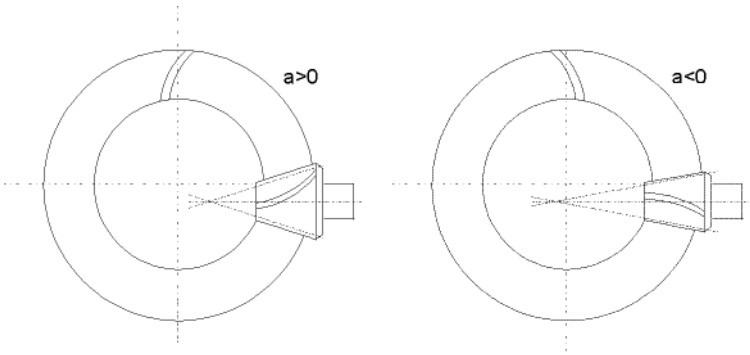


Figure 14.5: Hypoid bevel gear configurations. Positive offset (center distance) (a > 0): Gear 1 left-hand spiral, Gear 2 right-hand spiral. Negative offset (center distance) (a < 0): Gear 1 right-hand spiral, Gear 2 left-hand spiral

14.2.9 Number of teeth

You will find reference values for bevel gears with a shaft angle of 90 degrees in Table 14.1.

u	1	1.25	2	2.5	3	4	5	6
z_1	18..40	17..36	15..30	13..26	12..23	10..18	8..14	7..11

Table 14.1: Recommended pairing transmission ratio u - number of teeth, pinion z1 in accordance with Niemann [66]

14.2.10 Facewidth

The facewidth should not usually be larger than the one given in the recommendations (ratio to cone length, module ratio (see page II-482)). The contact pattern deteriorates if the facewidth is too great.

14.2.11 Profile shift coefficient

You will find reference values for the profile shift coefficient for bevel gears with a shaft angle of 90 degrees in Table 14.2.

U	1	1.12	1.25	1.6	2	2.5	3	4	5	6
x^*	0.00	0.10	0.19	0.27	0.33	0.38	0.40	0.43	0.44	0.45

Table 14.2: In accordance with Niemann, 24/4 [66], recommended pairings for transmission ratio u- profile shift coefficient x^*

Click on the button to the right of the profile shift coefficient input field to display the minimum profile shift coefficient for the pinion required to prevent undercut as well as the recommended value according to Niemann [66].

NOTE

The ISO23509 standard defines two different data types that can be used to describe tooth height factors and profile shift. The formulae used to convert data between these two data types are listed in ISO23509, chapter 7. The Gleason calculation sheets also give partial descriptions of factors K and C1. Although these are very similar to data type II, there are slight differences. Click the button to convert data type II data.

14.2.12 Tooth thickness modification factor

You will find reference values for bevel gears with a shaft angle of 90 degrees in Table 14.3.

u	1	1.12	1.25	1.6	2	2.5	3	4	5	6
x_s	0.00	0.010	0.018	0.024	0.030	0.039	0.048	0.065	0.082	0.100

Table 14.3: Recommended pairing transmission ratio u - tooth thickness modification factor xs in accordance with Niemann [66]

NOTE

If you are using standard cutters, such as those used for a Klingelnberg palloid or cyclo-palloid gear®, you must use the tooth thickness modification factors specified in the standard.

14.2.13 Quality

In this input field, you specify the accuracy grade in accordance with the standard shown in brackets. To change the standard used for this calculation, select `Calculation> Settings > General > Input of quality`. The accuracy grade defined in ISO17485 is very similar to that specified in DIN 3965.

You will find notes about the toothing quality in the Manufacturing process (see page II-468).

14.2.14 Addendum angle and root angle

All the necessary data required to create the bevel gear drawing can be calculated from the addendum angle and dedendum angle. These are the tip and active root diameter on the outside and inside bevel, and the tooth thickness on the outside and inside cone diameter (see Figure 14.6). The values shown here are output in the main report. In the case of bevel gears with spiral teeth, the addendum angle and dedendum angle are calculated using the selected method [ISO 23509, DIN3971]. In the case of bevel gear type 2 (Gleason), you can input the addendum angle. The root cone of the counter gear is then calculated from this value.

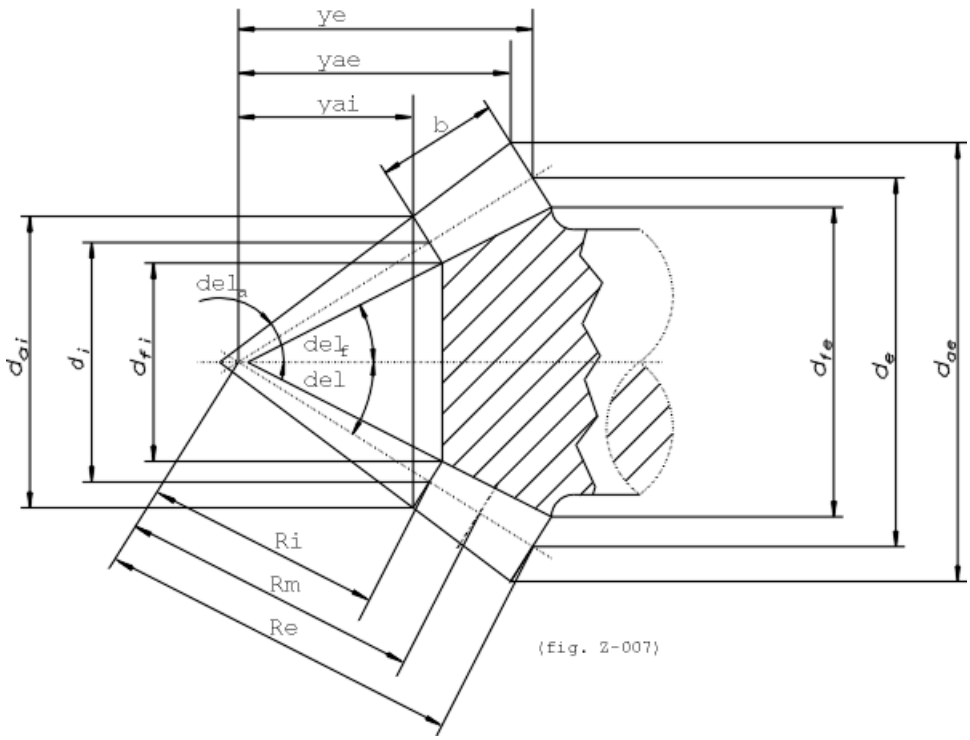


Figure 14.6: Dimensioning a bevel gear

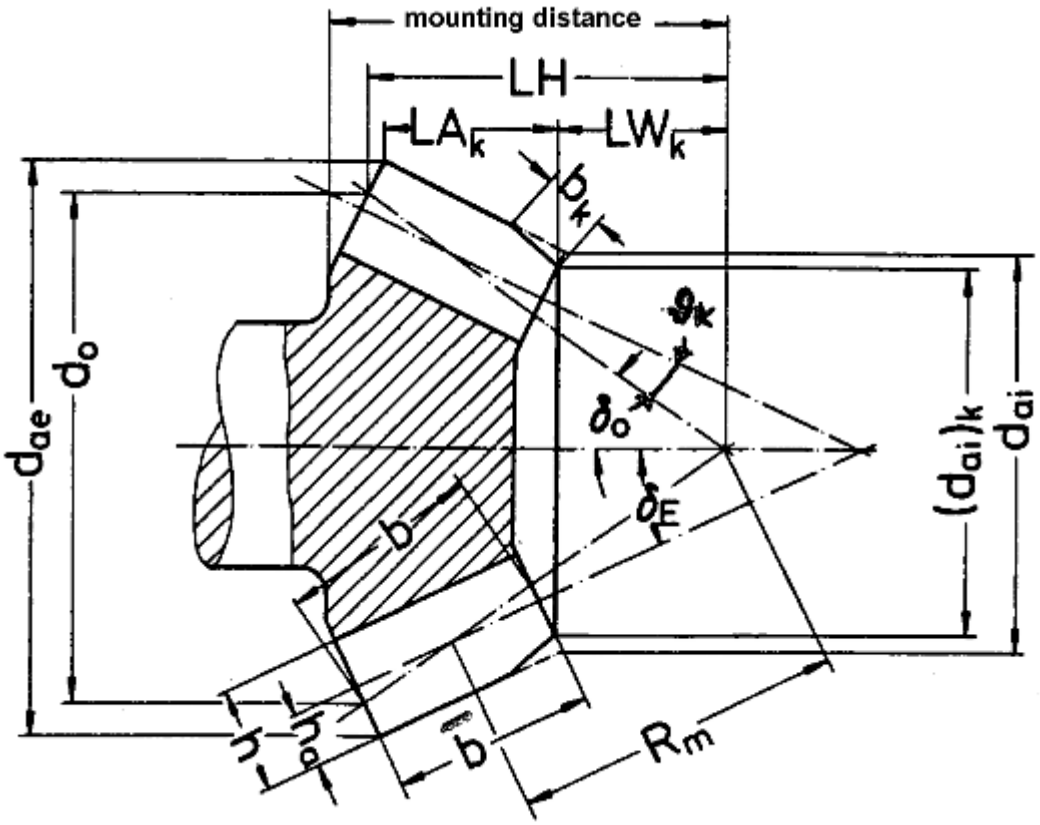


Figure 14.7: Dimensioning a bevel gear in accordance with Klingelnberg

14.2.15 Angle modifications

In some less than ideal situations it may happen that the cutter tip cuts into any shaft pins that are located immediately next to the toothing. If this cannot be prevented by modifying either the design or the toothing data, the cutter tip level at the calculation point at d_m of the gear and pinion can be tilted by a slight angle ϑ_c from its intended position $\delta_{o1,2}$ towards the reference cone angle $\delta_{E1,2}$. See Figure 14.7.

14.2.16 Geometry details

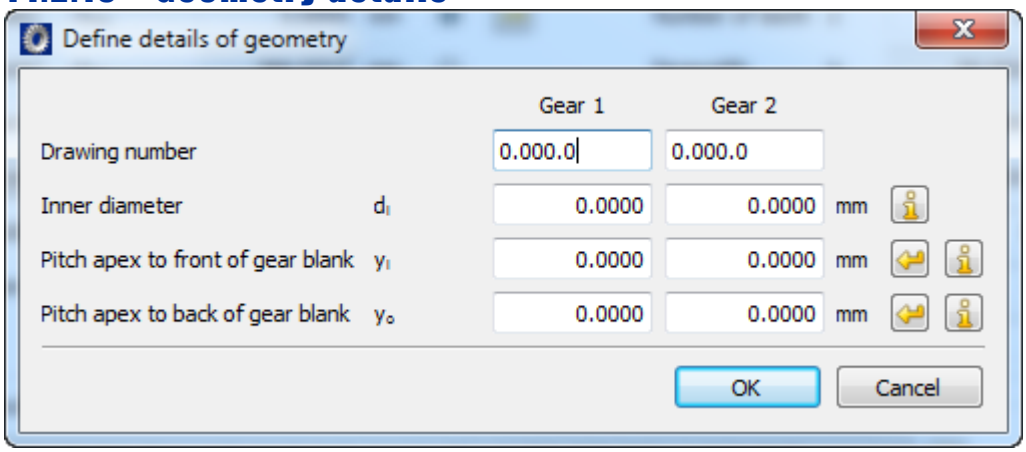


Figure 14.8: `Define details of geometry` dialog window

Click the `Details...` button in the upper right-hand part of the Geometry group to open the `Define details of geometry` dialog window. You can enter these parameters here.

The following parameters are not described here:

- Inside diameter (see page II-259)

14.2.16.1 Reference cone apexes on the outside/inside of the unworked part

The reference cone apex on the inside of the blank part is the distance in axial direction of the reference cone apex to the front face of the blank part.

The reference cone apex on the outside of the blank part is the distance in axial direction of the reference cone apex to the rear face of the blank part.

14.2.17 Manufacturing process

Tab. 14.10 shows the interrelationship between the manufacturing process and the achievable accuracy grade.

Process	**Achievable accuracy grade** **(ISO17485, DIN 3965)**
Milling only	8
Lapping	7
Skiving	6
Grinding	6

Table 14.10: Interrelationship between manufacturing process and achievable accuracy grade

14.3 Manufacture

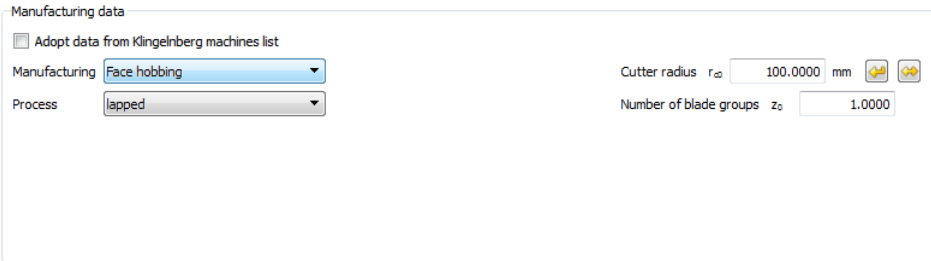


Figure 14.2: `Manufacture` input window

The process used to manufacture spiral teeth bevel gears is closely linked to this process. There are two basic processes used here. The arc of circle toothing process (traditionally known as the Gleason process) and the continuous face hobbing (traditionally referred to as the Klingelnberg and Oerlikon process). For more details see under Calculation process.

14.3.1 Cutter radius

In the case of spiral teeth bevel gears, the size of the cutter radius r_{c0} influences the beveling of the flanks and therefore also the properties of the pair of bevel gears. This effect applies both to the position of the contact pattern and the strength, and must be taken into account when calculating the transverse coefficient K_{Fa} in accordance with ISO 10300.

NOTE

This parameter is not present if you use the Klingelnberg method to calculate strength. In that case you select the cutter radius together with the machine type.

14.3.2 Number of blade groups, tool

The number of blade groups describes the number of cutter blade groups on the cutter head used to manufacture bevel gears with spiral teeth and, when face hobbing is in use, it, together with the cutter radius, influences the bevel of the tooth length. You must enter the number of blade groups as defined in ISO 23509, Annex E or as specified in the manufacturers' instructions.

14.4 Load

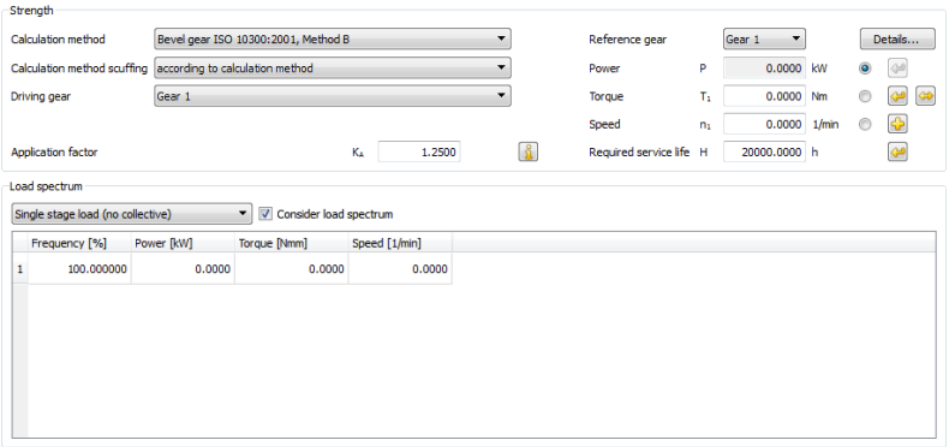


Figure 14.9: `Load` input window

14.4.1 Methods used for strength calculation

You can select the following methods:

1. **Bevel gears, only geometry calculation**
   Does not calculate strength. This method only calculates the geometric values, such as the path of contact.

2. **Bevel gears, static calculation**
   The strength calculation for cylindrical gears (see section "Calculation method" on page II-266) is implemented here.

3. **Differential gear, static calculation**
   The static calculation method is used for differential gears. The calculation is performed with the greatest circumferential force F_1 or F_2, see Figure 14.10

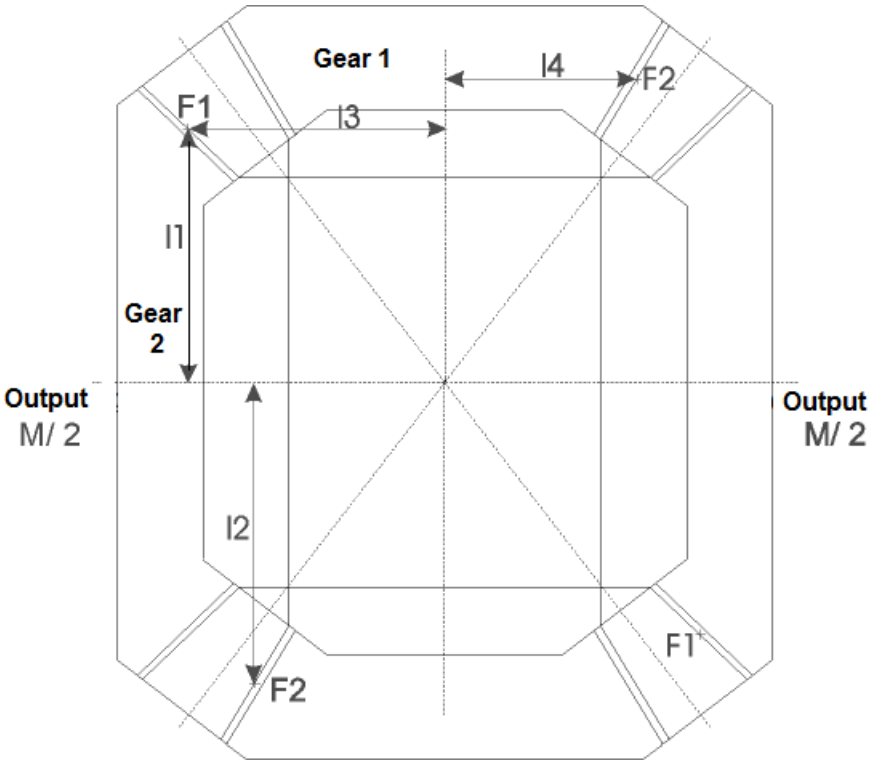


Figure 14.10: Bevel gears in differential gears

$$M = 2 \cdot F_1 \cdot l_1 + 2 \cdot F_2 \cdot l_2 \tag{13.1}$$

$$F_1 = F_2 \cdot \frac{l_4}{l_3} \tag{13.2}$$

$$l_1 = \sqrt{R_{m1}^2 - \left(\frac{d_{m1}}{2}\right)^2} - 0.5 \cdot \tan(\delta_1) \cdot (d_{am1} - d_{m1}) \tag{13.3}$$

$$l_2 = \frac{d_{am2}}{2} \tag{13.4}$$

$$l_3 = \frac{d_{am1}}{2} \tag{13.5}$$

$$l_4 = \sqrt{R_{m2}^2 - \left(\frac{d_{m2}}{2}\right)^2} - 0.5 \cdot \tan(\delta_2) \cdot (d_{am2} - d_{m2}) \tag{13.6}$$

4. **Bevel gears, ISO 10300, Method B (C)**
   ISO 10300, Parts 1, 2, 3: Load capacity calculation for bevel gears.
5. **Bevel gears as specified in ISO/CD 10300 (2011)**
   Draft version of the next edition of ISO 10300.
6. **Bevel gears as specified in AGMA 2003-B97 or AGMA 2003-C10**
   ANSI/AGMA 2003-B97 or AGMA 2003-C10: Rating the Pitting Resistance and Bending Strength of Generated Straight Bevel, Zerol Bevel and Spiral Bevel Gear Teeth
7. **Bevel gears**
   as specified in DIN 3991, Parts 1, 2, 3, 4: Load capacity calculation for bevel gears.
   This calculation is usually performed as defined in method B, and the tooth form factor is calculated with Method C.
8. **Bevel gears Klingelnberg KN 3028/KN 3030**
   This calculation is the same as the Klingelnberg in-house standards KN 3028 and KN 3030. These are mainly based on DIN standards. The calculation supplies the same results as the reference program used by Klingelnberg.

9. **Bevel gears Klingelnberg KN 3025/KN 3030**
   This calculation is the same as the Klingelnberg in-house standards KN 3025 and KN 3030. These are mainly based on DIN standards. The calculation supplies the same results as the reference program used by Klingelnberg.

10. **Bevel gears Plastic**
    This calculates the equivalent cylindrical gear pair (see also DIN 3991). Here the calculation is performed in accordance with Niemann/VDI/VDI-mod. in the same way as the cylindrical gear calculation (see page II-248).

11. **DNV41.2, Calculation standard for ships' engines**
    The Det Norske Veritas calculation guideline [93] for ships' engines corresponds in principle to ISO 10300 (root, flank) and ISO 13989 (scuffing). However, it does have some significant differences, especially where Woehler lines are concerned. These differences are detailed in our kisssoft-anl-076-DE-Application_of_DNV42_1.pdf information sheet, which is available on request.

12. **Hypoid bevel gears in accordance with ISO 10300**
    Hypoid bevel gears as specified in ISO 10300 with the suggested extension in accordance with FVA411. ISO 10300 (2001 edition) applies to bevel gears. The feasibility of extending the calculation method to include hypoid gears is under discussion. In the Federal Republic of Germany, an extension as part of the FVA411 research project has been proposed. This method has already been documented in the "Bevel gears" manual produced by Klingelnberg [87]. The method specified in FVA411 is only slightly different from the proposed ISO 10300 extension that has not yet been published.

13. **Hypoid bevel gears, geometry only**

14. **Hypoid bevel gears, in accordance with Klingelnberg KN3029/KN3030**
    This calculation is the same as the Klingelnberg in-house standards KN 3029 and KN 3030. These are mainly based on DIN standards. The calculation supplies the same results as the reference program used by Klingelnberg.

15. **Hypoid bevel gears, in accordance with Klingelnberg KN3026/KN3030**
    This calculation is the same as the Klingelnberg in-house standards KN 3026 and KN 3030. These are mainly based on DIN standards. The calculation supplies the same results as the reference program used by Klingelnberg.

NOTE

You will find more information about the strength calculation in accordance with Klingelnberg in section 14.5.

14.4.2 Required service life

You enter the required service life directly in this input field.

Click the button to size this value. Based upon the minimum safety value for the tooth root and flank strength, this process calculates the service life (in hours) for every gear and for every load you specify. The service life is calculated in accordance with ISO 6336-6 with the Palmgren-Miner rule. In the range of endurance limit you can select a modified form of the Woehler line as an alternative to ISO 6336. The system service life and the minimum service life of all the gears used in the configuration is displayed. You can size the service life using the button either with or without defining a load spectrum (see page II-287). You will find more detailed information about defining load spectra in section 13.19 (see page II-287).

14.4.3 Power, torque and speed

Click the button next to the power input field (for the torque) to calculate the power (torque) so that a predefined safety minimum (see page II-441) can be maintained. Click the button next to the power input field to apply a frequency distribution for power, torque and speed in the `Define load spectrum (see page II-287)` window. Click the button on the right of the Speed input field to open the `Define sense of rotation` window in which you can specify the direction in which the bevel gear rotates in accordance with Figure 15.6 on page II-503.

14.4.4 Strength details

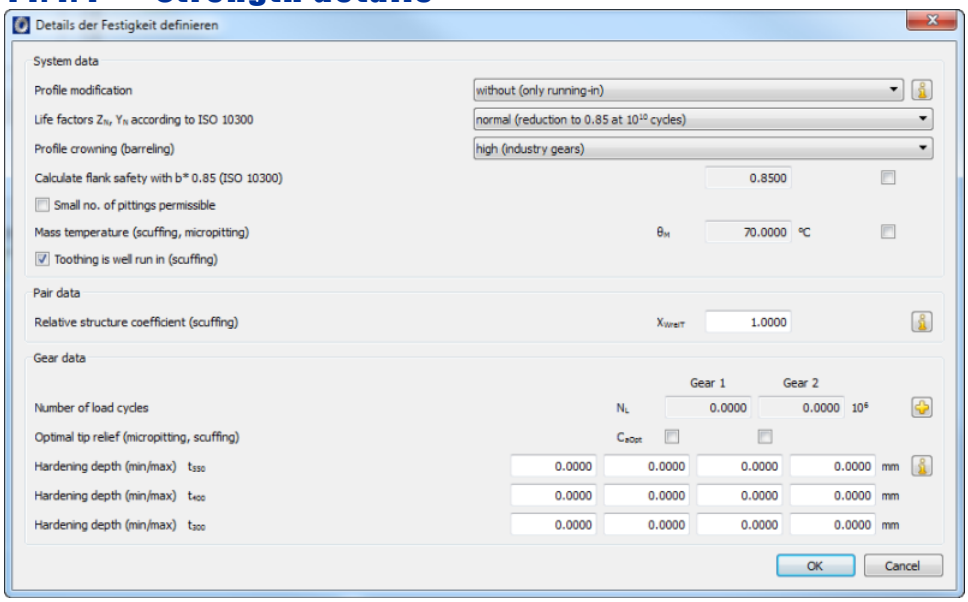


Figure 14.11: `Define details of strength` dialog window

Click the `Details...` button in the upper right-hand part of the `Strength` group to open the `Define details of strength` dialog window.

The parameters described in other places are:

- Fatigue strength (see page II-274)
- The input of the type of `profile crowning (barreling)`: influences the calculation of the contact area (only for ISO 10300) and the load distribution coefficient Z_{LS}. The 2001 edition of ISO 10300 does not yet use this variant.
- Small pitting (see page II-279)
- Relative structure coefficient (see page II-279)
- Number of load cycles (see page II-280)
- Alternating bending factor (see page II-293)

14.4.4.1 Profile modification

Modifying the profile of bevel gears is unusual. Please contact the manufacturer first to see whether it is feasible to do so. The run-in amount specified in ISO 10300 is the most commonly used.

14.4.4.2 Calculate flank safety with 0.85*b (ISO 10300)

Flank safety as defined in ISO 10300 is calculated with the length of the contact line up to the tooth depth middle l_{bm}. Select this checkbox to perform this calculation with a modified width instead of using ISO 10300

$$b' = (faktor) \cdot b$$

.

The usual contact pattern width is 0.85*facewidth (for example, as specified by DIN 3991.) If you have sufficient experience, you can modify this value.

NOTE

You can only input this value if you are using the ISO10300 calculation method.

14.4.5 Application factor

The application factor compensates for any uncertainties in loads and impacts, whereby $K_A \geq 1.0$. Table 14.4 illustrates the values that can be used for this factor. You will find more detailed comments in ISO 10300, ISO 6336, DIN 3990 and DIN 3991.

Operational behavior of the driving machine	**Operational behavior of the driven machine**			
	uniform	**moderate shocks**	**medium shocks**	**heavy shocks**
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 14.4: Assignment of operational behavior to → application factor

14.5 Factors

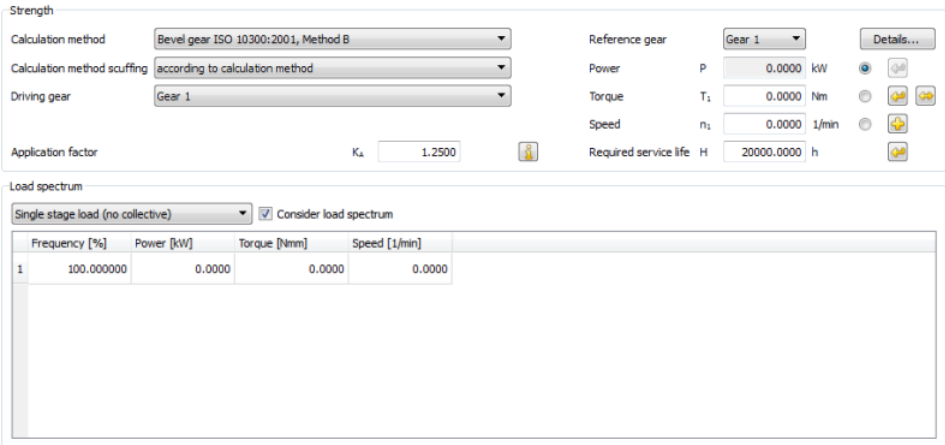


Figure 14.10: `Load` input window

14.5.1 Bearing application factor

Tables 14.5 ÷14.7 show the bearing type → bearing application factor for different standards.

Support for pinion and ring gear	**Bearing application factor**		
	a	**b**	**c**
both on both sides	1.00	1.05	1.20
one on both sides, one floating	1.00	1.10	1.32
both floating	1.00	1.25	1.50

a	:	Contact pattern in the gearbox tested under full load
b	:	Contact pattern in the gearbox tested under part load
c	:	Contact pattern only tested in specific tests

Table 14.5: Bearing application factor in accordance with ISO 10300

Support for pinion and ring gear	**Bearing application factor**

both on both sides	1.10
one on both sides, one floating	1.25
both floating	1.50

Table 14.6: Bearing application factor in accordance with DIN 3991

Support for pinion and ring gear	**Bearing application factor**
both on both sides	1.10
one on both sides, one floating	1.10
both floating	1.25

Table 14.7: Bearing application factor in accordance with AGMA 2003

The face load factors $K_{H\beta}$,$K_{F\beta}$ and $K_{B\beta}$ are calculated as follows from the bearing application factor $K_{H\beta be}$ as defined in the standard:

$$K_{H\beta} = K_{F\beta} = K_{B\beta} = 1.5 \cdot K_{H\beta be} \tag{14.7}$$

14.5.2 Dynamic factor

To calculate the dynamic factor K_v, as defined by Klingelnberg, use the coefficient K_1 either for preliminary calculations based on the planned manufacturing method (lapped, HPG) or on the basis of the derived accuracy grade (see also Klingelnberg standard KN 3030, Table 5.2-1 or 5.2-2).

14.5.3 Bevel gear factor at flank and root

To calculate the strength of bevel gears, you use the virtual cylindrical gear with equations that apply to strength calculation for cylindrical gears. The bevel gear factors are then used to correct the systematic differences in the calculation between cylindrical gears and bevel gears. These factors are defined in the corresponding standards.

Standard	**Bevel gear factor of flank Z_K**

ISO 10300	0.80
Niemann	0.85

Table 14.8: Bevel gear factor of flank Z_K as defined in the standard

Standard	**Bevel gear factor of root Y_K**
ISO 10300	is calculated, see Part 3 of the standard
Niemann	1.00

Table 14.9: Bevel gear factor of root Y_K as defined in the standard

14.6 Reference profile

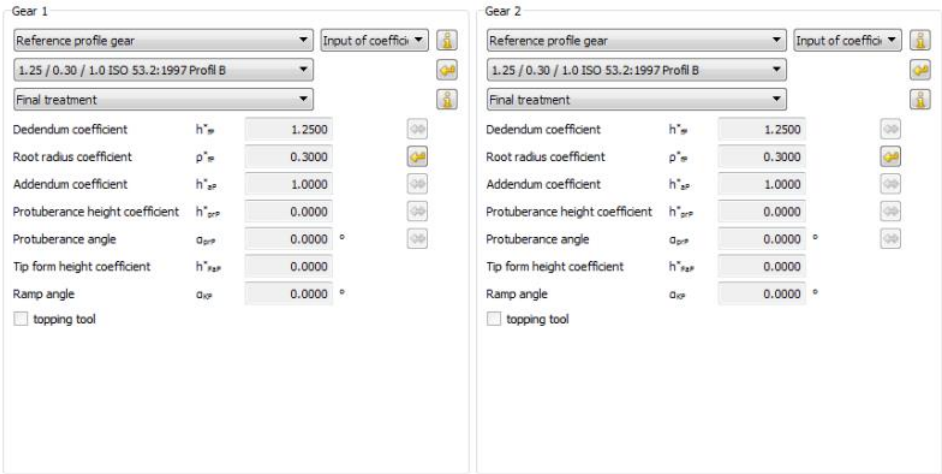


Figure 14.12: Reference profile tab

14.6.1 Default values for tip base clearance

The tip base clearance for spiral bevel gears is usually 0.2 to 0.3 times the average normal module. However, a greater amount of play is used for toothings that are manufactured with tilt. This prevents the tooth tip intersecting with the root of the opposing gear.

Default values are (as stated in the "Kegelräder" book produced by Klingelnberg [87]):

"Gleason, modified slot width" procedure:	0.3
"Gleason, constant slot width" procedure:	0.35
"Klingelnberg, Palloid" procedure:	0.3
"Klingelnberg, Cyclo-Palloid" procedure:	0.25
"Oerlikon" procedure:	0.25

14.6.2 Default values for addendum coefficients

The addendum coefficient is usually 1.0.

14.7 Rough sizing

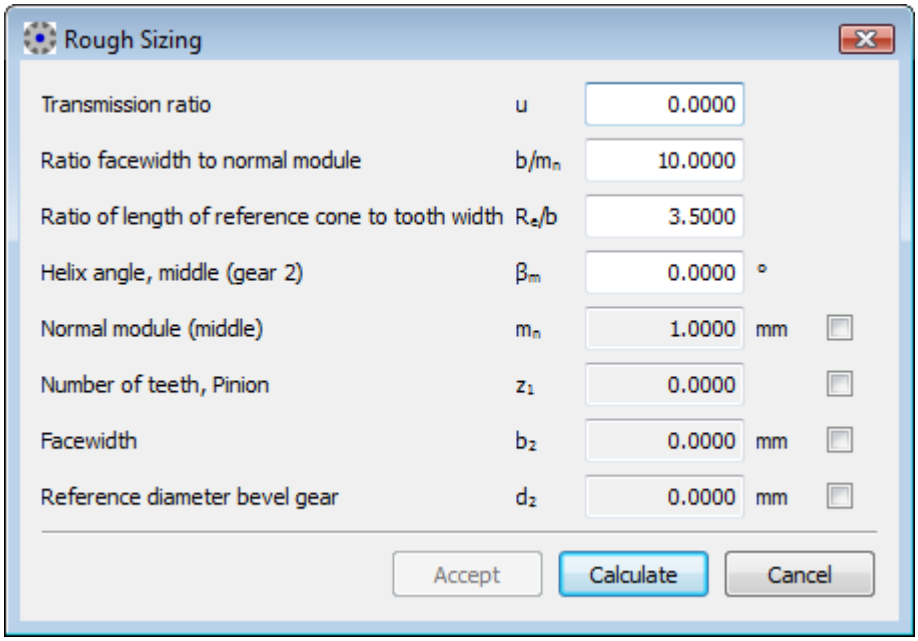


Figure 14.13: `Rough sizing` dialog window

The method is developed by Klingelnberg, in accordance with the suggestions from technical literature [Kegelräder, Hrsg. Klingelnberg] to size bevel and hypoid gears provides geometrically satisfying recommendation of gear pairs. This proposal gives sufficiently precise solutions to the problems of achieving the required safeties against tooth fracture and pitting because it is based on values gathered through years of experience. If you verify gear teeth that have been dimensioned in accordance with this method, you may discover certain deviations from the required safety values.

However, you can easily achieve these safety levels by simply changing the module and the facewidth.

14.7.1 Face width ratio

Depending on how and where a gearbox is to be used, the facewidth should be in a specific ratio to the cone distance and correspond to the following values:

Light and medium-heavy load gearboxes for machines and vehicles	$3.5 \leq (Re/b) \leq 5.0$
Heavy load gearboxes for machines and vehicles	$3.0 \leq (Re/b) \leq 3.5$

14.7.2 Module ratio

The normal module mn should be in a ratio to the facewidth b within specific limits which can only be exceeded (or not reached) for exceptional reasons:

surface hardened bevel gears at risk of tooth fracture	$7 \leq (b/mn) \leq 12$
At risk of pitting or through hardened or not hardened	$10 \leq (b/mn) \leq 14$

14.8 Notes on calculations in accordance with the Klingelnberg standard

14.8.1 Bevel gears with cyclo-palloid® gear teeth

Geometry, practicability of manufacturing and strength calculation of bevel gears in accordance with the Klingelnberg cyclo-palloid® process.

As stated in the Klingelnberg KN 3028 in-house standard (geometry and manufacture) and KN 3030 (strength calculation) a complete calculation is performed for cyclo-palloid® toothing:

- Calculate machine distance for machine types FK41B, AMK400, AMK635, AMK855, AMK1602 with all corresponding cutters, cutter radii and numbers of starts. A warning is displayed if you select an incorrect machine type or cutter tip.
- You can specify any shaft angle, or angle modification here.
- Overall geometry, modules (inside, middle, outside), spiral angle (inside, outside), checks on cut back, undercut space, calculation of profile shift for balanced sliding, checks on backwards cut, checking and calculating the necessary tip reduction on the inside diameter, profile and overlap ratio, tooth form factor and stress correction factor.
- Calculation of all toothing dimensions.
- Calculation of pitting, tooth root and resistance to scoring (as defined by the integral temperature criterion) with all modifications in the in-house standard KN 3030.

14.8.2 Hypoid gears with cyclo-palloid gear teeth

Geometry, manufacturability and strength calculation of hypoid gears (bevel gears with offset(center distance)) as defined in the Klingelnberg process.

As stated in the Klingelnberg KN 3029 in-house standard (geometry and manufacture) and KN 3030 (strength calculation) a complete calculation is performed for cyclo-palloid toothing:

- Calculation of machine distance for machine types FK41B, KNC40, KNC60, AMK855, AMK1602 with all corresponding cutter head, cutter radii and numbers of blade groups. A warning is displayed if you select an incorrect machine type or cutter tip.

- You can use any value as the shaft angle, angle modification, pressure angle for the driving and driven flank.
- Overall geometry with calculation of the facewidths, modules (inside, middle, outside), spiral angle (inside, outside), undercut boundary, calculation of gap widths, checks on backwards cut, checking and calculating the necessary tip reduction on the inside diameter, profile and overlap ratio, tooth form factor and stress correction factor either for the tension (driving) or coast (driven) flank.
- Calculation of all toothing dimensions.
- Calculation of pitting, tooth root and resistance to scoring (as defined by the integral temperature criterion for the replacement spiral-toothed gear wheel) with all modifications in the in-house standard KN 3030.

14.8.3 Normal module ranges for Klingelnberg machines (cyclo-palloid)

Machine	**Cutter radius** r_{∞}	**Normal module** m_{mn}	
FK41B	25	0.25 to	1.6
	30	0.25 to	1.6
	40	0.25 to	1.6
AMK400	55	1.1 to	4.0
	100	2.4 to	5.2
	135	3.5 to	8.0
	170	3.5 to	13.0
AMK635	55	1.1 to	4.0
	100	2.4 to	5.5
	135	3.5 to	8.0
	170	6.5 to	13.0
	210	7.0 to	13.0
AMK855	135	3.5 to	8.0
	170	6.5 to	13.0
	210	7.0 to	15.5
	260	7.0 to	15.5
AMK1602	270	8.0 to	17
	350	14.0 to	25.0

	450	17.0 to	34.0
KNC25	30	0.5 to	5.5
	55	0.5 to	5.5
	75	0.5 to	5.5
	100	0.5 to	5.5
KNC40	30	1.0 to	1.6
	55	1.1 to	4.0
	75	2.0 to	4.5
	100	2.4 to	5.5
	135	3.5 to	8.0
KNC60	75	2.0 to	4.5
	100	2.4 to	5.5
	135	3.5 to	8.0
	170	6.5 to	14.0

Table 14.11: Normal module ranges for Klingelnberg machines

14.8.4 Bevel gears with Palloid toothing

Calculate the geometry and strength of bevel gears using the Klingelnberg procedure.

A complete calculation for palloid toothing is performed in accordance with the Klingelnberg KN3025 in-house standard (Geometry, Edition No. 10) and KN3030 (strength calculation).

- Taking into account Palloid cutter dimensions by including a smaller diameter dK and cutter cut length SF, you can also input special cutters here
- A warning is issued if the cutters do not cover the crown gear at either the inner or outer end of the tooth
- You can select any shaft angle, or angle modifications
- Overall geometry, modules (inside, middle, outside), spiral angle (inside, middle, outside), checks on profile shift for balanced sliding and undercut space, checking and calculating the necessary tip reduction on the inside

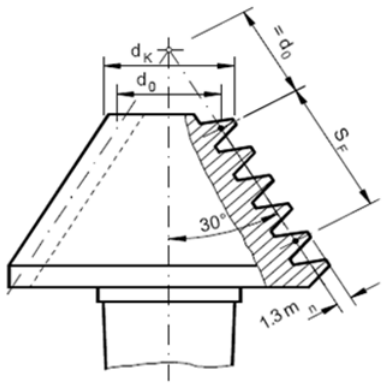
diameter, profile and overlap ratio, tooth form factor and stress correction factor

- Calculation of all toothing dimensions
- Calculate forces for contact pattern position for cone distances length Rpr and Rm
- Calculate pitting, tooth root and resistance to scoring (as defined by the integral temperature criterion for all modifications in the Klingelnberg standard KN 3030 (taking into account the forces of cone distance Rpr)

NOTE

The forces at bevel length Rm are used for the transfer to KISSsys, to ensure that forces can be calculated independently of the toothing procedure. However, including the theoretical contact pattern core in the Klingelnberg in-house standard is very difficult to implement in the manufacturing process.

14.8.5 Definitions and dimensions of standard cutters for palloid toothing



$$d_0 = d_k - 2 \cdot 1.3 \cdot m_n \cdot \cos 30°$$

	Row A		Row B		Row C	
m_n	d_k	S_F	d_k	S_F	d_k	S_F
1	14	13	18	16	18	19
1.25	17	16	21	20	21	24
1.50	20	19	24	24	24	29
1.75	23	22	27	28	27	33
2	26	26	30	32	30	38
2.25	29	29	32	36	32	43
2.50	32	32	35	40	35	48
2.75	34	35	37	44	37	52
3	36	38	39	48	39	57
3.25	38	41	42	52	42	62
3.50	40	44	44	55	44	66
3.75	42	48	46	59	46	71
4	44	51	49	63	49	76
4.25	46	54	51	67	51	81
4.50	48	57	53	71	53	85
5	52	63	58	79	58	95
5.50	56	70	63	87	63	104
6	60	76	67	95	67	114
6.50	64	82	72	103	72	123
7	68	88	77	110	77	132
7.50			81	118		
8			86	126		

Figure 14.14: Dimensions of standard cutters

14.8.6 Minimum safeties

We recommend you use the following minimum safeties:

Application	Minimum safeties	
Flank	1.1 to 1.2	
Root	1.5 to 1.6	
Scuffing	1.8 to 2.0	

Table 14.12: Recommended minimum safeties

14.8.7 Surface roughness at tooth root

Treatment	Roughness [mm]
through hardened	0.016
lapped	0.016
with hardened toothing	0.008

Table 14.13: Surface roughness values

14.8.8 Accuracy grade bevel gears

Treatment	Quality number
through hardened	7
lapped	7
with hardened toothing	6

Table 14.14: Accuracy grade for bevel gears

14.8.9 Characteristic number

The product of the lubrication, speed and roughness factor $Z_L Z_V Z_R$ for different surface treatments is shown in Table 14.15:

Treatment	Characteristic number $Z_L Z_V Z_R$
through hardened	0.85

lapped	0.92
with hardened toothing	1.0

Table 14.15: Characteristic number $Z_L Z_V Z_R$ for different surface treatments

NOTE

You will find a similar definition in ISO 10300-2:2001, Section 14.4. Here the characteristic number is also dependent on the defined level of roughness R_z.

14.8.9.1 Single normal pitch deviation

This is calculated in accordance with DIN 3965.

14.8.9.2 Contact spring stiffness

The contact stiffness is assumed to be constant.

$$c_g = 20 \frac{N}{mm \cdot \mu m}$$

14.9 Settings

In the `Calculation` menu you will find the `Settings` option. Click this sub menu to open the `Module specific settings` window. From here you can access the tabs listed below to input other calculation parameters. (the following parameters not described here (see page II-428))

14.9.1 Calculations

14.9.1.1 Coefficient of friction for hypoid gears

Due to longitudinal sliding, hypoid gears have more power loss than spiral bevel gears. For this reason, the calculation of toothing forces in KN3030 takes the coefficient of friction account. If necessary, you can enter the size of the coefficient of friction in the Module-specific settings.

Chapter 15
Face gears

Face gears are a special type of bevel gears. Although the pinion is a normal cylindrical gear, a face gear has a complex 3D-tooth form. Unlike a bevel gear, a face gear is absolutely not affected by axial displacement. For this reason, face gears are much easier to assemble.

The KISSsoft `Face gears` calculation module calculates the geometry of pairs of straight or helical cylindrical gear pinions with face gears with offset (center distance) and with any shaft angle Σ. In this case, the strength and 2D geometry are calculated for an offset (center distance) of 0 mm and a shaft angle Σ=90°. In every other case, you can perform the presizing with these restrictions and then add the required offset (center distance) and shaft angle to the 3D volume model. In the `Geometry` docking window you can display the tooth form of a face gear for its inside, middle and outside diameter or for any number of sections all at the same time. You use this tool to check for undercut and pointed teeth on the inside or external diameter of the face gear. In the `Modifications` input window (tab), you will find the value/length of the tip alteration at outside (inside) hake(i), lake(i) input fields which contain additional parameters that will help you prevent pointed teeth occurring in the gear. The tooth form on a face gear is calculated by simulating manufacture with a pinion type cutter. The strength calculation is based on the use of established standards for cylindrical or bevel gears.

15.1 Underlying principles of calculation

A face gear has features in common with a curved rack. However, unlike this simplest of all gears, when assembling and installing a face gear, engineers are always confronted with the restrictions posed by that very bending. As the tooth flank in a spur geared face gear must run parallel to one radius of the face gear - the contacting pinion has flanks parallel to its own axis - the immediate result of the theorem of intersecting lines is that the pressure angle must reduce from outside to inside. The equation shown here can be regarded as the main formula used to size the geometry of face gears. To keep this as simple as possible, only a gear with spur-toothed gear teeth is considered here. Equation [3]

$$d_2 = \frac{m_n \cdot z_2 \cdot \cos\alpha_n}{\cos\alpha_2} \tag{15.1}$$

contains

d_2	diameter of face gear
m_n	normal module pinion
z_2	number of teeth on face gear
α_n	pinion pressure angle on the reference circle
α_2	pressure angle on face gear for diameter d_2

From this, you can, for example, define the pressure angle from the external diameter to the inside diameter. If the inside tooth flanks are steep, the involute will be short and only bear a small part of the tooth depth. The risk of an undercut grow in the direction of the crown gear center. Any undercut here would further reduce the usable area. The result is a minimum inside diameter and a maximum external diameter, which limit the total face width of the face gear. This is where a face gear differs fundamentally from a bevel gear: whereas you can increase the facewidth on a bevel gear to enable it to transmit higher torques, strict limits are set here for face gear to cylindrical gear-pinion pairs. However, if you select the right axial offset b_v, i.e. by moving the facewidth middle compared to the reference circle, you can optimize the maximum permitted facewidth.

When assembling a face gear it is a good idea to define a minimum and a maximum pressure angle and then the achievable inside and external diameter. If external conditions limit this diameter (this usually affects the external diameter), you can use the conversion in equation (15.1) to change the range available for the module.

$$m_{\min/\max} = \frac{d_{2,\min/\max} \cdot \cos(\alpha_{2,\min/\max})}{z_2 \cdot \cos(\alpha_n)} \tag{15.2}$$

In addition to having the figures to hand, you may find it helpful to view the teeth as a graphic in this situation.

The vast majority of applications use face gears with spur gears. However face gears with helical teeth, when arranged correctly, do offer a number of benefits such as noise reduction and strength. Unfortunately, these benefits are offset by the problem that the tooth flanks are not symmetrical, i.e. the left flank no longer corresponds to the right flank. In practice this means that any undercut that occurs will happen earlier on one flank than on the other. This differences in the flanks also have a significant influence on strength, which results in a difference between the directions of rotation when the gear transmits power. However, if only one sense of rotation is used, such as for electrical tools, you can optimize the flank involved without having to take the effect on the rear flank into account.

Experience has shown that theoretical observations of geometry to decide which involute functions, lines, and arcs of circles, to use, to describe a tooth form, will sooner or later reach their limit. A tried and tested, and much more reliable, means of calculating tooth forms is to simulate the generation process or, even better simulate the manufacturing process. To do this, the trajectory of a point on the active surface of the tool is followed until its speed relative to the tool surface normally reaches a zero crossing (see Figure 15.1).

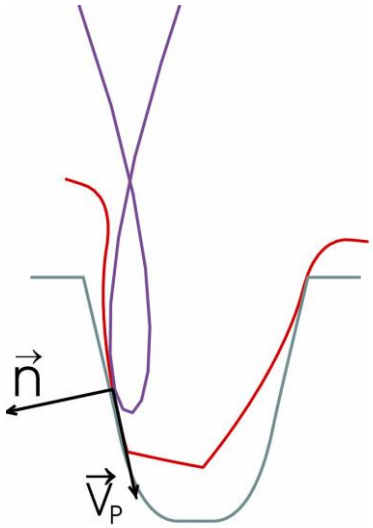


Figure 15.1: Spur curve (blue) of the pinion type cutter tool (red) on the face gear (green)

These points are potential points on the tooth form surface. You must then separate the actual points on the surface from the imaginary points at which the nominal speed also disappears but the underlying points are shown as being outside the material. How to separate the real from the imaginary points is one of the most difficult aspects of the process described here. In addition to referring to the usual standard algorithms for classifying points in a level, you must also use empirical approaches that use the known properties of the tooth form in order be sure of achieving a well-defined tooth form with sufficient safety. You can therefore match the data derived from calculating a 3D tooth form of a face gear with the data derived from generating using a pinion type cutter using a classic manufacturing method. By outputting the 3D body in IGES, STEP or SAT format you can then design the form in any CAD system. The face gears can then be manufactured in either an injection molding, sintered or precision forging process. However 2D cross section view is much more suitable if you want to check a face gear for undercut or pointed tooth tips. This displays the inside, middle and outside of the face gear tooth form all at the same time. If you then rotate the gears step by step, you can check every aspect of the gear generation very precisely. If a tooth is pointed, or if the meshing ratios are not good enough, you must reduce the tooth depth in the same way as you do for hypoid gears. To reduce the gear's sensitivity to errors in the axis alignment or the center distance, you can allow crowning on the tooth flank (tooth trace). You can generate this quite easily for face gears by using a pinion type cutter that has one or more teeth more than the pinion in the manufacturing process [79]. When you compare the tooth forms you can see the effect the increased number of teeth on the pinion type cutter had on the generated tooth form. However, if the face gear has a large axial offset b_v, you can move the barreling to one side! In every axis section through the cylindrical gear, the face gear drive corresponds to a pinion-rack gear drive. Using the rack theory as a basis, you can therefore define the pressure angle, the lines of contact and the contact ratio in each section.

The examples in this section are based on a publication in [50].

15.2 Basic data

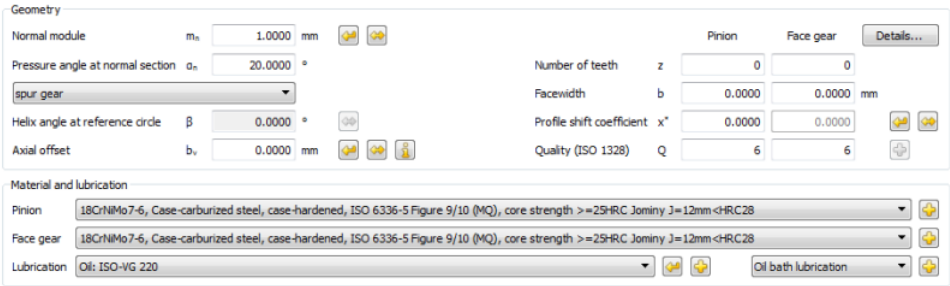


Figure 15.2: `Basic data` input window in the `Face Gears` module

15.2.1 Normal module

Enter the normal module. However, if you know the pitch, transverse module, or diametral pitch, instead of this, click on the button to open a dialog window in which you can perform the conversion. If you want to transfer the diametral pitch instead of the normal module, you can select `Input normal diametral pitch instead of normal module` by selecting `Calculation > Settings > General`.

If you have already defined all aspects of the geometry of a face gear, the following message appears after you click the button:

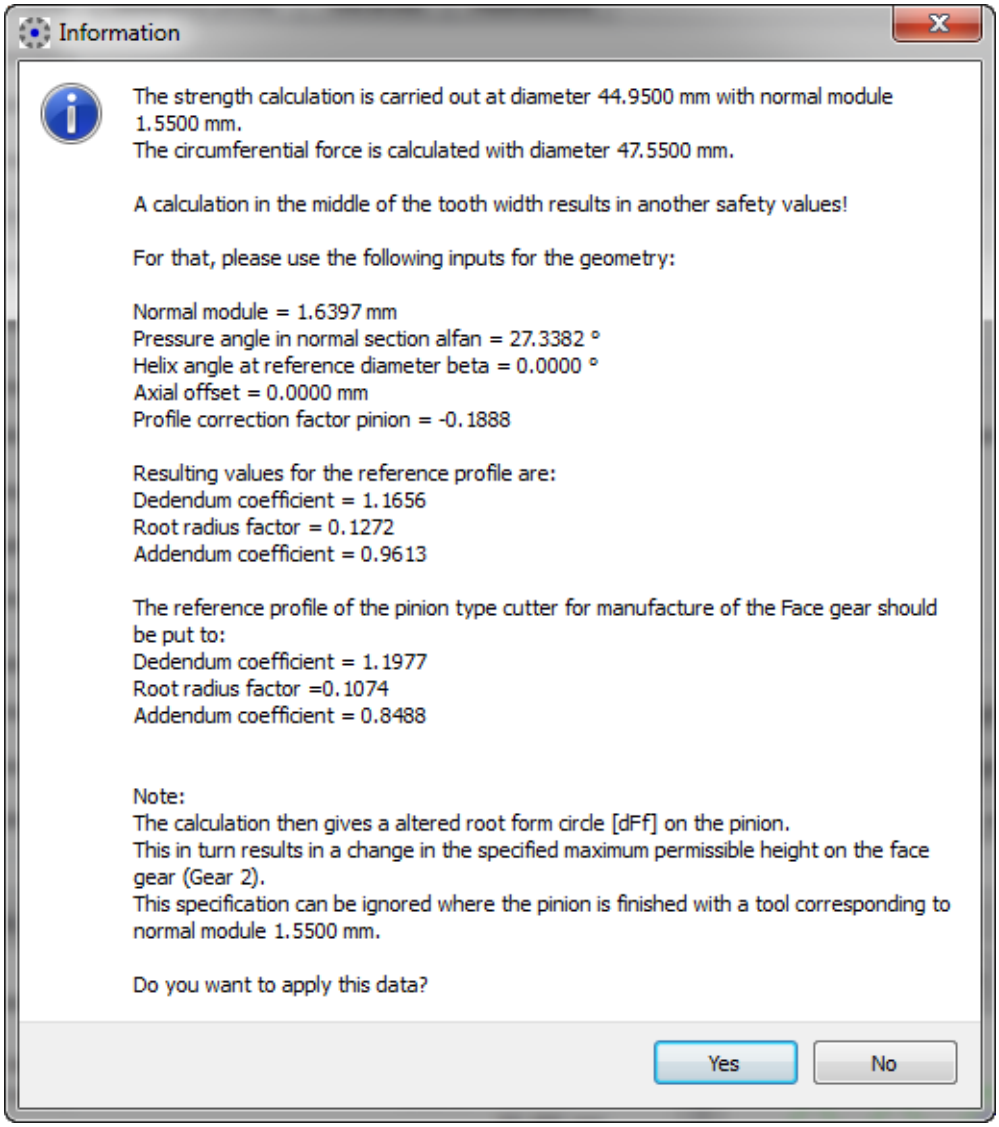


Figure 15.3: Information window for sizing the normal module

As part of the bevel gear calculation performed in accordance with ISO 10300 or DIN 3991, the strength calculation is performed for the middle diameter of the face gear. If the axial offset b_v <> 0, the conditions for this type of calculation have not been met. For this reason the button supports the conversion of normal module m_n and pressure angle α_n, to ensure that $b_v = 0$. Although this changes the root radius of the pinion, the shape of flank remains the same.

NOTE

We recommend you only use this conversion method when you perform the strength calculation. The conversion changes the module and you can no longer use the tool. This is why you must save your geometry data before you perform the conversion.

15.2.2 Pressure angle at normal section

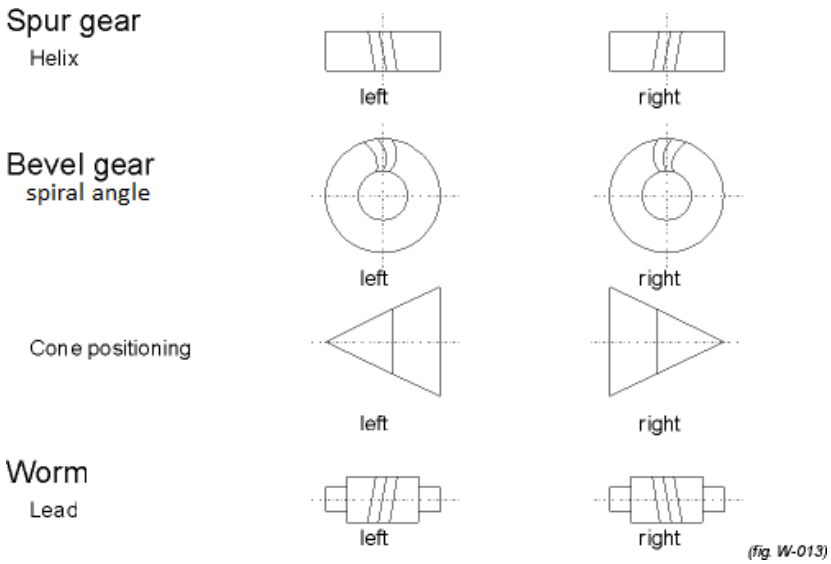
The normal pressure angle at the reference circle is also the flank angle of the reference profile. For standard toothings the pressure angle is $\alpha_n = 20^o$. You can use smaller pressure angles for a larger number of teeth to achieve higher contact ratios. Greater pressure angles increase the strength and allow a smaller number of teeth to be used without undercut. In this situation, the contact ratio decreases and the radial forces increase.

NOTE

The operating pressure angle α_{wt} changes across the width of the toothing.

15.2.3 Helix angle at reference circle

Enter the helix angle in [^{o}]. You can either convert this from the helix angle on the base circle β_b or from the helix angle at tip circle β_a by clicking the button in the `Convert helix angle` window. Helical gear teeth usually generate less noise than spur-toothed gear teeth. However, they also have the disadvantage that they involve additional axial force components.



(fig. W-013)

Figure 15.4: Helix angle

15.2.4 Axial offset

The axial offset is the distance of the pinion center from the middle diameter of the face gear.

Click the button to the right of the `Axial offset` input field to calculate the largest possible width of the face gear (see page II-508) b_2 and the corresponding axial offset b_v, so that the pressure angle lies within the predefined limits.

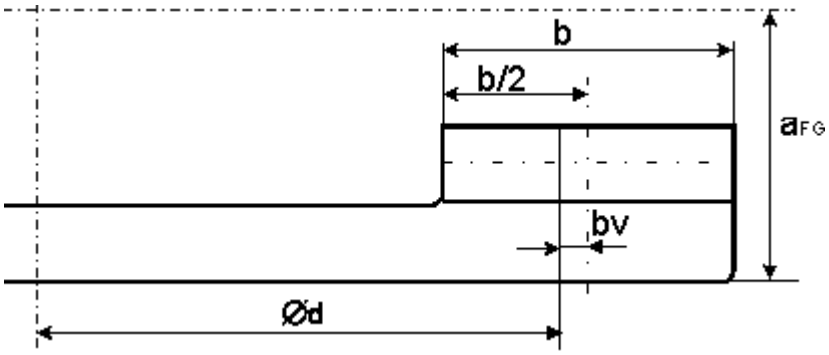


Figure 15.5: Axial offset of the face gear

15.2.5 Profile shift coefficient (on the pinion)

The tool can be adjusted during manufacture. The distance between the production pitch circle and the tool reference line is called the profile shift. To create a positive profile shift, the tool is pulled further out of the material, creating a tooth that is thicker at the root and narrower at the tip. To create a negative profile shift the tool is pushed further into the material, with the result that the tooth thickness is smaller and there is more danger of undercut. In addition to the effect on tooth thickness, the sliding velocities will also be affected by the profile shift coefficient.

You can modify the profile shift according to different criteria. To do this, use the various sizing options in the `Sizing of profile shift` window. Here, click the button:

- **For undercut boundary**
- **For minimum topland per gear.**
  You can specify the minimum thickness of the topland in `Calculation >`

`Settings > General > Coefficient for minimum tooth thickness at the tip`.

NOTE

The pinion should have a reasonable high value for the tooth thickness at the tip because the pinion type cutter used to manufacture a face gear has a somewhat higher tip and, despite that, must not be permitted to become pointed.

Click the button and KISSsoft will determine whether the profile shift coefficient (see page II-253) is to be taken from measured data or from values given in drawings.

15.2.6 Quality

In this input field you specify the accuracy grade in accordance with the standard shown in brackets. To change the standard used for this calculation, select `Calculation > Settings > General > Input of quality`. The accuracy grade in accordance with ISO 1328 is very similar to that in DIN 3961 or AGMA 2015.

Achievable qualities are shown in Table 15.6.

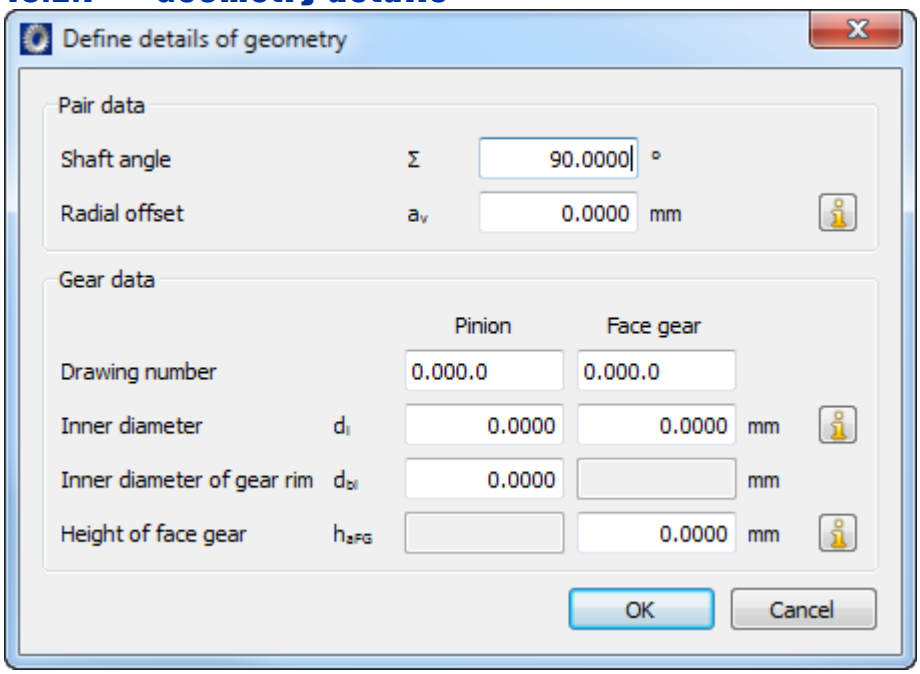
Manufacturing process	**Quality in accordance with DIN/ISO**		
Grinding	2	. . .	7
Shaving	5	. . .	7
Hobbing	(5)6	. . .	9
Milling	(5)6	. . .	9
Shaping	(5)6	. . .	9
Punching, Sintering	8	. . .	12

Table 15.6: Quality values for different manufacturing processes

NOTE

The values in brackets can only achieved in exceptional situations.

15.2.7 Geometry details



Click the `Details...` button in the upper right-hand part of the Geometry group to open the `Define details of geometry` dialog window. You can enter these parameters here.

15.2.7.1 Shaft angle

You can select your own shaft angle here. However, to perform a strength calculation you should set it to Σ = 90.

15.2.7.2 Inside diameter

The inside diameter is needed to calculate the mass moment of inertia. As defined in ISO or AGMA, the gear rim thickness does affect the strength. For complete gears, enter 0, for external gears with a gear teeth, enter the appropriate diameter d_i as shown in Figure 15.7.

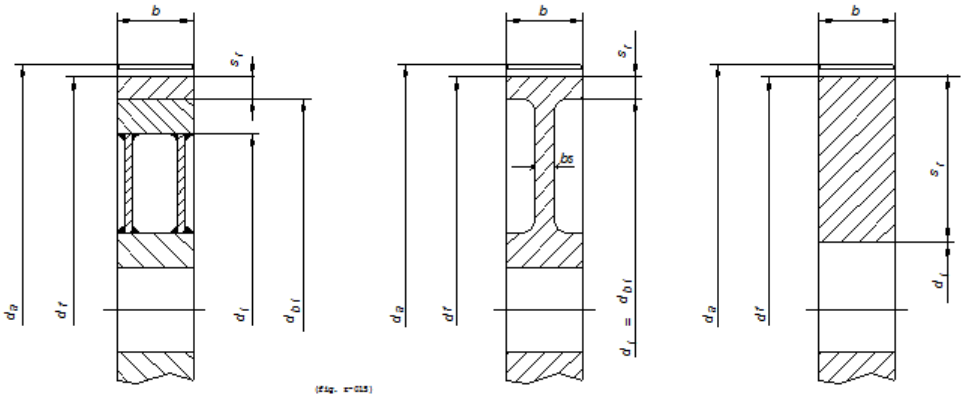


Figure 15.7: Dimesnsioning the diameter

The inside gear rim diameter is required for calculations in accordance with ISO or AGMA. Where thin gear rims are used, this factor can greatly influence the calculation results. as illustrated by → Figure on page II-537.

15.2.7.3 Height of face gear

For information on defining the height of face gear h_{aFG} →see Figure (see page II-506)

15.2.8 Material and lubrication

The materials displayed in the drop-down lists are taken from the materials database. If you cannot find the material you require in this list, you can either select `Own Input` from the list or enter the material in the database (see section "External tables" on page I-122) first. Click the button to open the `Material pinion(face gear)` window in which you can select a list of materials that are available in the database. Select the `Own Input` option to enter specific material characteristics. This option corresponds to the `Create a new entry`window in the database tool.

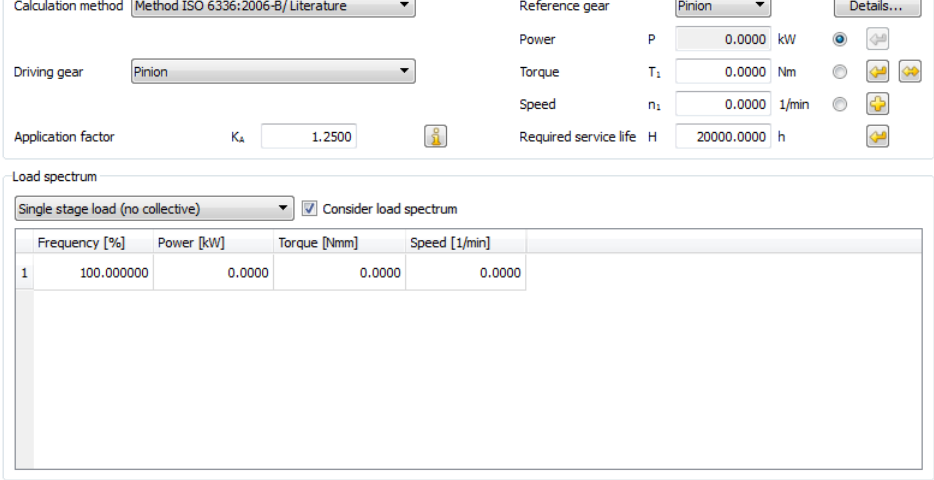


Figure 15.3: `Load` input window in the `Face gears` module

15.3.1 Methods used for strength calculation

To allow developers to use the calculation method they require, KISSsoft can perform strength calculation in accordance with ISO 6336, DIN 3990, DIN 3991, ISO 10300 or DIN 3991.

15.3.1.1 Only geometry calculation

If you select this method, no strength calculation is performed. Therefore you no longer need to enter the data that is only required for the strength, such as power, application factor, etc.

15.3.1.2 Static strength

The strength calculation for cylindrical gears (see section "Calculation method" on page II-266) is implemented here.

15.3.1.3 Method ISO 6336-B/Literature

We recommend you use the method described here.

The method used to calculate the strength of face gears as originally proposed by Crown Gear [3], is based on the cylindrical gear calculation in accordance with DIN 3990. The inclined lines of contact in a face gear increase the total contact ratio due to pitch overlap. This can be compared with the overlap ratio in helical gear cylindrical gears (an overlap ratio is also present in helical face gears due to the helix angle β_n). You can therefore derive the virtual helix angle β_v from the inclination of the lines of contact.In the strength calculation this effect is taken into account by helix angle factors Y_β and Z_β. The value at the middle of the facewidth is then used as the transverse contact ratio ε_a. It is clear that the face load factor $K_{H\beta}$

and transverse coefficient K_{Ha} in accordance with DIN 3990 cannot be used for face gears. In crown gear calculations these values are usually set to $K_{H\beta} = 1.5$ and $K_{Ha} = 1.1$, and therefore allow for the same procedure to be used as the one used to calculate bevel gears (DIN 3991, ISO 10300). However, the international acceptance of the strength calculation method specified in ISO 6336 makes it a logical alternative to DIN 3990. As ISO 6336 is very similar to DIN 3990, the same restrictions also apply.

In contrast to the Crown Gear program, the following data is used in the calculation:

- The arithmetical facewidth (pitting) corresponds to the minimum line of contact length (Lcont)

- The circumferential force Ft is derived from dPm (middle facewidth)

15.3.1.4 Method Crown Gear (DIN 3990)

This calculation method produces results that correspond to those produced by the Crown Gear program. The underlying principle of calculation is described earlier in the "ISO 6336/Literature" method (see page II-501).

The main differences between it and the "ISO6336/Literature" method are:

- The calculation is based on the method defined in DIN3990.
- The mathematical facewidth (pitting) corresponds to the facewidth (also in cases where the minimum line of contact length is shorter than the facewidth).
- The circumferential force Ft is derived from dPd (reference circle = module * number of teeth), even if dPd is not the middle facewidth.

15.3.1.5 Analog to ISO 10300, Method B

As already mentioned, you can use ISO 10300 as a good alternative method for calculating the strength of bevel gears. Face gears are classified as bevel gears and can therefore be regarded as bevel gears where the cone angle is 0° (pinion) and 90° (face gear). The strength of bevel gears is calculated on the basis of the virtual spur gear (cylindrical gear with the same tooth form as the bevel gear). However, for a face gear, the virtual gear number of teeth for the pinion is $z_{1v} = z_1$ and for the gear z_{2v} it is infinite. If you verify the examples, using the Crown Gear program (method matches DIN 3990) and the ISO 10300 method in KISSsoft you will get a good match of values. The variation in root and flank safeties is less than 10% and usually less than 5%. This shows that both calculation methods in DIN 3990 and ISO 10300 (DIN 3991) are reliable and effective.

15.3.1.6 Analog to DIN 3991, Method B

The same notes as for the "Analog to ISO 10300" method (see page II-502) also apply here.

15.3.2 Service life

The value in the `Service life` input field is used together with the speed to calculate the number of load cycles.

15.3.3 Power, torque and speed

Click the button next to the power input field (for the torque) to calculate the power (torque) so that a predefined safety minimum (see page II-441) can be maintained. Click the button next to the Speed input field to enter the direction of rotation of the face gear as specified in the `Define sense of rotation` window in 15.9 .

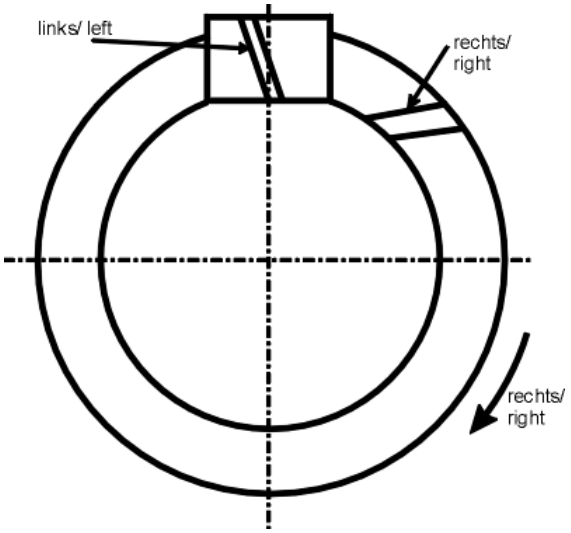


Figure 15.9: Helix angle on a face gear:right; helix angle on the pinion: left; sense of rotation: to the right

15.3.4 Application factor

The application factor compensates for any uncertainties in loads and impacts, whereby $K_A \geq 1.0$ applies. Table 15.8. illustrates the values that can be used for this factor. You will find more detailed comments in ISO 6336.

Operational behavior of the driving machine	**Operational behavior of the driven machine**			
	uniform	**moderate shocks**	**medium shocks**	**heavy shocks**
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 15.8: Assignment of operational behavior to → application factor

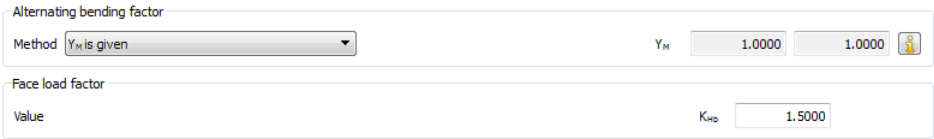


Figure 15.4: `Coefficients` input window in the `Face gears` module

15.4.1 Face load factor

The face load factors $K_{H\beta}$ take into account the effect of uneven load distribution across the facewidth on flank pressure, tooth root load and resistance to scoring. For face gears, we recommend you use approximately the same coefficients (see page II-477) as for bevel gears.

15.5 Modifications

The `Modifications (see page II-326)` (tab) input window in the `Face gears` calculation module basically includes the same functionality as for cylindrical gears. Its special features are listed below:

15.5.1 Addendum reduction

You specify the tip alteration h_{ak} and the length of the tip alteration l_{hak} (see Figure 15.7) in the `Modifications` input window in the `Modifications` area. The tip is then altered to prevent the tooth becoming pointed. When you specify a tip alteration, we recommend you display the entire modification for the 3D export, so that you can increase the number of sections calculated under `Calculation > Settings > General` (→ Additional information (see page II-507)).

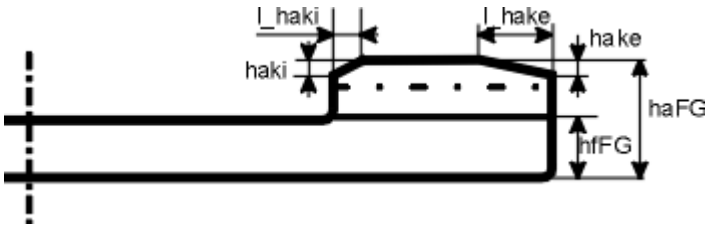


Figure 15.11: Characteristic values of a face gear

15.5.2 Type of modification

In the List of modifications (see section "Type of modification" on page II-328) you can only make changes to the pinion.

15.6 Settings

In the `Calculation` menu you will find the `Settings` option. Click this sub menu to open the `Module specific settings` window. From here you can access the tabs listed below to input other calculation parameters.

15.6.1 General

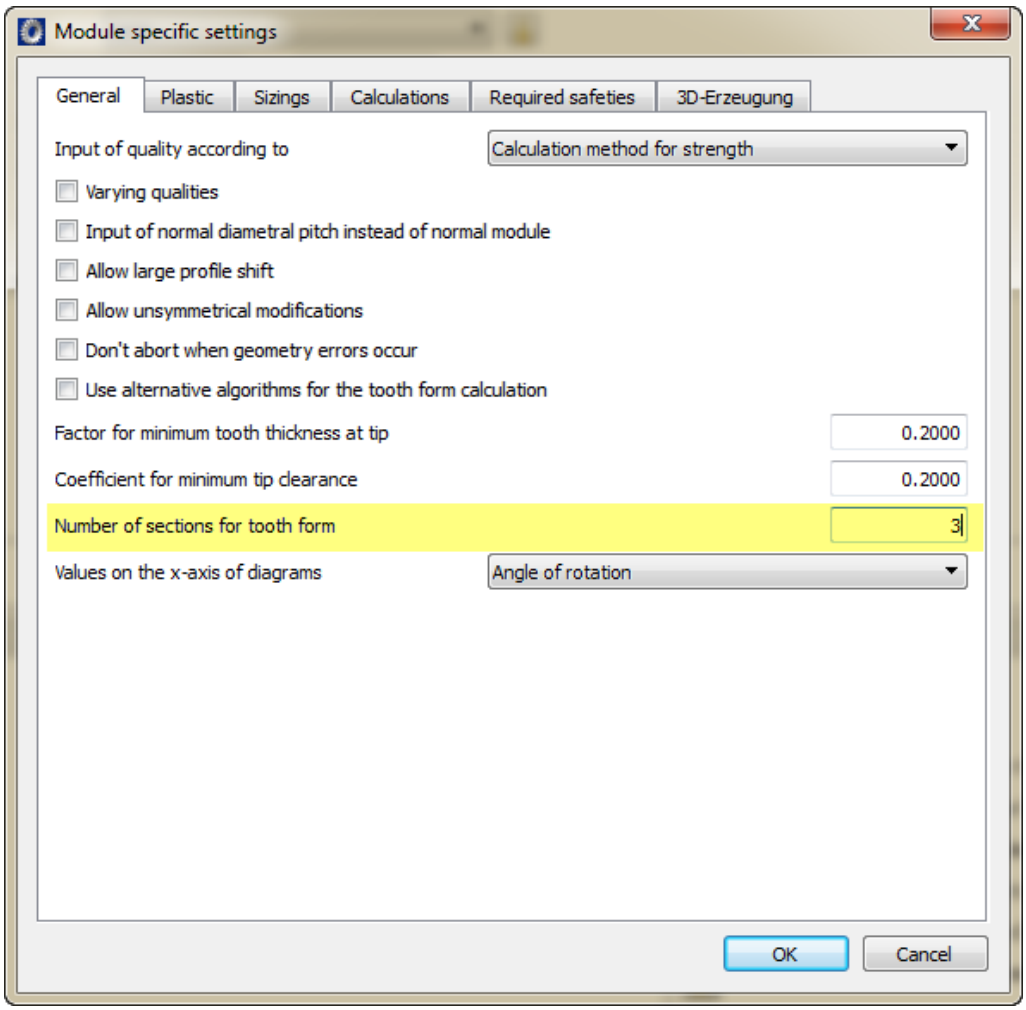


Figure 15.12: `General` tab in the `Module-specific settings window`

The `Number steps for tooth form calculation` input field defines how many equidistant section levels $N \geq 3$ are to be distributed between the outside and inside diameter of the face gear. The default value here is $N = 3$ which defines section levels $r_2 = d_{2i}/2$, $r_2 = d_{2e}/2$ and $r_2 = (d_{2i} + d_{2e})/4$.

NOTE

You should select $N > 10$ to ensure an adequate spatial resolution for your 3D export.

15.6.2 Sizings

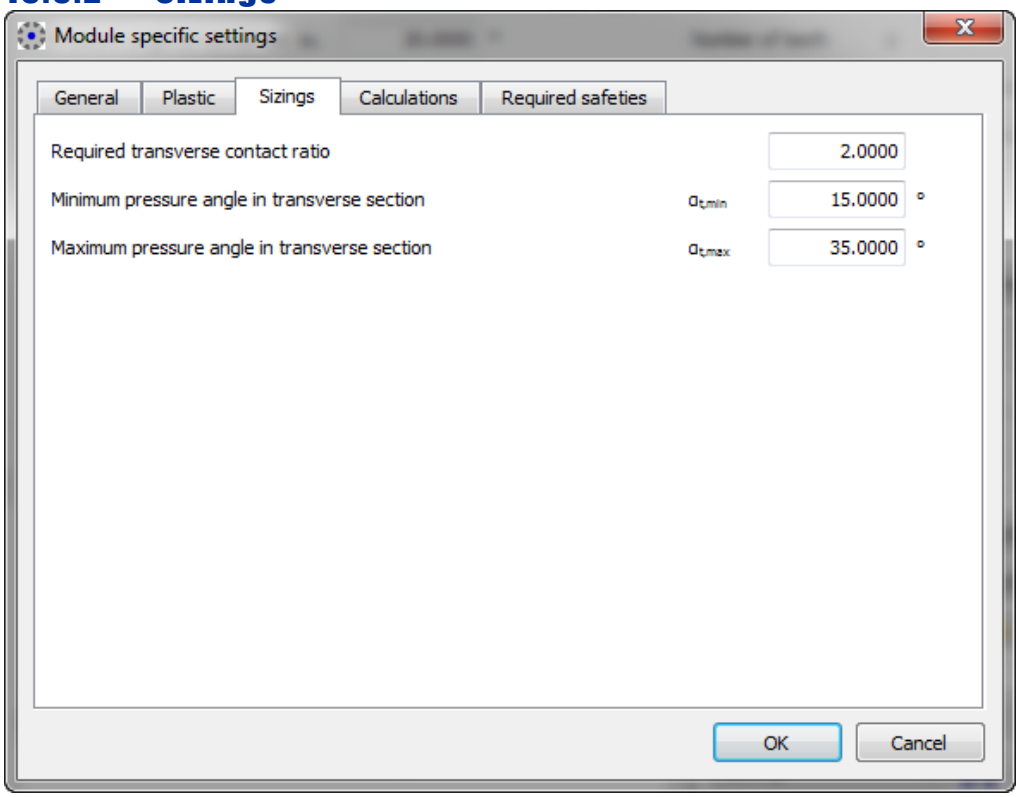


Figure 15.13: `Sizings` tab in the `Module-specific settings window`

The Minimum/maximum pressure angle in transverse section $\alpha_{t,min/max}$ input fields define the range in which the values for the pressure angle for the tooth flank for the face gear across its width may lie. These values are used, for example, when sizing the facewidth of face gear b_2 and axial offset b_v.

15.7 Notes on face gear calculation

15.7.1 Dimensioning

The complexity of dimensioning the tooth forms in face gears means that KISSsoft uses various procedures that differ extensively from other commonly used procedures, such as for cylindrical gears. For a face gear, the geometry you select must be such that it prevents the creation of pointed teeth on the outside face of the gear and ensures that no undercut (or only very little undercut) occurs on the inside face. You must perform these checks when you calculate the tooth form. The actual geometry calculation procedure converts the replacement bevel gear and the replacement cylindrical gear. In the tooth form calculation process, a face gear is calculated in a number of sections set along its facewidth. To specify the number of required sections, select the `Calculation` menu and then, under `Settings > General > Number of sections for the tooth form calculation` , define the number of sections. The `Geometry (meshing)` graphics window allows you to display the tooth form simultaneously on the inside diameter, outside diameter, and in the middle of the tooth. You can see here whether the normal crest width and undercut are tolerable.

You can take these measures to prevent pointed teeth and/or undercut:

- change facewidth b_v
- zoom out the facewidth
- change the pressure angle
- tip alteration in the outside part of the facewidth

NOTES

- To generate a crowned tooth form: You can generate crowning on the tooth trace of face gears by using a pinion type cutter that has one or two more teeth than the meshing pinion. Use the storage function in the `Graphics > Geometry > Meshing` 2D display to check the difference between the generated tooth forms. To do this, define a pinion type cutter with the same number of teeth as the pinion used to calculate the tooth form. Then save this cutter data by clicking the `Gear 2 Save` button and then increase the number of teeth on the pinion type cutter. If the face gear has a large axial offset b_v, you can displace the crowning to one side.

15.7.2 Pinion - Face gear with Z1 > Z2

No provision has been made for calculating a pinion – face gear pairing when the number of teeth on the face gear (Z2) is less than the number of teeth on the pinion (Z1), because this situation does not happen very often. However, under certain conditions, you can still determine the geometry of this type of pairing.

To do this, go to `Settings` and set the `Do not cancel when geometry errors occur` flag. Then, we recommend you follow these steps:

- Reduce the facewidth of the face gear (for example, by half)
- Starting with Z2 = Z1, zoom Z2 step by step, performing a calculation after every step and correcting the inside, middle, and outside aspect of the cuts and, if necessary the tooth depth, in the 2D display.
- Once you achieve the required number of teeth Z2, try to increase the facewidth of the face gear again, and modify it if necessary.

Chapter 16
Worms with enveloping worm wheels

Calculate worm geometry in accordance with ISO 14521 and DIN 3975. Tooth thickness and control mass (base tangent length, rollers and measurement over balls of the worm gear) as specified in ISO 21771. Manufacturing tolerances in accordance with DIN 3974.

You can size the facewidth, the center distance, the lead angle etc. Strength calculation as defined in ISO 14521 or DIN 3996 with: efficiency, temperature safety, pittings safety, wear safety, tooth fracture and bending safety. Data for various different worm wheel materials are supplied.

You can also calculate the starting torque under load, which is a critical value when sizing gear drives.

Flank forms: ZA, ZC, ZH, ZI, ZK, ZN.

For more information about the dimensions of a worm wheel, refer to Figure 16.1.

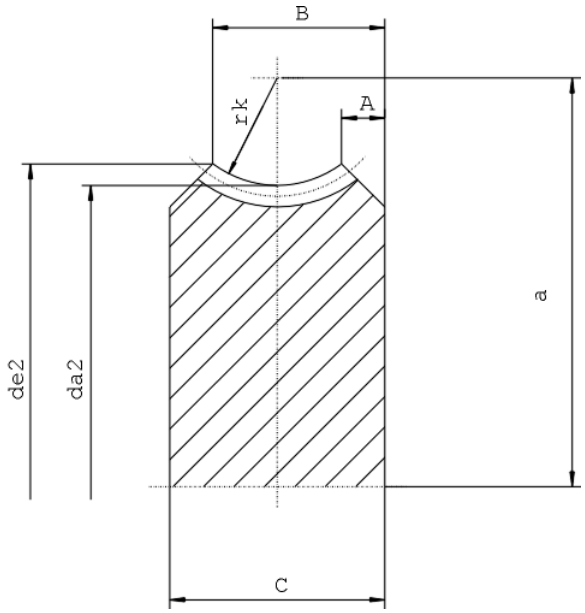


Figure 16.1: Dimensions of the worm wheel

16.1 Underlying principles of calculation

The underlying geometric interrelationships are defined in ISO14521 or DIN 3975. You will find additional information, and other important definitions, such as the various worm flank forms (ZA, ZC or ZI, ZH, ZK, ZN), in [66]. You calculate strength (tooth fracture, pitting, wear and temperature safety) in accordance with ISO14521 or DIN 3996. These calculations take much less time and effort to perform than those required for cylindrical gears. Worms can be checked throughout the manufacture process by using what are known as "three wire measurements". This corresponds to the principle of the measurement over two balls that is used for worm wheels (and also for cylindrical gears). However, the calculations involved in ascertaining the three wire measurement are very complex. A very useful method for standard flank forms has been developed by G. Bock [4] at the physikalisch-technisches Bundesanstalt (German national metrology institute) in Berlin. This method takes into account the shape of the worm's flank which is why it is used in KISSsoft.

NOTE

When you use the term "module" you must differentiate clearly between the axial and the normal module.

Note about how to use the application factor

In cylindrical gear and bevel gear calculations, application factor KA is usually multiplied by the power, for example, so that KA=1 with P= 5 kW gives exactly the same safeties as KA=2 and P=2.5 kW. However, this is different for worm calculations performed in accordance with ISO or DIN standards and may lead to confusion.

The forces and torques are multiplied with the application factor. In contrast, the power is not multiplied with the application factor when determining power loss PVLP and when calculating the total efficiency etaGes. Therefore, for KA=2 and P=2.5 kW instead of KA=1 with P= 5 kW - power loss [PV] will be smaller, and the total efficiency etaGes will be far too low.

Results for the example "WormGear 1 (DIN3996, Example 1).Z80":

	KA=1; P= 5 kW	KA=2; P=2.5 kW	
PVLP	0.140	0.070	<< (* 1/KA)
PVD+PV0	0.199	0.199	=
PVZ	0.530	0.530	=
PV	0.869	0.799	<

etaz	90.00	90.00	=
etaGes	85.19	75.77	<<
theS	76.6	76.6	=
theM	80.9	80.9	=
SW	1.386	1.386	=
SH	1.143	1.143	=
Sdel	2.369	2.369	=
SF	2.251	2.251	=
ST	1.306	1.306	=

This difference in the results is not logical so therefore, to determine PVLP and etaGes, the power is also multiplied with CA to achieve the same results.

16.2 Basic data

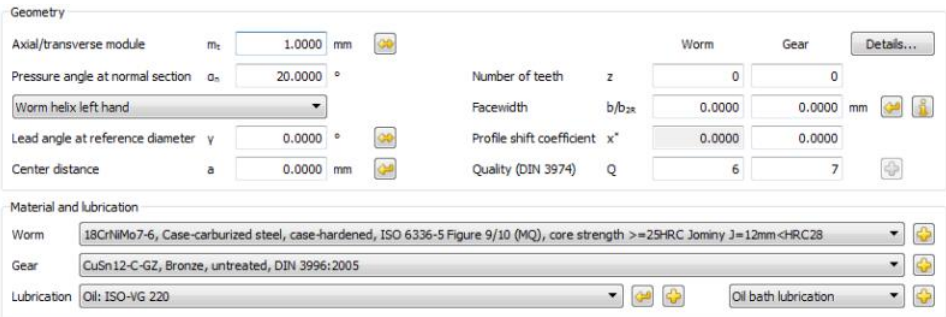


Figure 16.2: `Basic data` input window in the `Worms with globoid worm wheels` module

16.2.1 Axial/transverse module

The axial module of the worm and of the transverse module on a worm wheel are identical. In the `Calculation` menu, select `Settings > Calculations > Calculation with normal module instead of axial module` to use the normal module m_n instead of the axial module in future calculations.

NOTE

This changes the way the tip and root circles (see page II-529) are calculated.

16.2.2 Pressure angle at normal section

The normal pressure angle at the reference circle is also the flank angle of the reference profile. For standard toothings the pressure angle is $\alpha_n = 20^o$. You can use smaller pressure angles for a larger number of teeth to achieve higher contact ratios. Larger pressure angles increase the strength and allow a smaller number of teeth to be used without undercut. In this situation, the contact ratio decreases and the radial forces increase.

16.2.3 Lead angle at reference circle

The lead angle in a worm (gear 1) is the complement of the helix angle and is calculated in accordance with equation 16.1.

$$\gamma = 90° - \beta \tag{16.1}$$

Click the button to open the `Convert lead angle` dialog window in which you can calculate the lead angle from other toothing values. These options are available here: `from center distance`, `from reference diameter` and `from the reference circle and the center distance` (x_2* is modified). A larger lead angle produces greater efficiency, whereas you can design self-locking toothing if you use a smaller lead angle.

16.2.4 Center distance

Click the button to calculate the center distance from the values of profile shift coefficient x^*, number of teeth z and lead angle γ. In this case, you do not receive a message telling you that the calculation has been performed correctly.

16.2.5 Number of teeth

The number of teeth on a worm usually is in the range $1 \leq z1 \leq 4$.

16.2.6 Facewidth

For more information about the dimensions of gear teeth and wheel flange widths, please refer to Figure 16.3. Enter the width of the worm wheel in the facewidth b_{2R} input field. The facewidths b_{2H} and b_2 of the worm wheel are then calculated from this value.

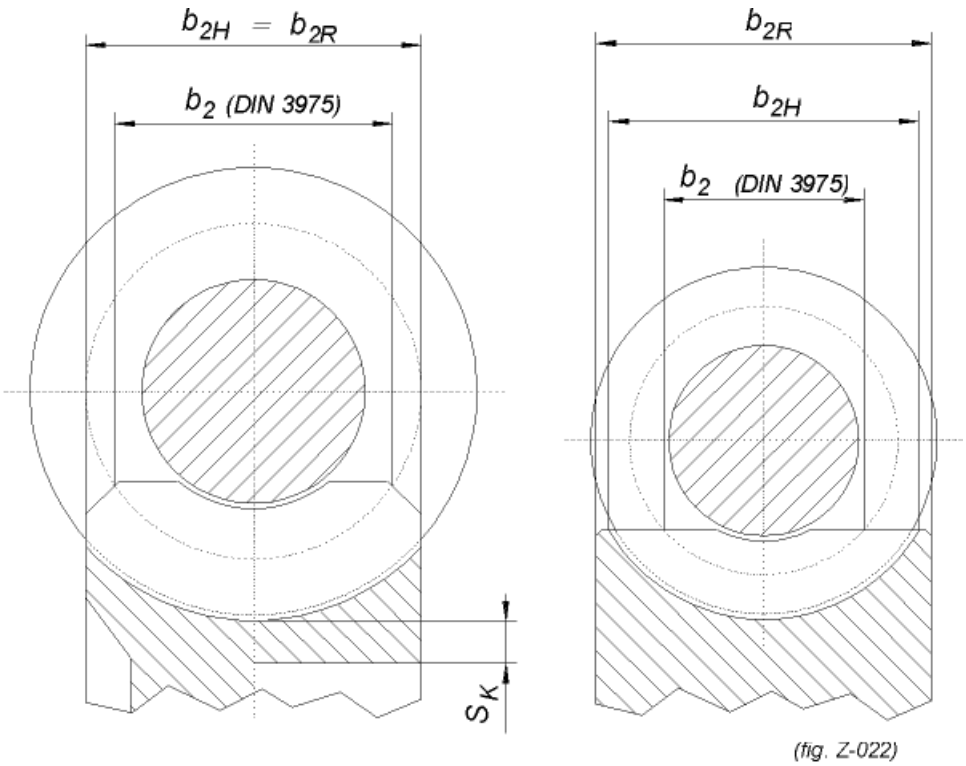


Figure 16.3: Dimensions of gear tooth and wheel flange width

16.2.7 Profile shift coefficient

In the `Worms with globoid worm wheels` calculation module, the profile shift for worm/gear 1 is set to zero (as defined in the ISO 14521 standard). You can only change the tooth thickness of the worm in the `Tolerances` input window.

NOTE

You should use the `Crossed helical gears and precision mechanics worms` calculation module if you require a worm where the profile shift coefficient is $x_1^* \neq 0$.

16.2.8 Tooth thickness modification factor

This factor should only be used in special cases. The factor xs changes the tooth thickness with ΔAs = 2 * xs * as it does with bevel gears. xs2 = -xs1 always applies so the clearance does not change if xs1 is changed Reasonable values for xs1 lie in the range -0.1 to +0.1.

It is useful to apply this factor in situations where: The worm is significantly harder than the gear. In operative use, the gear will wear and cause the drive to fail. This is because the gear teeth will have become thinner and thinner, and will break. If the worm's tooth thickness is changed, for example to xs1 = -0.1, the gear tooth will be thicker. This will result in a much longer service life.

16.2.9 Quality

In this input field you specify the accuracy grade in accordance with the standard shown in brackets. To change the standard used for this calculation, select `Calculation > Settings > General > Input of quality`.

Achievable qualities are shown in Table 16.1.

Manufacturing process	**Quality in accordance with DIN/ISO**		
Grinding	2	. . .	7
Shaving	5	. . .	7
Hobbing	(5)6	. . .	9
Milling	(5)6	. . .	9
Shaping	(5)6	. . .	9
Punching, Sintering	8	. . .	12

Table 16.1: Quality values for different manufacturing processes

16.2.10 Geometry details

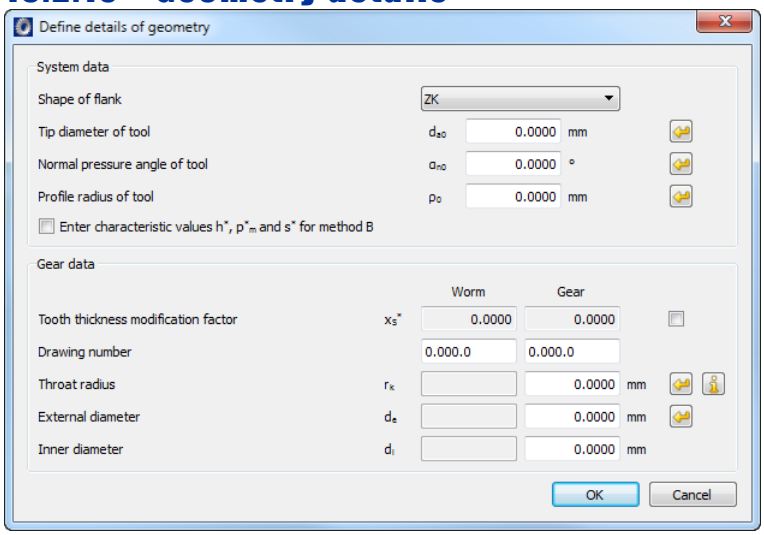


Figure 16.4: `Define details of geometry` window

Click the `Details...` button in the `Geometry` group to open the `Define details of geometry` window in which you can modify the parameters listed below.

16.2.10.1 Shape of flank

The flank shape is a result of the manufacturing process. ZA, ZN, ZK and ZI worms have very similar levels of efficiency and flank load capacity. Although ZC and ZH worms (hollow flanks) have better load capacity in some situations, they do have other major disadvantages.

ZA form:	manufactured on turning machine with tool (straight flanks), mounted in axial section
ZN-form:	manufactured on turning machine with tool (straight flanks), mounted in normal section
ZI form:	manufactured with hobbing cutter (worm flank is involute)
ZK form:	manufactured with grinding wheel (straight flanks), mounted in normal section
ZC, or ZH form:	manufactured with special tools to generate a hollow flank

For more information, please refer to: Dubbel [38], with figures on pages G136 and S79.

16.2.10.2 External diameter and throat radius

You specify values for the external diameter d_{e2} and throat radius r_k as specified in DIN 3975-1:2002-7. In accordance with equations (59) and (67) the following values are suggested for these two dimensions:

$$d_{e2} \approx d_{a2} + m_x$$

$$r_k \geq a - \frac{d_{a2}}{2}$$

with:

d_{a2}	- Tip diameter
m_x	- Axial module
a	- Center distance

16.2.11 Material and lubrication

Materials

The strength calculation method used for worms in accordance with ISO 14521 is based on empirical values determined using these materials:

Worm:

- Case hardening steels (especially 16MnCr5), HRC = 58 to 62
- Heat treatable steels (especially 42CrMo4), heat or induction-hardened, HRC = 50 to 56
- Nitriding steels (especially 31CrMoV9), gas-nitrided

Worm wheel:

- Bronze (GZ-CuSn12, GZ-CuSn12Ni, GZ-CuAl10Ni)
- Grey cast iron (GGG40, GG25)
- Polyamide (PA-12, cast)

To calculate strength you require very special materials data, in particular the wear values. The standard only specifies these values for the most commonly used worm wheel materials (mostly bronze). This is why the selection of materials in KISSsoft is limited. As defining data for materials that are not already documented takes a great deal of time and effort, we strongly recommend you select a material from the list that is closest to the material you actually want to use.

Lubricants

Selecting the right lubricant for a worm gear is extremely important. Synthetic lubricants (polyglycols or polyalfaolephine) can reduce loss and wear enormously.

16.3 Load

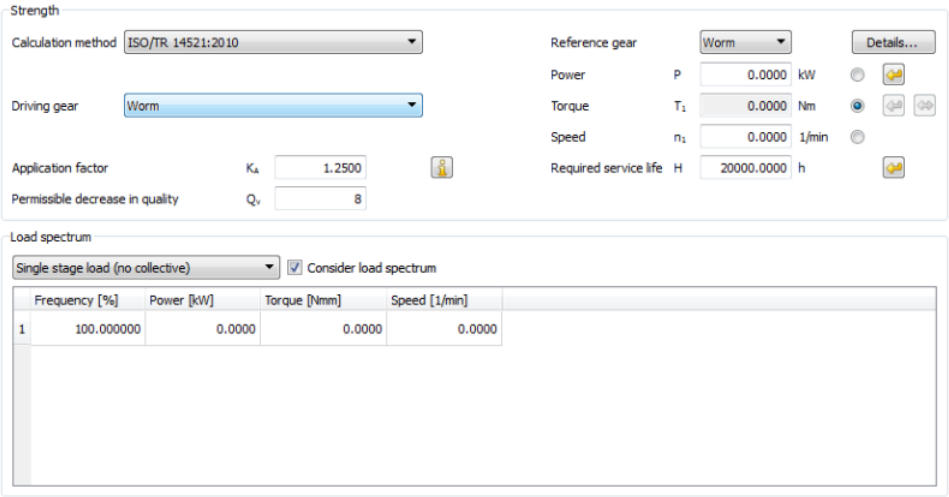


Figure 16.3: `Rating` input window in the `Worms with globoid worm wheels` module

16.3.1 Strength calculation methods

The calculations defined in ISO 14521 and E DIN 3996:2006 are identical.

However, strength calculation as defined in ISO 14521 includes a number of different methods (A,B,C,D;). KISSsoft uses the most precise documented method which usually corresponds to Method B. This calculation method is not suitable for every material (see section "Material and lubrication" on page II-519) because some of the empirical values are missing.

The ISO 14521 standard provides a calculation method for determining:

- Efficiency
- Wear and Wear safety
- Pitting safety
- Root safety
- Bending safety
- Temperature safety

NOTES:

- To calculate strength you require very special materials data, in particular the wear values. The standard only specifies these values for the most commonly used worm wheel materials (mostly bronze). This is why the selection of materials in KISSsoft is limited.
- Grease lubrication: grease lubrication is not mentioned in DIN 3996. In this situation, KISSsoft performs the calculation as for oil bath lubrication. This assumption is permissible, because the lubrication type has very little influence on the calculation.
- Endurance limit values for tooth root load capacity: The standard provides two different values. If you enter the smaller value in the database, no decrease in quality due to plastic deformation of the teeth will be accepted.

16.3.2 Service life

The value in the `Service life` input field is used together with the speed to calculate the number of load cycles.

16.3.3 Application factor

The application factor compensates for any uncertainties in loads and impacts, whereby $K_A \geq 1.0$. Table 16.2. illustrates the values that can be used for this factor. You will find more detailed comments in ISO 6336.

Operational behavior of the driving machine	**Operational behavior of the driven machine**			
	uniform	**moderate shocks**	**medium shocks**	**heavy shocks**
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 16.2: Assignment of operational behavior to → application factor

16.3.4 Permissible decrease in quality

Depending on the construction type of the worm wheel, it may experience a decrease in quality over time due to wear. This value must not sink below the value

specified in this input field. A decrease in quality is linked to the plastic deformation of the material and therefore a higher material property. This, in turn, results in a higher safety against plastic deformation in the root.

16.3.5 Power, torque and speed

Click the button next to the power input field (for the torque) to calculate the power (torque) so that a predefined safety minimum (see section "Required safeties" on page II-441) can be maintained.

16.3.6 Strength details

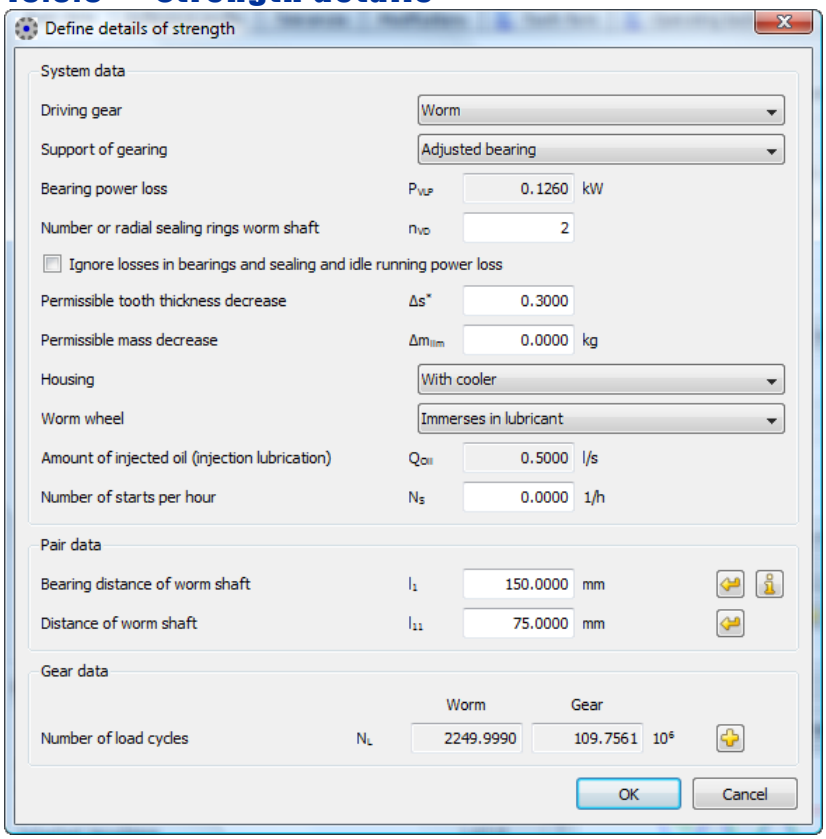


Figure 16.5: `Define details of strength` window

Click the `Details...` button in the `Strength` group to open the `Define details of strength` window in which you can change the following parameters.

16.3.6.1 Support of gearing

The calculation method used to ascertain bearing power loss of the integral worm shaft identifies two different types of bearing.

16.3.6.2 Bearing power loss

If roller bearings are used, the power loss is calculated using the empirical formulae defined in ISO 15451. If sliding bearings are used, you must specify the power loss manually.

16.3.6.3 Number of radial sealing rings, worm shaft

To calculate the power loss in sealing, you must enter the number of radial sealing rings on the integral worm shaft. The sealing gaskets on the worm shaft are not taken into account because their slow rotation speed means they lose very little power (the calculation formulae are defined in ISO 15451).

16.3.6.4 Permissible tooth thickness decrease

The permissible tooth thickness decrease (on the gear) is necessary for calculating the wear safety and taken into account when calculating the root safety. If this input field contains the value 0, the permissible tooth thickness decrease is not checked.

16.3.6.5 Permissible mass decrease

You can limit the permissible mass decrease in kg on the worm wheel (for example, by specifying oil change intervals). This limiting value is also used to define wear safety. If this input field has the value 0, the mass decrease will not be checked.

NOTE

The decrease in mass experienced on the worm is not calculated, because the standard assumes that the worm is harder than the worm wheel and therefore will not be subject to wear.

16.3.6.6 Dimension of the integral worm shaft

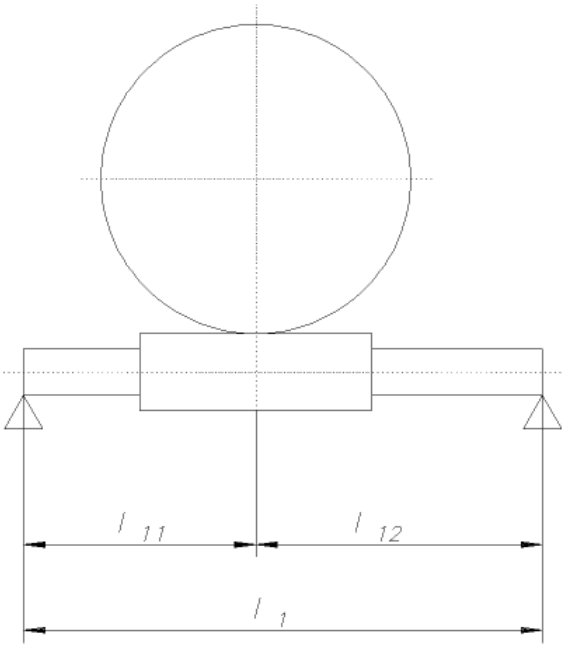


Figure 16.6: Dimensions of the worm-worm wheel

l_1 Distance between the bearings on the integral worm shaft

l_{11} Distance from bearing 1 to the middle of the worm

You need these values to calculate the bending safety. The position of the drive has no effect on the calculation.

16.4 Tolerances

The structure and functionality of the `Tolerances (see page II-322)` input window in the `Worms with globoid worm wheel calculation` module is the same as the Tolerances input window for cylindrical gears. When you enter dimensions for worm calculations, we recommend you click on the `Tooth thickness tolerance` drop-down list and select either the `Worm as defined in Niemann` or `Worm wheel as defined in Niemann` option. The corresponding data is based on recommendations in Niemann [66].

16.5 Settings

In the `Calculation` menu you will find the `Settings` option. Click this sub menu to open the `Module specific settings` window. From here you can access the tabs listed below to input other calculation parameters.

16.5.1 General

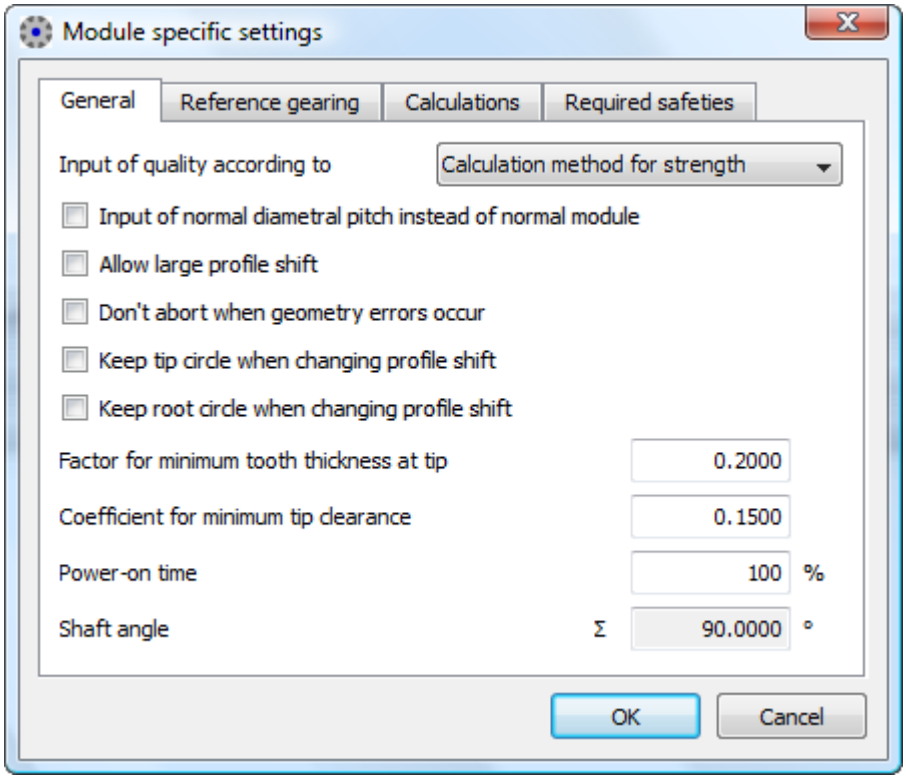


Figure 16.7: `General` tab in the `Module-specific settings` window

(entries that are not detailed here (see page II-428))

16.5.1.1 Power-on time

To calculate the service life, multiply the power-on time with the number of load cycles. The temperature calculation also takes into account the power-on time when it determines the amount of heat generated.

16.5.1.2 Shaft angle

The default value of the shaft angle is set to 90 degrees because this is the default value specified in the strength calculation method defined in DIN 3996. However, you can calculate the geometry with shaft angle that is not 90 degrees by using the `Crossed helical gears and precision mechanics worms` (see page II-532) calculation module.

16.5.2 Reference gearing

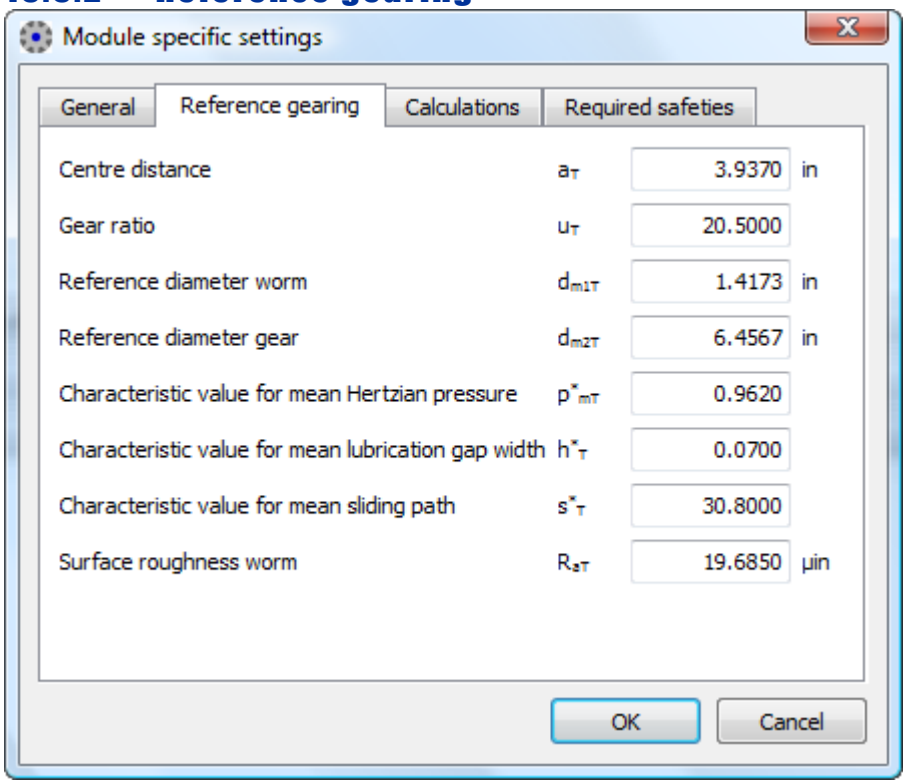


Figure 16.8: `Reference gearing` tab in the `Module-specific settings` window

This calculation is based on a standard reference gearing, on which tests have been performed. The default data corresponds to the reference gearing in ISO 14521. However, if you have the results of your own tests or empirical values, you can modify this calculation to take advantage of this expertise. For a more detailed description, please refer to ISO 14521.

16.5.3 Calculations

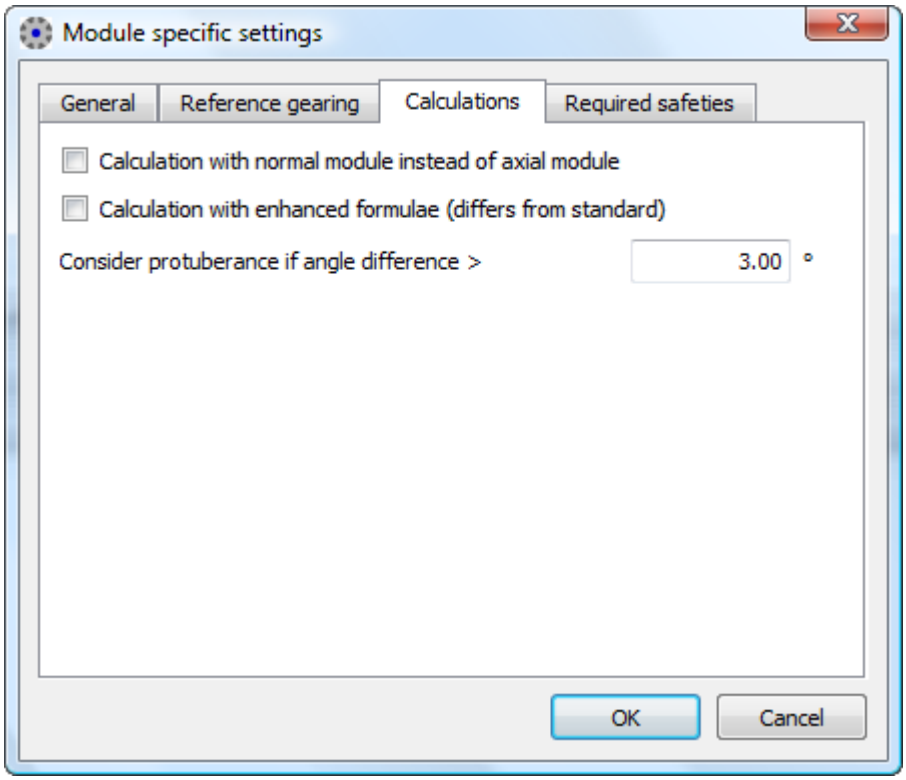


Figure 16.9: `Calculations` tab in the `Module-specific settings` window

16.5.3.1 Calculation with normal module instead of axial module

The geometry of worm gear pairs is usually calculated with the axial module (or transverse module of the worm wheel). If you click on this checkbox, all the values used for the reference profile are calculated with the normal module (tool module). This particularly affects the tip and root circle. In contrast, the profile shift $x^*_x\, m_x$ (m_x for the axial module) remains unchanged.

The formula for the tip circle (m_n for the normal module) is then:

$$d_{a1} = d_{m1} + 2\, m_n\, h_{aP}$$
$$d_{a2} = d_2 + 2\, m_x\, x_2 + 2\, m_n\, h_{aP}$$

For the root circle, the following apply:

$$d_{f1} = d_{m1} - 2\, m_n\, h_{fP}$$
$$d_{f2} = d_2 + 2\, m_x\, x_2 - 2\, m_n\, h_{fP}$$

16.5.3.2 Calculation with enhanced formulae (differs from standard)

If you select this checkbox, alternative calculation methods are used at these points:

- Effective tooth thickness on the tip (instead of formula (84): calculated in accordance with DIN or formula (110) in accordance with ISO)
- Loss of power on toothing PVZ with coefficient 1/9.550 Instead of 0.1

16.5.4 Required safeties

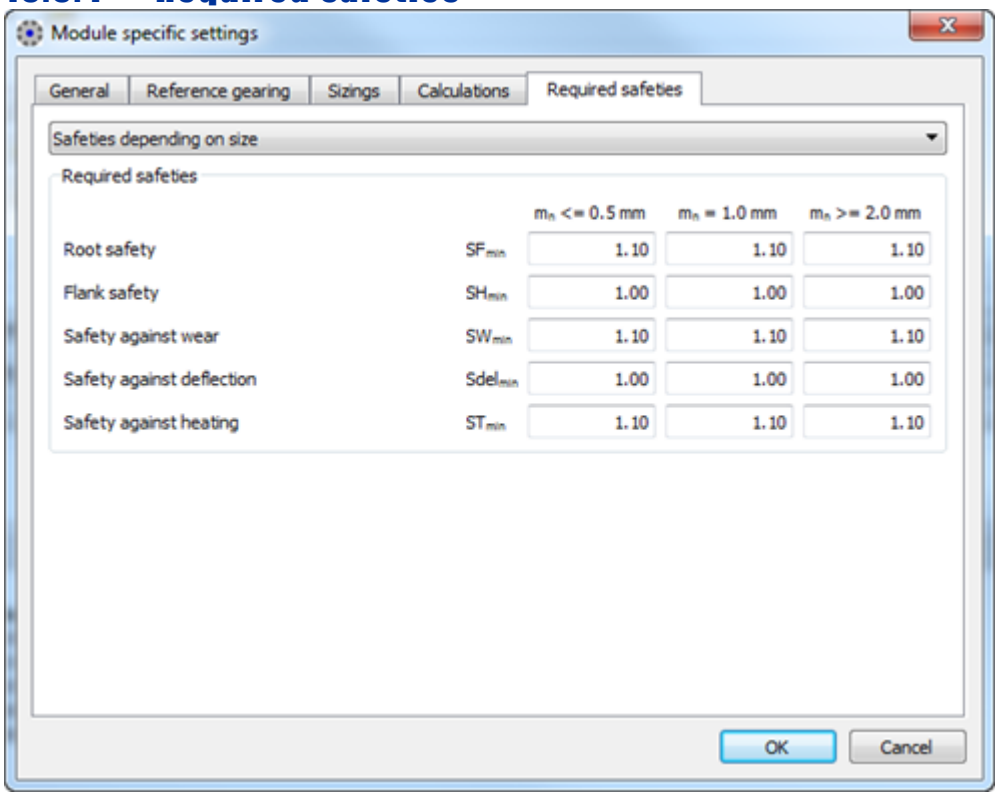


Figure 16.10: `Required Safeties` tab in the `Module-specific Settings` window

KISSsoft issues an error message if the specified required safeties have not been reached after you completed the calculation. Sizing is always calculated on the basis of the required safeties for tooth fracture, pitting and wear. If you do not wish to use one or more of these criteria, set the appropriate required safety to zero. In accordance with ISO 14521 you must ensure the following safeties:

Root safety	:	1.1
Pitting safety	:	1.0
Wear safety	:	1.1
Bending safety	:	1.0
Temperature safety	:	1.1

You can change these values as required to reflect your own findings.

Chapter 17
Crossed helical gears and precision mechanics worms

Crossed helical gears are helical gears that are mounted on crossed axes. The shaft angle is usually $\Sigma = 90^{o}$. In contrast to the line contact shown in enveloping worms, crossed helical gears only contact at one point when they are generated. As a result, they can only transmit very small forces and are primarily used for control purposes.

In precision engineering, a worm wheel is often manufactured in the same way as a helical gear. This makes it easier to produce and assemble than a globoid gear manufactured using a worm-shaped cutter. In this situation, you should calculate the geometry of the worm wheel in the same way as a helical gear. This is because, if the profile shift total is not equal to zero, the helix angle of the gear will not match the lead angle of the worm. Both gears have the same hand of gear. If the worm has right hand teeth, then the worm wheel is also right hand. The total of both helix angles at the operating pitch circle/spiral is exactly the same as the shaft angle. However, due to the profile shifts, the total of helix angles at the reference circle is not identical to the shaft angle.

In special cases, the shaft angle can also be smaller than the helix angle of gear 1. In this situation, gear 2 has the opposite hand of gear to gear 1.

17.1 Underlying principles of calculation

The method used to calculate crossed helical gears (cylindrical gears with crossed axes) is defined in [66]. The current version of this standard describes methods used to calculate and check the geometry of spiral toothed gears for any shaft angle. The measures used for checking and fabrication are determined arithmetically.

Although the method detailed in Niemann [66] is used to calculate the root and flank strength and the scuffing safety as concept, the individual equations used are following ISO 6336. (Niemann uses equations from an old edition of DIN3990).

17.2 Basic data

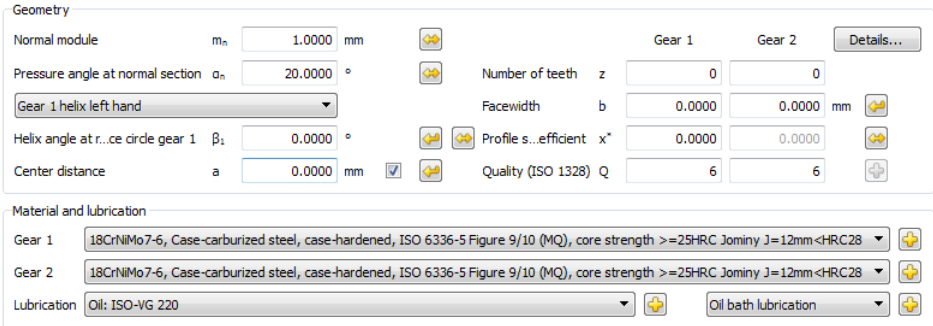


Figure 17.1: `Basic data` input window

17.2.1 Normal module

Enter the normal module. However, if you know the pitch, transverse module or diametral pitch instead of this, click on the button to open a dialog window in which you can perform the conversion. If you want to transfer the diametral pitch instead of the normal module, you can select `Input normal diametral pitch instead of normal module` by selecting `Calculation > Settings > General`.

17.2.2 Pressure angle at normal section

The normal pressure angle at the reference circle is also the flank angle of the reference profile. For standard toothings the pressure angle is $\alpha_n = 20^o$. You can use smaller pressure angles for a larger number of teeth to achieve higher contact ratios. Greater pressure angles increase the strength and allow a smaller number of teeth to be used without undercut. In this situation, the contact ratio decreases and the radial forces increase.

17.2.3 Helix angle reference circle gear 1

The center distance, number of teeth, profile shift (x^*_1, x^*_2) and shaft angle are used to calculate the helix angle of gear 1. It often happens that several helix angles meet the requirements of the toothing geometry. In this situation, when you click the button you see an `Information` window that lists the possible values. Here the solution that is closest to **the current value** is selected automatically. However, if only one value is suitable for the sizing, it is transferred into the input

field without any messages being displayed. If the sizing function is unable to find any solutions, it displays a warning message and you must then change either the center distance or the module.

17.2.4 Center distance

The center distance is calculated on the basis of the helix angle of gear 1, the shaft angle, the profile shift (x^*_1, x^*_2) and the number of teeth.

17.2.5 Facewidth

Because the facewidth must have a minimum value, the input field has a button which you can use to define the minimum width based on the parameters you have already defined.

17.2.6 Profile shift coefficient

The tool can be adjusted during manufacture. The distance between the production pitch circle and the tool reference line is called the profile shift. To create a positive profile shift, the tool is pulled further out of the material, creating a tooth that is thicker at the root and narrower at the tip. To create a negative profile shift the tool is pushed further into the material, with the result that the tooth is narrower and undercutting may occur sooner. In addition to the effect on tooth thickness, the sliding velocities will also be affected by the profile shift coefficient.

Click the button and KISSsoft will determine whether the profile shift coefficient (see page II-253) is to be taken from measured data or from values given in drawings.

NOTE

If one of the two profile shift values appears in gray, this means it will be calculated by KISSsoft. This is what happens when you activate the checkbox for entering the center distance. If you overwrite a gray field, it will become active and KISSsoft will calculate the value for one of the other gears.

17.2.7 Quality

In this input field, you specify the accuracy grade in accordance with the standard shown in brackets. To change the standard used for this calculation, select `Calculation> Settings > General > Input of quality`. The

accuracy grade in accordance with ISO 1328 is very similar to that in DIN 3961 or AGMA 2015.

Achievable qualities are shown in Table 17.1.

Manufacturing process	Quality in accordance with DIN/ISO		
Grinding	2	. . .	7
Shaving	5	. . .	7
Hobbing	(5)6	. . .	9
Milling	(5)6	. . .	9
Shaping	(5)6	. . .	9
Punching, Sintering	8	. . .	12

Table 17.1: Quality values for different manufacturing processes

17.2.8 Define details of geometry

Click the `Details...` button in the `Geometry` group to open the `Define details of geometry` window in which you can modify the parameters listed below.

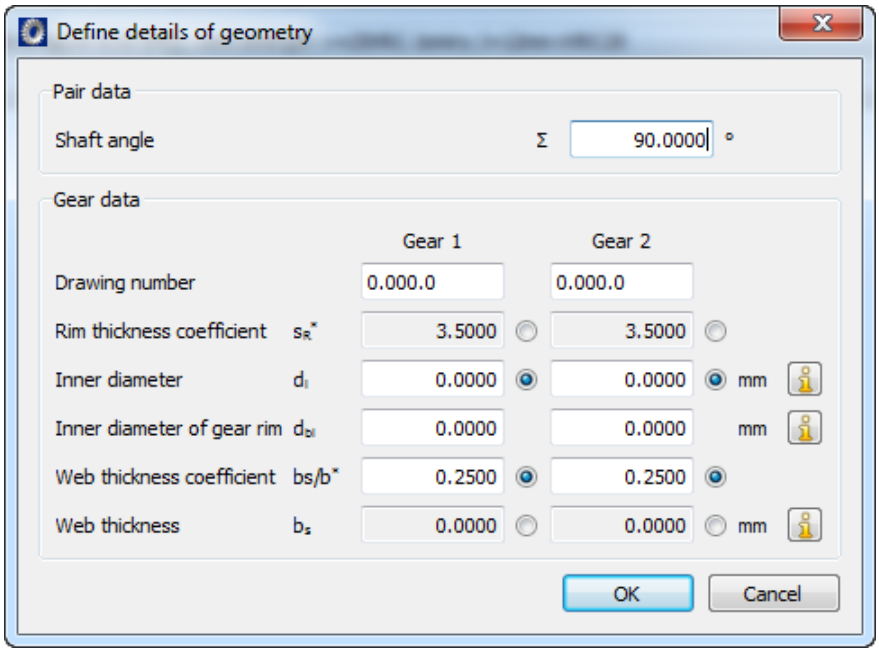


Figure 17.2: `Geometry details` input window

17.2.8.1 Shaft angle

The shaft angle is usually $\Sigma = 90^{o}$, but you can specify your own value here.

17.2.8.2 Inside diameter

The inside diameter is needed to calculate the mass moment of inertia. As defined in ISO or AGMA, the gear rim thickness does affect the strength. For complete gears, enter 0, for external gears with a web, enter the appropriate diameter d_i as shown in Figure 17.3. For internal wheels, enter the external diameter of the gear rim.

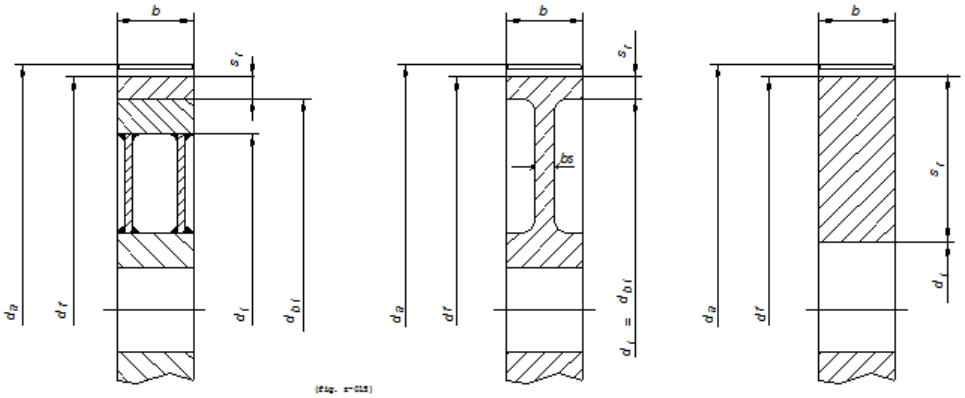


Figure 17.3: Measures of the diameter

The inside diameter of the gear's flange is required for calculations in accordance with ISO or AGMA. Where thin gear rims are used, this factor can greatly influence the calculation results. See also the figure shown above <Kap16:>.3.

17.2.9 Material and lubrication

The materials displayed in the drop-down lists are taken from the materials database. If you cannot find the material you require in this list, you can either select `Own Input` from the list or enter the material in the database first (→ Additional information (see page I-114)). Click the button to open the `Material gear 1(2)` window in which you can select a material from the list of materials available in the database. Select the `Own Input` option to enter specific material characteristics. This option corresponds to the `Create a new entry`window in the database tool.

17.3 Load

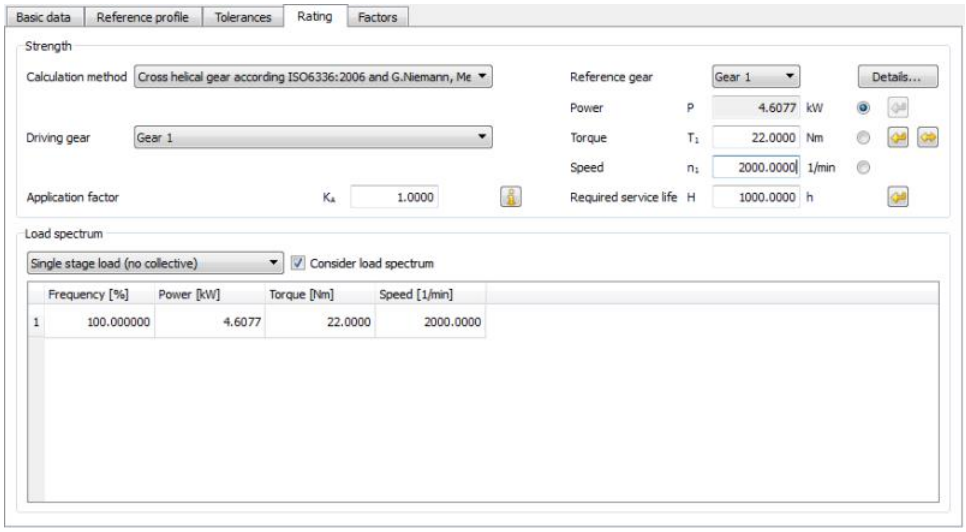


Figure 17.2: `Load` input window

17.3.1 Methods used for strength calculation

As yet, no binding standard has been drawn up for the calculation of crossed helical gears. KISSsoft therefore recommends you use the ISO 6336 (see page II-539) Calculation of load capacity of spur and helical gears.

You can use one of three different methods to calculate the strength of worms:

17.3.1.1 Strength calculation in acc. with Hirn

The method used to calculate worms as defined by H.Hirn is based on an obsolete edition of Niemann's machine elements. It calculates the temperature safety, the flank safety, the root safety, and the bending safety. Although the material values cannot be compared with the values for worm calculation as defined in DIN 3996, the safeties are, however, similar.

We do not recommend you to use this obsolete method.

NOTE

The calculation method defined in Hirn also selects a material pairing. This must match the material pair selected in the `Materials and lubrication` area. Shaft angle $\Sigma = 90^{o}$ and $z_1 < 5$.

17.3.1.2 Strength calculation in acc. with Hoechst

You can use the strength calculation in accordance with Hoechst for worm wheels made from Hostaform® (POM), paired with steel worm gears [80]. The permitted load coefficient is c [N/mm^2], see Gln. (17.1) ÷ (17.3) is a value that defines the temperature resistance. This method also checks the worm's permitted flank pressure and blocking safety. The decisive value for blocking safety is maximum load, not continuous load.

$$c = \frac{F_2}{f_z \cdot b \cdot m \cdot \pi} \tag{17.1}$$

$$b = \sqrt{d_{a1}^2 - d_{m1}^2} \tag{17.2}$$

$$m = \frac{m_n}{\cos(\gamma_m)} \tag{17.3}$$

where

F_2	circumferential force on the worm wheel
f_z	coefficient for number of teeth
b	usable width
m_n	normal module
γ_m	mean lead angle
d_{a1}	tip diameter of worm
d_{m1}	reference diameter of worm

NOTE:

Shaft angle $\Sigma = 90^o$ and $z_1 < 5$. The calculation method involves a worm made of steel and a crossed helical gear made of plastic.

17.3.1.3 Strength calculation in acc. with ISO 6336/Niemann

You can perform the strength calculation for crossed helical gears with $z_1 \geq 5$ as defined in Niemann [66]/ISO 6336. As stated in Niemann, the contact ellipse is calculated using a for the width and b for the height of the half axes. An effective facewidth of 2a is assumed for flank safety (pitting). The same value plus twice the

module value is used to calculation the strength of the tooth root. This corresponds to the specifications given in ISO 6336, if the facewidth is greater than the contact width. Scuffing safety is calculated as defined in Niemann [66]. This method differs from the DIN 3990-4 guideline because of the high sliding velocities of the crossed helical gears. It is more similar to the method applied to hypoid bevel gears. It supplies proof of tooth root resistance, flank load capacity and resistance to scoring.

NOTE:

If the number of teeth is $z < 5$, this calculation supplies tooth root and contact stress safeties that are too high.

17.3.1.4 Strength calculation as defined in VDI 2736

This VDI guideline is still at the draft stage. It defines how precision mechanics worms are to be calculated.

17.3.1.5 Static calculation

The static calculation performs a static estimate of safety against fracture and yield point. This calculation is performed in accordance with the formulae documented in 13.2.11 Static calculation.

The calculation for worm gears returns safeties that tend to be too great, because worms are usually checked for safety against shearing.

17.3.1.6 Static calculation on shearing

Verification of a worm wheel on shearing

$\tau_F = F_t^2 * K_A * Y_E / A_\tau$

$A_\tau = b_{max}/5*(4*s_{tda2}-s_{tdx2})$

$d_{x2} = 2* a-d_{a1}$

This calculation is performed automatically and is documented in the report in Section 6A.

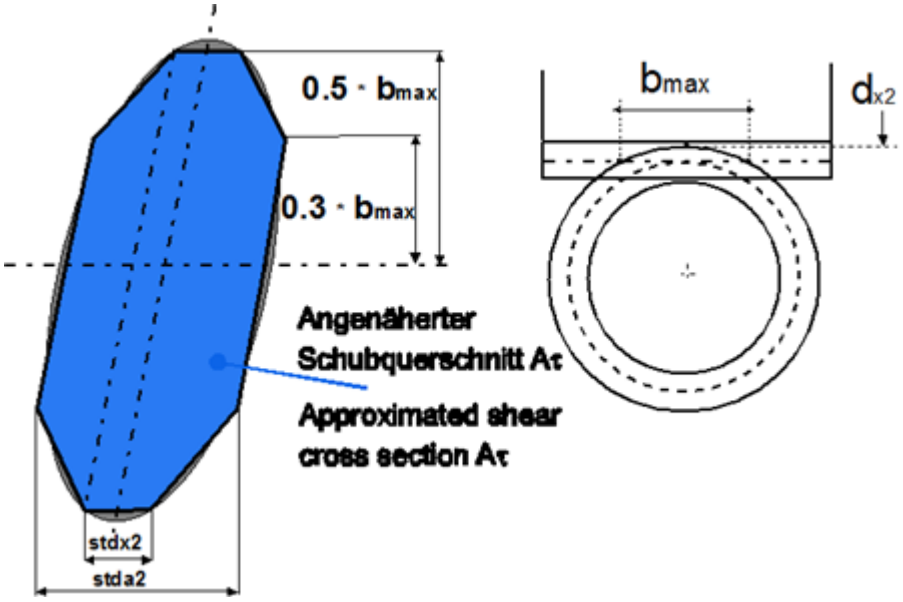


Figure 17.4: Dimensions of the shear cross section.

17.3.2 Service life

The system displays the required service life in the input field. To enter it directly, and perform sizing, click the button. Based upon the minimum safety value for the tooth root and flank strength, this process calculates the service life (in hours) for every gear and for every load you specify. The service life is calculated in accordance with ISO 6336-6:2006 using the Palmgren-Miner Rule. In the range of endurance limit, you can also select a modified form of the Woehler line instead of ISO 6336 or DIN 3990. The system service life and the minimum service life of all the gears used in the configuration is displayed. You can size the service life using the button either with or without defining a load spectrum (see page II-287). You will find more detailed information about defining load spectra in section 13.19 (see page II-287).

NOTE

Only the ISO 6336 method includes a calculation for the service life.

17.3.3 Application factor

The application factor compensates for any uncertainties in loads and impacts, whereby $K_A \geq 1.0$. Table 17.4 illustrates the values that can be used for this factor. You will find more detailed comments in ISO 6336.

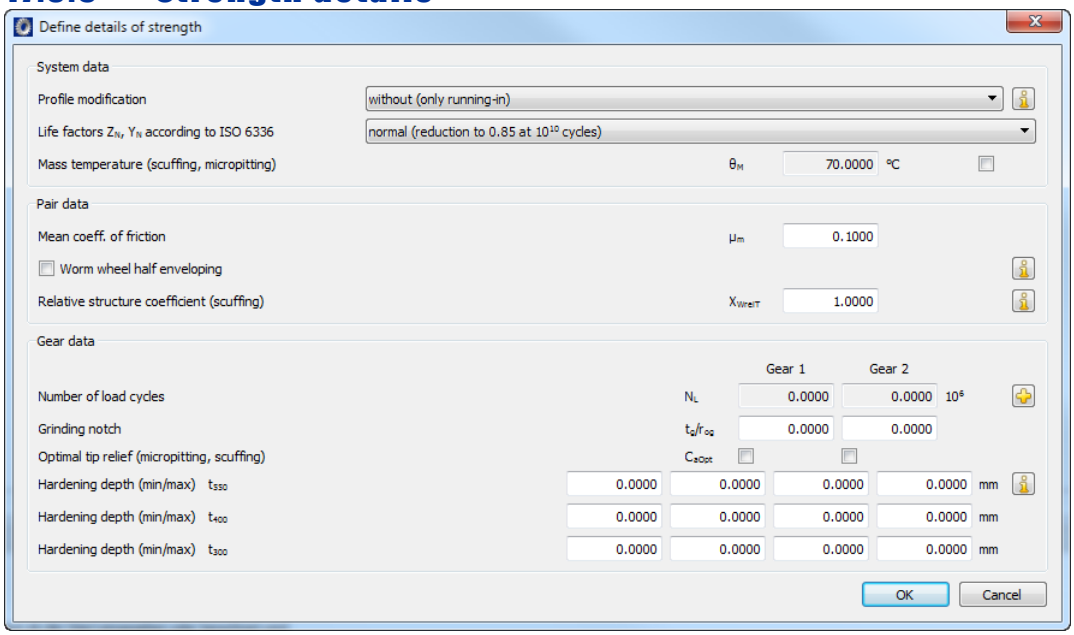
Operational behavior of the driving machine	**Operational behavior of the driven machine**			
	uniform	**moderate shocks**	**medium shocks**	**heavy shocks**
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 17.4: Assignment of operational behavior to application factor

17.3.4 Power, torque and speed

Click the button next to the power input field (for the torque) to calculate the power (torque) so that a predefined safety minimum (see page II-441) can be maintained. Click the button next to the power input field to apply a frequency distribution for power, torque and speed in the `Define load spectrum (see page II-287)` window.

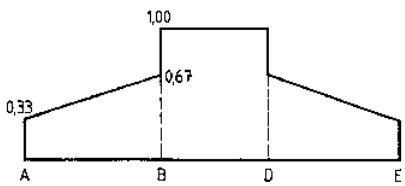
17.3.5 Strength details



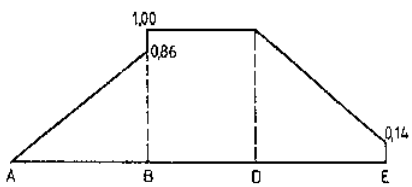
Click on the `Details...` button to open the `Define details of load` window which is divided into `System data`, `Pair data` and `Gear data`.

17.3.5.1 Profile modification

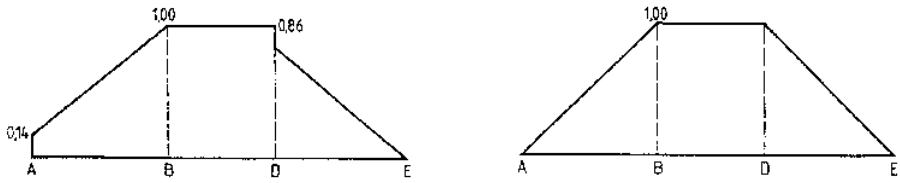
You can modify the theoretical involute in high load capacity gears by grinding/polishing the teeth. You will find suggestions for sensible modifications (for cylindrical gears) in KISSsoft Module Z15 (see section "Modifications" on page II-326). The type of profile modification has an effect on how scuffing safety is calculated. The load sharing factor X_Γ is calculated differently according to the type of profile modification used. The main difference is whether the profile has been modified or not. However, the differences between `for high load capacity` and `for smooth contact` are relatively small. The strength calculation standard presumes that the tip relief C_a is properly dimensioned but does not provide any concrete guidelines. The resulting load sharing factor X_Γ in accordance with DIN 3990, depends on the type of profile modification:



(a) no profile modification



(b) high performance gears; pinion drives



(c) high performance gears; gear drives (d) balanced contact

Figure 17.9: Load sharing factor X_Γ for different profile modifications

17.3.5.2 Lifetime factors as defined in ISO 6336

The fatigue limit factor Z_{NT} reduces the permitted material stress in accordance with ISO 6336-2:2006:

$$\sigma_{H\lim red} = Z_{NT} \cdot \sigma_{H\lim} \tag{12.14}$$

$$\sigma_{F\lim red} = Y_{NT} \cdot \sigma_{F\lim} \tag{12.15}$$

As stated in ISO 6336, this value is important for cylindrical gear calculations and is the reason for the lower safety values for the range of endurance limit when compared with DIN 3990.

1. **normal (reduction to 0.85 at 10^{10} cycles):** The permitted material stress in the range of endurance limit (root and flank) is reduced again. Fatigue strength factors Y_{NT} and Z_{NT} are set to 0.85 for $\geq 10^{10}$ load cycles.
2. **increased with better quality (reduction to 0.92):** Y_{NT} and Z_{NT} at $\geq 10^{10}$ load cycles are set to 0.92 (in accordance with the data in ISO 9085).
3. **with optimum quality and experience (always 1.0):** This removes the reduction and therefore corresponds to DIN 3990. However, this assumes the optimum treatment and monitoring of the materials.

17.3.5.3 Relative structure coefficient (scuffing)

The relative structure phase coefficient takes into account differences in materials and heat treatment at scoring temperature. However, the standards do not provide any details about how to proceed when different types of material have been combined in pairs. You must input this coefficient yourself because it is not set automatically by KISSsoft.

Relative structure phase coefficient as defined in DIN 3990, Part 4:

Through hardened steels	1.00
Phosphated steel	1.25
Coppered steel	1.50
Nitrided steel	1.50
Case-hardened steels	1.15 (with low austenite content)
Case-hardened steels	1.00 (with normal austenite content)
Case-hardened steels	0.85 (with high austenite content)
Stainless steels	0.45

The standard does not provide any details about how to apply the coefficient when the pinion and gear are made of different material types. In this case it is safer to take the lower value for the pair.

17.3.5.4 Number of load cycles

KISSsoft calculates the number of load cycles from the speed and the required service life. If you want to influence the value, you can define it in the `Number of load cycles for gear n window`. Click the button to access this. In this window you can select one of five different options for calculating the number of load cycles.

1. **Automatically** The number of load cycles is calculated automatically from the service life, speed, and number of idler gears.
2. **Number of load cycles** Here you enter the number of load cycles in millions. You must select this option for all the gears involved in the calculation to ensure this value is taken into account.
3. **Load cycles per revolution** Here you enter the number of load cycles per revolution. For a planetary gear units with three planets, enter `3` for the sun and `1` for the planets in the input field.

   Note:
   If the `Automatically` selection button in the calculation module is enabled, KISSsoft will determine the number of planets in the `Planetary stage calculation` module.
4. **Load cycles per minute** Here you enter the number of load cycles per minute. This may be useful, for example, for racks or gear stages where the direction of rotation changes frequently, but for which no permanent speed has been defined.

5. **Effective length of rack** The rack length entered here is used to calculate the number of load cycles for the rack. The rack length must be greater than the gear's perimeter. Otherwise, the calculation must take into account that not every gear tooth will mesh with another. You must enter a value here for rack and pinion pairs. Otherwise the values N_L(rack) = N_L(pinion)/100 are set.

NOTE

This calculation method is used for transmissions with a slight rotation angle.

This scenario assumes that a reduction is present

$$i = \frac{z_2}{z_1}$$

and a pivoting angle w in [°] from gear 2, where gear 2 constantly performs forwards and backwards movements by the angle value w. The effective endurance is given as the service life. The two coefficients N_1 and N_2, which reduce the absolute number of load cycles, N_L, are now calculated. To do this:

a) Set the alternating bending factor of the pinion and gear to 0.7, or calculate it as defined in ISO 6336-3:2006. In this case, a complete forwards/backwards movement is counted as a load cycle.

b) For the pinion, coefficient N_1 is determined as follows:

$$N_1 = 0.5 \qquad \text{für } w \cdot i > 360$$

$$N_1 = 0.5 \cdot \max(1.0, 360 / w / i) \qquad \text{für } w \cdot i \leq 360$$

c) The number of load cycles of teeth in contact in gear 2 is smaller by a coefficient of N_2 when compared with the number of load cycles during continuous turning.

$$N_2 = 0.5 \cdot 360 / w$$

Factor 0.5 takes into account both the forwards and backwards movements.

d) Enter coefficients N_1 and N_2 in the `Load cycles per revolution` input field.

The correct number of load cycles can now be calculated on the basis of the data entered in steps a to d.

17.3.5.5 Optimal tip relief

To calculate safety against micropitting as specified by Method B in ISO 15144, you must specify whether or not the profile modification is to be assumed to be optimal. The same applies to calculating the safety against scuffing. The software checks whether the effective tip relief (Ca) roughly corresponds to the optimum tip relief (Ceff). If this check reveals large discrepancies, i.e. Ca < 0.333*Ceff or Ca > 2.5*Ceff, a warning is displayed. In this case, the value you input is ignored and documented accordingly in the report.

17.3.5.6 Hardening depth, known by its abbreviation "EHT"

You can input the intended hardening depth (for hardness HV400, for nitrided steels, or HV550 for all other steels). You can also input the hardness HV300. This value is then used to display the hardening curve as a graphic. The input applies to the depth measured during final treatment (after grinding).

When you input this data, the safety of the hardened surface layer is calculated automatically according to DNV 41.2 [93]. Here a minimum value of t400 (nitrided steel) or t550 (all other steels) is used. If only the value for HV300 is known, this value is then used. However, the calculation should then only be seen as an indication. The calculation is performed as described in the section in [93] "Subsurface fatigue". The values required to define the EHT coefficient YC as specified in DNV 41.2 are also needed. The calculation is performed using different solutions than the calculation of the proposal for the recommended hardening depth, but still returns similar results (proposal for hardening depth). To achieve a proposal for a sensible hardening depth, we recommend you call the calculation in `Report -> Proposals for hardening depth`. A maximum value for the hardening depth is only used to check the hardening depth at the tooth tip. It is mainly used for documentation purposes.

17.4 Settings

In the `Calculation` menu you will find the `Settings` option. Click this sub menu to open the `Module specific settings` window. From here you can access the tabs listed below to input other calculation parameters. (the following parameters are not described here (see page II-428))

17.5 Notes

17.5.1 Checking the contact pattern

The collision check shown in the 2D graphic (Meshing (see page II-575)) can only be used to a limited extent for crossed helical gears because it only works for a shaft angle of 90°, does not take flank line modifications into account, and only represents generating in the axial section.

A better option here is to generate a 3D model which includes all the flank line modifications and works for any shaft angle. The "Skin model" 3D variant can be used to represent the contact pattern and check it exactly when the gears are being generated. To do this, click the appropriate function button to rotate one gear slightly against the other until the contact pattern appears, and then generate the two gears. To ensure the gears do not engage too fully, we recommend you set the number of rotation steps to 30 or higher (in Properties).

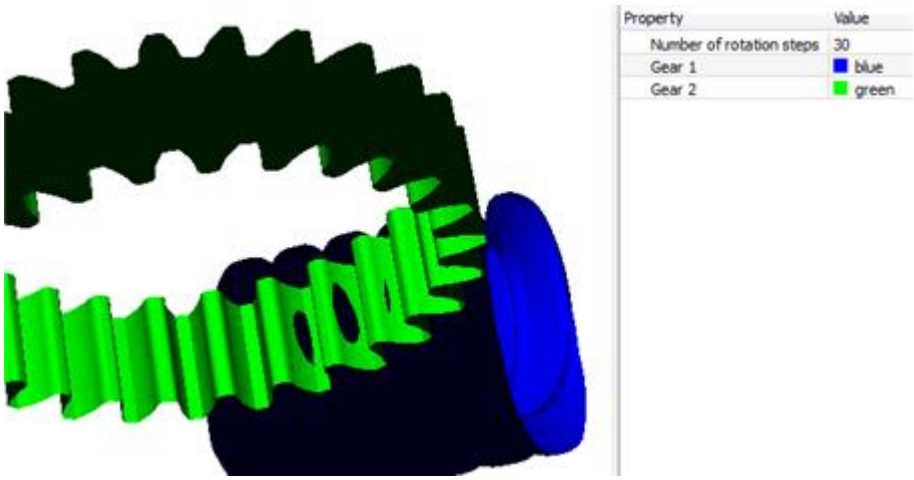


Figure 17: Contact pattern of a worm gear toothing

Chapter 18
Beveloid gears

This module is a prototype for the 2014 release. Although the majority of the functions are ready for use, a few points have yet to be completed.

It may not always be possible to actually generate every tooth form that can be input. We therefore recommend you use tooth forms without preliminary treatment in the Reference profile tab.

As no standards have yet been defined for beveloid gears, the calculation is based on well known technical documentation, and also more recent publications. The geometry is calculated on the basis of these two sources:

- K. Roth; Zahnradtechnik – Evolventen-Sonderverzahnungen zur Getriebeverbesserung (special meshings for involutes for improving gear units(, Springer DE, 1998
- S.-J. Tsai, S.-H. Wu; Geometrical Design of Conical Gear Drives with Profile-shifted Transmission; 12th IFToMM World Congress, 2007

Kapitel 19
Non circular gears

KISSsoft's noncircular gear analysis allows you to calculate gears with noncircular gear bodies.

19.1 Input data

Input the geometry, generation and tolerance values in the **Basic data** tab.

Then, enter the details for generating noncircular gears in the **Reference profile** tab.

19.1.1 Geometry

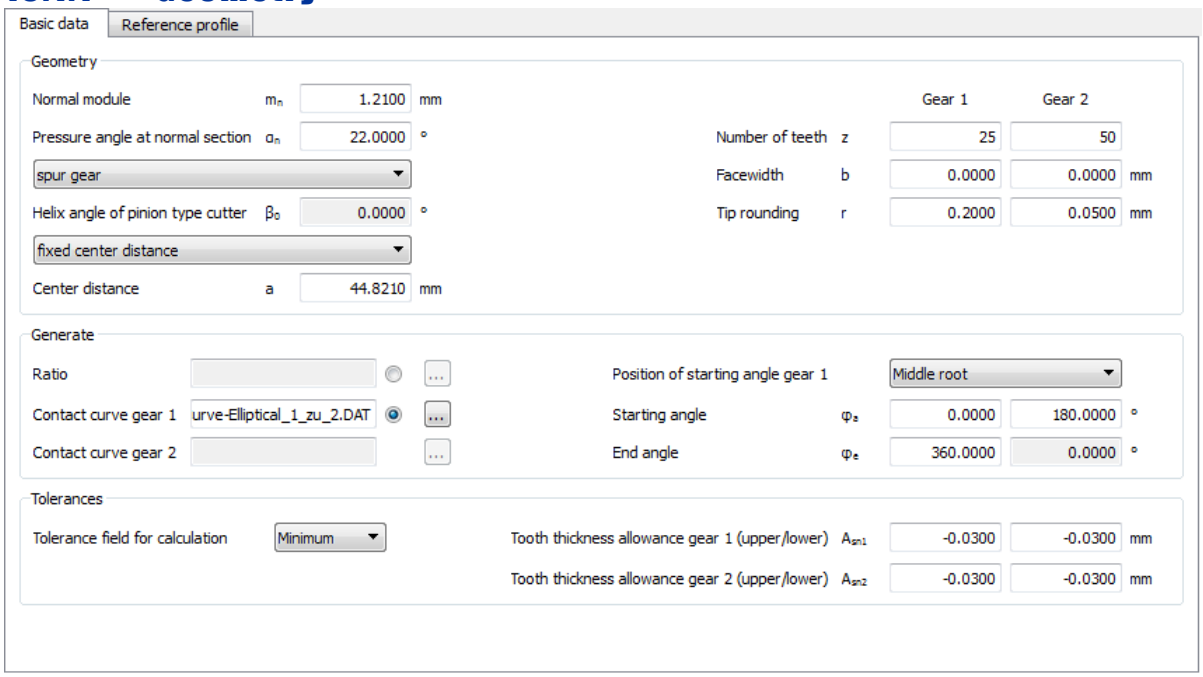


Figure 19.1: `Basic data` tab: entries for a noncircular gear pair

The module is defined from the "**Results window**" (total length of contact curve/[number of teeth* π]=module).

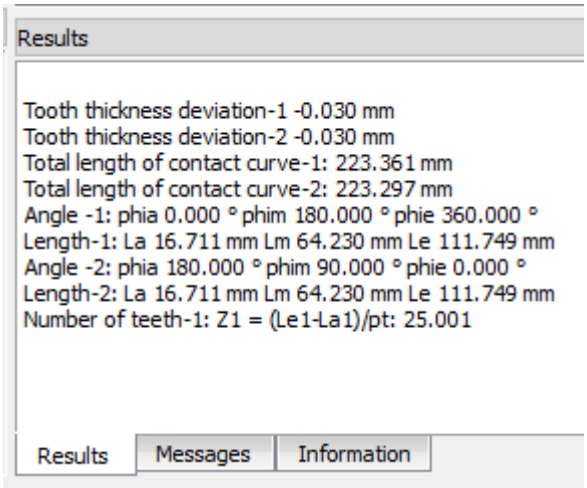


Figure 19.2: Results window

To save time in the first phase of the sizing process, we recommend you do not enter the total number of teeth z. We suggest you perform the calculation with a lower number of teeth (e.g. 2). In this case, although all the contact curves are calculated completely, only the specified number of teeth (2) are calculated and displayed.

Initially, start the calculation with a pressure angle in the normal section α_n of 20°. Later on you can change this angle instead of the profile shift or to optimize the tooth form.

19.1.1.1 Generate

The start and end angles φ_a and φ_e are important values because they determine the contact curve area of gear 1, i.e. the area that will be generated. In closed curves the angle φ_a is 0° and φ_e is 360°.

The contact curves or the ratio progression are then defined in files. The files must be in either "**dat**" or "**dxf**" format. These files can be stored in any directory. It is important to register these files correctly using the ... button.

Contact curves are also stored in the *.Z40 file. As a consequence, when you load a new calculation, you do not need to access the *.dat file. In this case you see a message to tell you the file cannot be found, and existing data will be used instead.

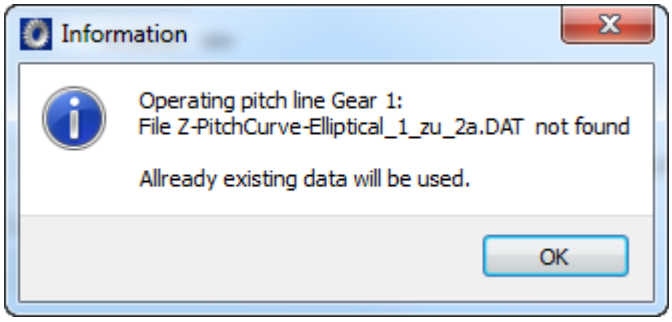


Figure 19.3: Message

NOTE

The progression (ratio or contact curve) must be defined from at least the starting angle to the end angle. To achieve clean intermeshing for the curve, the curve must have approximately 30° forward motion and follow-up movement. If the curve has no forward motion and/or follow-up movement, the software extends it automatically.

Input format for data in imported files

You can predefine one or two contact curves or the ratio progression. The imported files must have "dat" as their file extension.

A maximum of 7800 lines can be processed during noncircular gear calculation. Lines that start with # are comments and are ignored. To predefine the ratio progression, input the angle on gear 1 and the ratio.

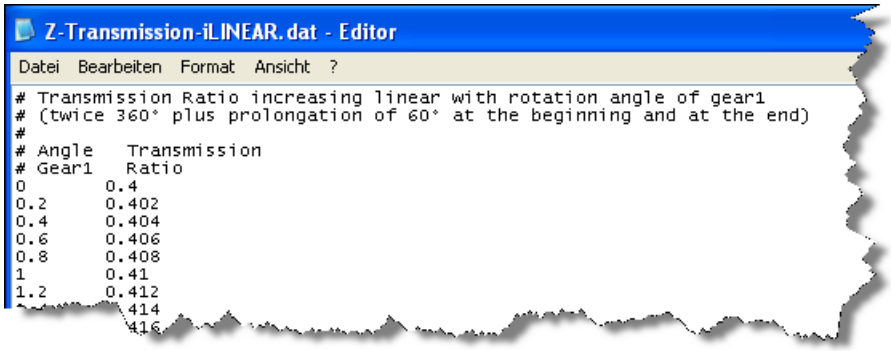


Figure 19.5: Example of ratio progression

To predefine the contact curve progression, input the radius and the angle.

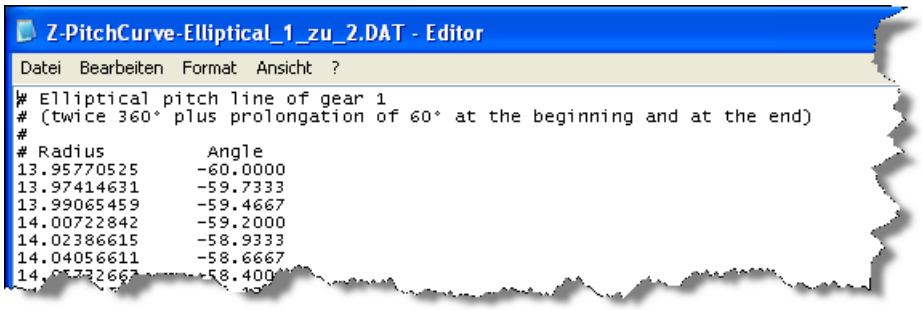


Figure 19.6: Example of a contact curve

19.1.2 Tolerances

We recommend you enter sufficiently large tooth thickness allowances A_{sn} (e.g. -0.10/-0.12 for module 2).

19.1.3 Reference profile

You must specify a topping pinion type cutter. The same pinion type cutter is usually defined for both Gear 1 and Gear 2.

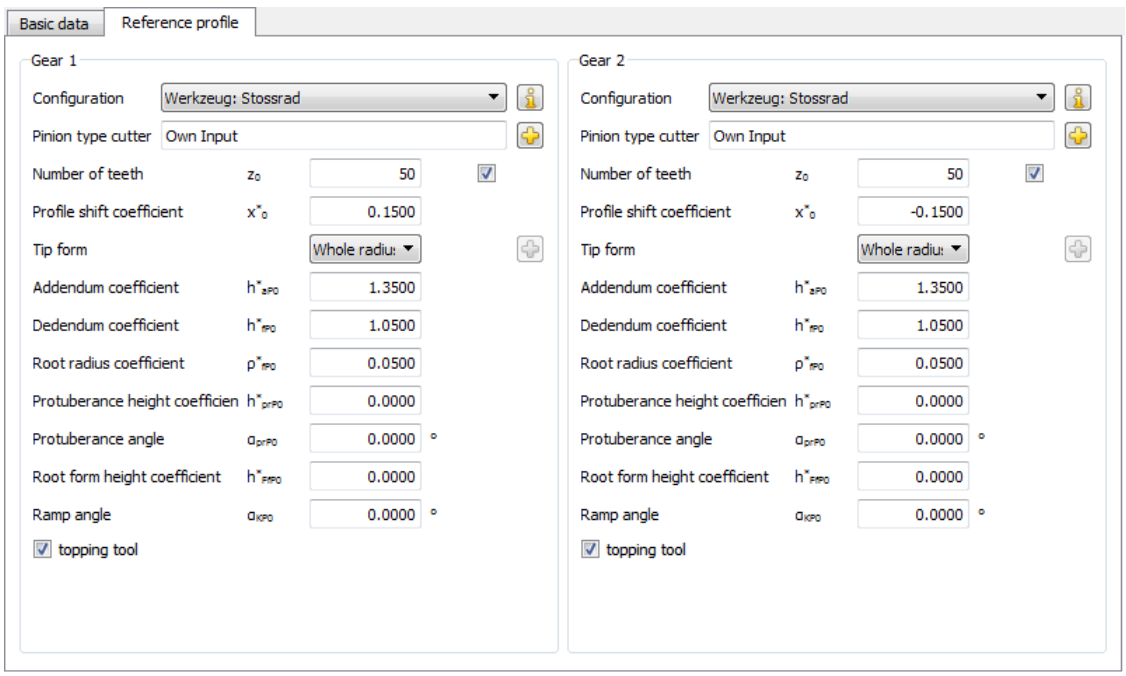


Figure 19.6: `Reference profile` tab: : entries for a noncircular gear pair

Problems may arise unless the profile shift coefficient of the pinion type cutter is set to 0. You must then carefully check exactly how the gears are generated.

19.2 How to use KISSsoft

19.2.1 Angle error

When you input a closed curve (gear 1), using a contact curve or gear reduction progression, it must start at 0° and finish at 360°. For this reason, the rotation of gear 2 must also be 360° (or a multiple of this). If not, this will result in an error.

Starting angle	φ_s	0.0000	0.0000	°
End angle	φ_e	210.0000	-183.2583	°

Figure 19.7: Minor error for Gear 2: φ_e is 179.9489 instead of 180°

However, this error has no effect because the predefined intermeshing allowance is large enough.

19.2.2 Checking the meshing

A useful way of checking the meshing is to change the number of rotation steps (per 360°) to rotate the gear in larger or smaller steps. You change the step sizes, as usual, in the Graphics window.

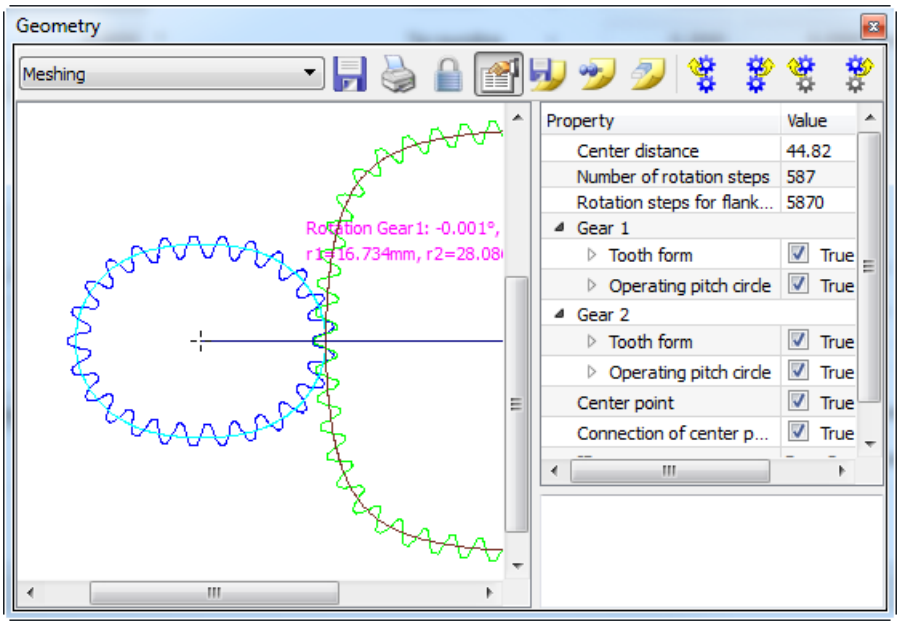


Figure 19.8: Changing the rotation steps

When you generate gears with allowances, we recommend you click the button to bring the gears into flank contact with each other.

NOTE

If, when you click the "`Rotate independently to the right`" button, the torsion exerted by one gear rotates against the other is too large (or not large enough) , you must adjust the number of "rotation steps" accordingly!

19.2.3 Improve tooth form

You can change the tooth form of circular gears quite significantly by changing the profile shift. In the current version of the program for noncircular gears, we recommend you set the profile shift coefficient of the pinion type cutter $x^*_0=0$. Despite this, you can still modify the tooth form by changing the pressure angle α_n.

19.2.4 Accuracy of the tooth form

Select `Calculations -> Settings` to predefine the accuracy (and therefore also the size of the file) for an IGES or DXF export.

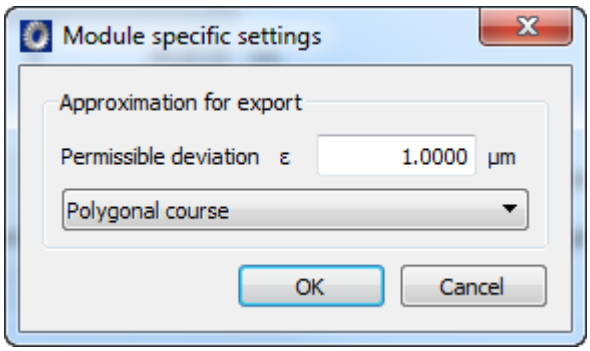


Figure 19.9: Module specific settings

This input only influences IGES or DXF files.

In the program, the tooth form (for each flank) is calculated with 100 points. You will find these results in the TMP files (and in the report). If you want to modify the number of internally calculated points, simply change the corresponding entry in the *.Z40 file:

Go to a saved *Z40. file and search for the lines:

ZSnc.AnzPunkteProFlanke=100;

and enter, for example, 40 instead of 100. As a result, only 40 points per flank will be calculated.

19.2.5 Export individual teeth

Go to a saved *Z40. file and search for the lines:

ZRnc[0].AusgabeKontur=0, for Gear 1 or

ZRnc[1].AusgabeKontur=0, for Gear 2.

There, change the variable to the required value, e.g. ZRnc[0].outputcontour=3.

The LEFT flank of the x-th tooth space (therefore the 3rd gap of Gear 1 in the example) is always output.

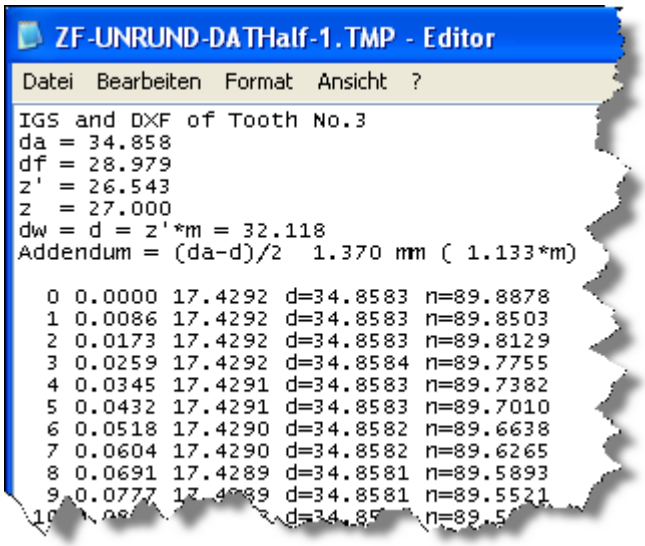


Figure 19.10: Temporary file for exporting teeth (ZRnc[0].outputcontour=3, for Gear 1)

19.2.6 Report

If you select (Detailed) in `Report settings` this report will also be very extensive. If you want a shorter version, set "Extent of data" to 5 (standard).

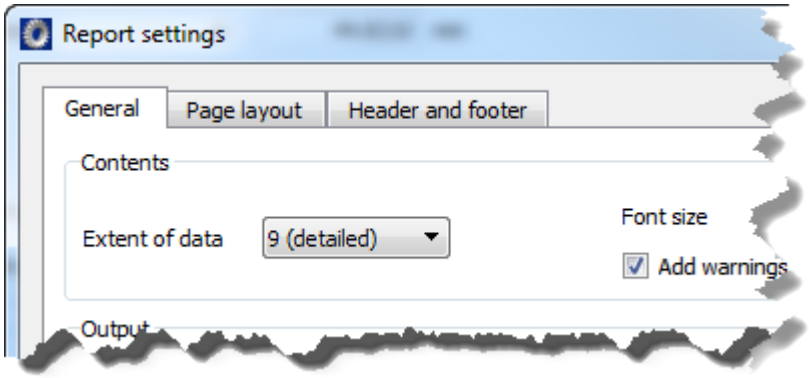


Figure 19.11: Report settings with a changed amount of data for output to a report

19.2.7 Temporary files

When a calculation is performed, KISSsoft automatically generates temporary files. The directory in which these files are generated by KISSsoft must be specified in KISS.ini in the "Path" section. You will find KISS.ini in the KISSsoft main directory. Before changing the default setting you must ensure that you have read and write permissions for the changed directory. You will find more detailed information in Section 2 of the manual, "Setting Up KISSsoft".

ZF-H1_Gear 1 (step 1).tmp: ZF-H1_Gear 2 (step 1).tmp:	Insignificant, contains information about generating the pinion type cutter (cutter/tool)
ZF-H1_Gear 1 (step 2).tmp: ZF-H1_Gear 2 (step 2).tmp:	Not important information: contains details, flank for flank, about generating the noncircular gear
ZF-UNRUND-1.TMP:	Contains interesting information about contact curve 1; defining pitch points on contact curve 1 calculating contact curve 2 from contact curve 1 contact curve lengths documentation about the intermeshing (individual points) of noncircular gear 1 with X, Y, normal, diameter and angle.
ZF-UNRUND-2.TMP:	Contains interesting information Documentation about the intermeshing (individual points) of noncircular gear 2 with X, Y, normal, diameter and angle
ZF-UNRUND-DAT-1.TMP: ZF-UNRUND-DAT-2.TMP:	Possible further uses of the intermeshing (individual points) X,Y coordinates
ZF-UNRUND-OPLINE-1.TMP:	Possible further uses of the intermeshing (individual

ZF-UNRUND-OPLINE-2.TMP:	points) X,Y coordinates
Z-WalzKurve-1.TMP: Z-WalzKurve-2.TMP:	Possible further uses of the contact curve (individual points) r, ϕ -coordinates (*); the format corresponds exactly to the format of the DAT file (see "Import format" section)
Z-OpPitchPoints-1.TMP: Z-OpPitchPoints-2.TMP:	Possible further uses of pitch points on each tooth in r, ϕ coordinates

Chapter 20
Report menu

20.1 Drawing data

To display the toothing data you require to add to a drawing, select `Drawing data`. Use the `Z10GEAR1?.RPT` file (for Gear 1), and the `Z10GEAR2?.RPT` file (for Gear 2), etc. (? = d/e/f/i/s for the required language) to modify the template to your own requirements.

All the angle data for the user-specific Z10GEAR1?.rpt to Z10GEAR4?.rpt reports is given in degrees-minutes-seconds, and displayed in brackets after the decimal point.

For example the number 20.3529° is displayed as:

20° 21' 10" (20.3529)

20.2 Manufacturing tolerances

Click on the `Manufacture tolerances` menu item to generate a report that displays all the manufacturing tolerances as defined in the ISO 1328, DIN 3961, AGMA 2000, AGMA 2015 and BS 436 standards.

You will find notes on how to calculate tolerances as defined in ISO 1328 and DIN 3961

This calculation is performed in accordance with the formulae documented in the standard. As specified in the standard, these formulae use the geometric average of the edge values in the corresponding area instead of the effective geometric values (such as module, reference circle, etc.).

Example:

As specified in ISO1328-1:1995 the following area limits are predefined for the module m_n:

0.5 / 2 / 3.5 / 6 / 10 / 16 / 25 / 40 / 70 mm

For module m_n = 2.1, the equations use the geometric mean value of the range limits 2 and 3.5 instead of the ratio 2.1. Therefore sqrt(2*3.5)=2.645.

Exceptions (the standards do not provide any notes about exceptions):

- If the size is less than the lowest range limit, the lower range limit is used as the value if the value >= 0.8*range limit, otherwise, the effective value is used.

- If the size is greater than the upper range limit, the effective value is used.

Example:

Module m_n = 85 -> 85 is used.

Module m_n = 0.44 -> 0.5 is used (0.8*0.5 = 0.4 is less than 0.44).

Module m_n = 0.31 -> 0.31 is used.

Note:

In some circumstances, calculations performed in accordance with the formulae in ISO 1328 may return data that is slightly different from the tables specified in the standard. We have also discovered that the formulae in DIN 3961 may return slightly different values from the data given in the tables in DIN 3962 and 3963. However, the majority of the variations are at most 1µm.

20.3 Summary

You use the summary function to compare the current toothing with the results of `fine sizing`.

20.4 Service life

This report shows the most important data that is used to calculate service life either with or without a load spectrum (see page II-287). You can also call the service life calculation by clicking the Sizing button next to the Service life input field. This then displays the service life that should be achieved if required safeties are used.

20.5 Sizing of torque

The sizing of torque displays the most important data required to calculate the transmissible torque (or the maximum transmissible power) with or without load spectrum. You can also call the sizing of torque function directly by clicking the checkbox next to the Torque or Power input fields. You then see a value for the torque that should be achieved if required safeties are used.

20.6 Proposal for the hardening depth EHT

A wide range of different proposals for the hardening depth EHT as specified in the standards have been documented. The data specified in the ISO, AGMA and Niemann standards are often very different, because of the very rough approximations involved. The most accurate calculation, which uses the shear stress criterion from the Hertzian law to define the required hardening depth, is documented in the upper part of the report. You can also specify the safety factor which is to be used for the calculation (safety factor for calculating shear stress for EHT (see page II-439)). You will see this displayed as a graphic in the "Hardening depth" section.

Chapter 21
Graphics menu

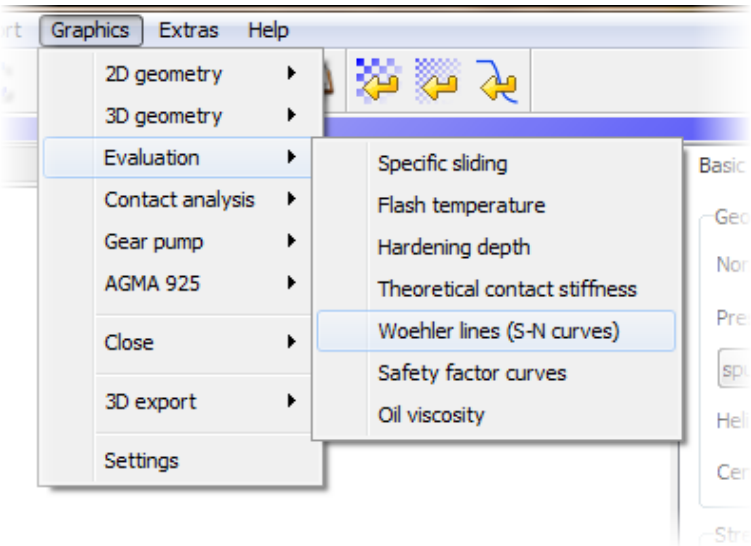


Figure 21.1: `Graphics` menu in the KISSsoft menu bar -User interface

In the `Graphics` menu you can select various menu items to help you display toothing and functional processes.

NOTE

In the Graphics window, hold down the left-hand mouse button and move the mouse to select the range of values you want to increase. Click the right-hand mouse button to open a context menu that contains other zoom functions.

Table 21.1 shows which of the options in the `Graphics` menu are supported by particular tooth calculation modules and where you can find the relevant documentation in this section.

Menu item	Options	Section										
AGMA 925	Temperature in contact	21.1.1		✔	✔	✔	✔	✔				

	Thickness of lubrication film			✔	✔	✔	✔	✔					
	Hertzian pressure			✔	✔	✔	✔	✔					
	Specific thickness of film			✔	✔	✔	✔	✔					
Evaluation	Specific sliding	21.4.1		✔	✔	✔	✔	✔	✔				
	Flash temperature	21.4.2		✔	✔	✔	✔	✔					
	Hardening depth	21.4.3		✔	✔	✔	✔	✔	✔				
	Theoretical contact stiffness	21.4.7		✔		✔	✔	✔					
	S-N curve (Woehler line)	21.4.4		✔	✔	✔	✔	✔					
	Safety factor curves	21.4.5		✔	✔	✔	✔	✔					
	Stress curve	21.5.9								✔			
	Path of contact (pinion/face gear)	21.4.8								✔			
	Safety scuffing	21.4.10								✔			
	Sliding velocity	21.4.10								✔			
	Oil viscosity	21.4.6	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	
Contact analysis	Axis alignment	21.5.1 (see page II-596)	✔	✔	✔	✔	✔	✔					
	Specific sliding	21.5.8	✔	✔	✔	✔	✔	✔					
	Transmission error	21.5.2		✔	✔	✔	✔	✔					
	Transmission error acceleration	21.5.3		✔	✔	✔	✔	✔					
	FFT of the transmission error	21.5.4		✔	✔	✔	✔	✔					
	Normal force curve (line load)	21.5.5		✔	✔	✔	✔	✔					
	Normal force distribution (line load)	21.5.5		✔	✔	✔	✔	✔					
	Torque progression	21.5.6		✔	✔	✔	✔	✔					

	Stiffness curve	21.5.7		✔	✔	✔	✔	✔					
	FFT of the contact stiffness	21.5.8		✔	✔	✔	✔	✔					
	Bearing force curve	21.5.9		✔	✔	✔	✔	✔					
	Bearing force curve in %	21.5.9		✔	✔	✔	✔	✔					
	Direction of the bearing forces			✔	✔	✔	✔	✔					
	Kinematics	21.5.10	✔	✔	✔	✔	✔	✔					
	Specific sliding per gear	21.5.11	✔	✔	✔	✔	✔	✔					
	Specific power loss	21.5.12		✔	✔	✔	✔	✔					
	Heat development	21.5.13		✔	✔	✔	✔	✔					
	Heat development along the tooth flank	21.5.13		✔	✔	✔	✔	✔					
	Flash temperature	21.5.15		✔	✔	✔	✔	✔					
	Lubricating film	21.5.16		✔	✔	✔	✔	✔					
	Specific thickness of film	21.5.16		✔	✔	✔	✔	✔					
	Safety against micropitting	21.5.16		✔	✔	✔	✔	✔					
	Stress curve	21.5.14		✔	✔	✔	✔	✔					
	Bending stress in root area	21.5.14		✔	✔	✔	✔	✔					
	Stress distribution on tooth	21.5.14		✔	✔	✔	✔	✔					
	Wear along the tooth flank	21.5.17		✔	✔	✔	✔	✔					
2D geometry	Meshing	21.2.4		✔	✔	✔	✔	✔	✔	✔	✔	✔	✔
	Tooth form	21.2.1	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔
	Tool	21.2.2	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔
	Manufacture	21.2.3	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔
	Profile diagram	21.2.5	✔	✔	✔	✔	✔	✔				✔	
	Tooth trace diagram	21.2.5	✔	✔	✔	✔	✔	✔				✔	
	Flank curvature radii	21.2.6	✔	✔	✔	✔	✔	✔					
	Angle of flank normal	21.2.7	✔	✔	✔	✔	✔	✔					
	Drawing	21.2.8	✔	✔	✔	✔	✔	✔	✔				
	Assembly	21.2.9		✔		✔	✔	✔	✔		✔		

3D geometry	Tooth system	21.3.1		✔	✔	✔	✔	✔	✔	✔	✔	✔	
	Tooth form	21.3.2	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	
Graphics list		Graphics list	✔	✔	✔	✔	✔	✔	✔	✔	✔	✔	
Manufacturing drawing		Manufacturing drawing	✔	✔	✔	✔	✔	✔					

Table21.1: Graphics menu in the KISSsoft interface menu bar.

- Single gear, - Cylindrical gear pair, - Pinion with rack, - Planetary gear, - Three gears, - Four gears, - Bevel and Hypoid gears, - Face gears, - Worms with double enveloping worm wheels, - Crossed helical gears and precision mechanics worms, - Splines (Geometry and Strength)

21.1 AGMA 925

21.1.1 Thickness of lubrication film and specific oil film thickness

The thickness of lubrication film h_e in accordance with AGMA 925 is shown over the meshing cycle. Another figure shows the specific density of film λ, which is a critical value for evaluating the risk of micropitting. λ is the ratio of the thickness of lubrication film to the surface roughness, expressed in simple terms.

21.2 2D geometry

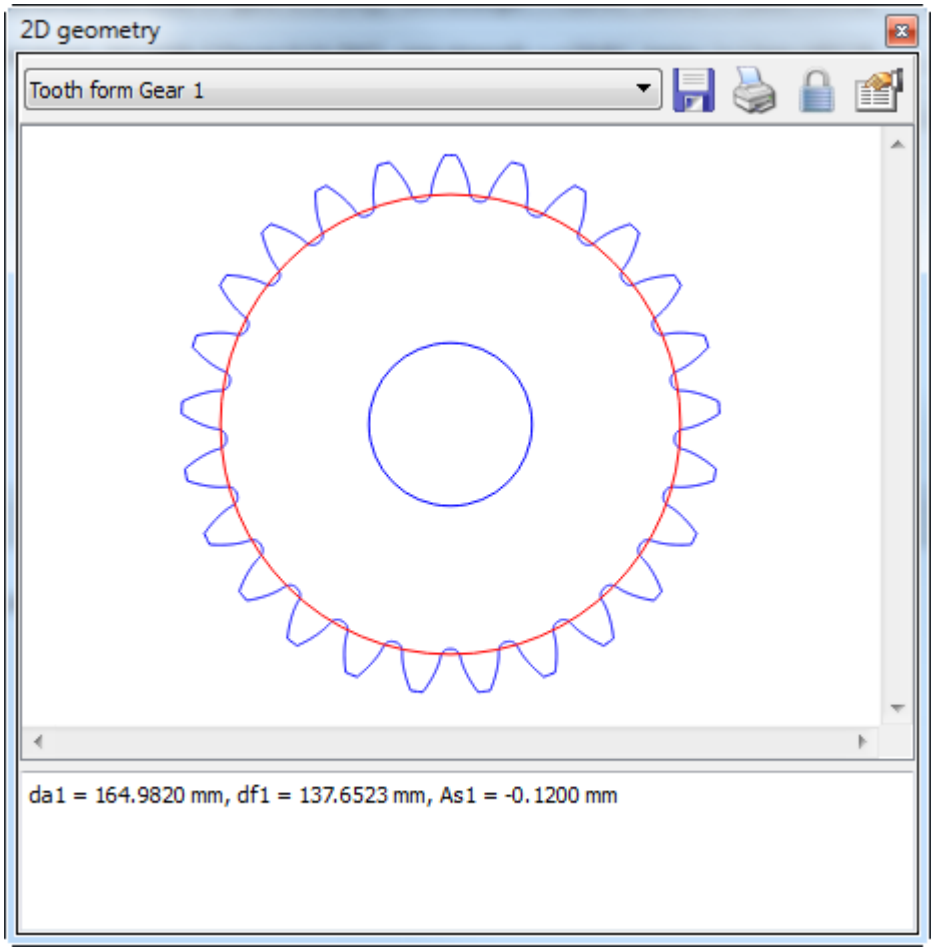


Figure 21.2: Graphics window: `Geometry`

You can select a number of different output options from the drop-down list in the tool bar of the `Geometry 3D` graphics window (see Figure 21.2:

21.2.1 Gear tooth forms

Display a gear tooth form.

NOTE:

Click the Property button above the graphic to specify the number of teeth that are to be displayed. You can select whether to display it in transverse section, normal

section or axial section. Selecting the "Half tooth for export" option is also very useful if you want to export the tooth form and reimport it into KISSsoft later on.

21.2.2 Gear tool

This displays the tool associated with the gear, if one is present.

21.2.3 Manufacturing a gear

Display the pairing: gear with cutter. Here the gear is shown in blue and the cutter in green.

21.2.4 Meshing

Displays the meshing of two gears.

NOTE ABOUT FACE GEARS:

In KISSsoft, the face gear is calculated by simulating the manufacturing process in different sections. You can display different sections at the same time. To do this, go to the `Property browser (PB)` in the graphics window and set the property in the `section` you require to `True` (see Figure 21.3).

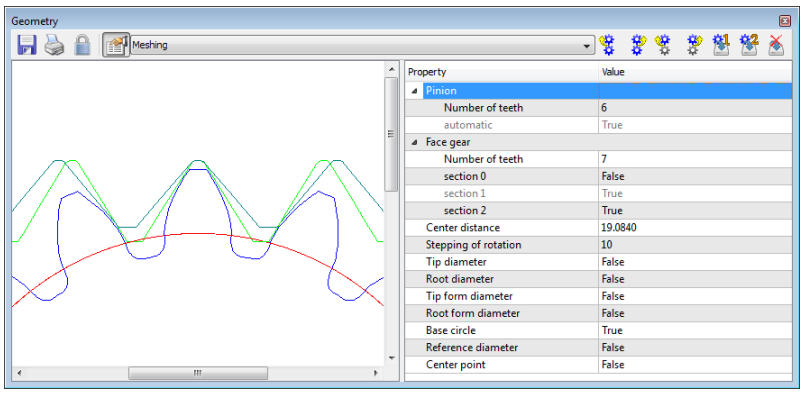


Figure 21.3: Graphics window: `Meshing` with `Property Browser`

The difference between the theory and the effective tooth form means that the tooth has an undercut! You can see this more clearly in the 2D view.

Collision check:

When generating two gears (in the graphical display) you can activate the collision display option. In the graphic, this shows (with squares) the points where the gears touch or where collisions may occur.

shown in brown: touch (between 0.005 * module distance and 0.001 * module penetration)

shown in red: collision (greater than 0.001 * module penetration)

The system identifies and marks collisions in all the meshing teeth. This option is particularly useful for analyzing the generation of non-involute tooth forms or measured tooth forms (using a 3D measuring machine) with a theoretical single flank generation check.

This functionality is also available for cylindrical gears and worm gears (but with restrictions for worm gears (see page II-549)).

21.2.5 Profile and tooth trace diagram

These diagrams are generated by placing two lines diagonally over the tolerance band, as described in ANSI/AGMA: 2000-A88 (figures 1 and 2).

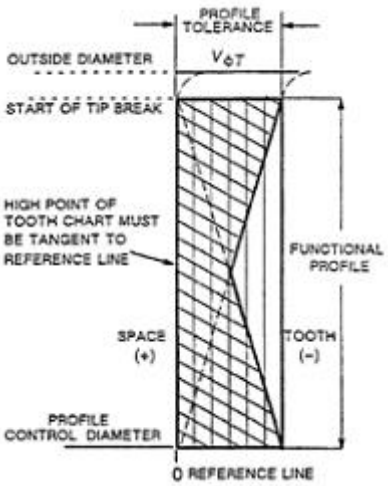


Figure 1 Profile diagram

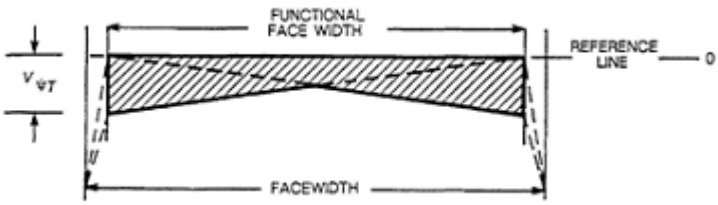


Figure 2: Tooth trace diagram

In the figures shown above, $V_{\phi T}$ is the profile tolerance and $V_{\psi T}$ is the tooth alignment tolerance which correspond to the profile total deviation (F_{α}) and the tooth helix deviation (F_{β}) as detailed in ISO 1328-1.

Although every company has its own method of creating profile and tooth trace diagrams, the AGMA method is recognized as the standard in the industry. ISO TR 10064-1 (and ISO FDIS 21771) also include a general description of profile and tooth trace diagrams, but without any explanations about the construction method.

In KISSsoft the profile and tooth trace modifications are defined in the `Modifications` tab. The corresponding diagrams are then generated using this data.

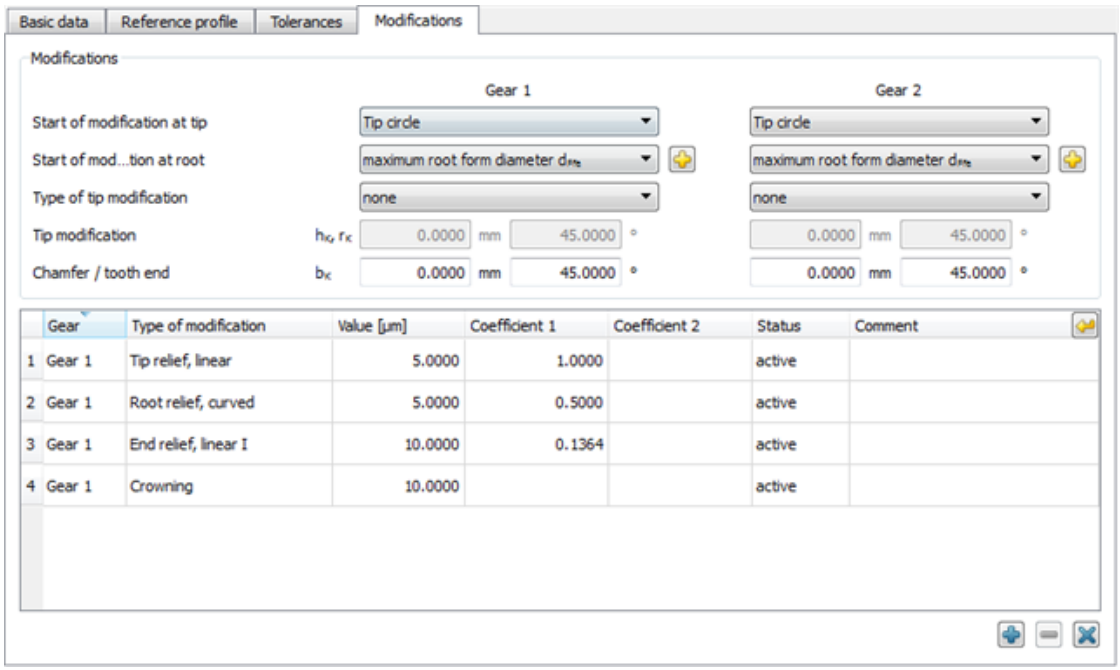


Figure 21.4: `Modifications` tab with modifications

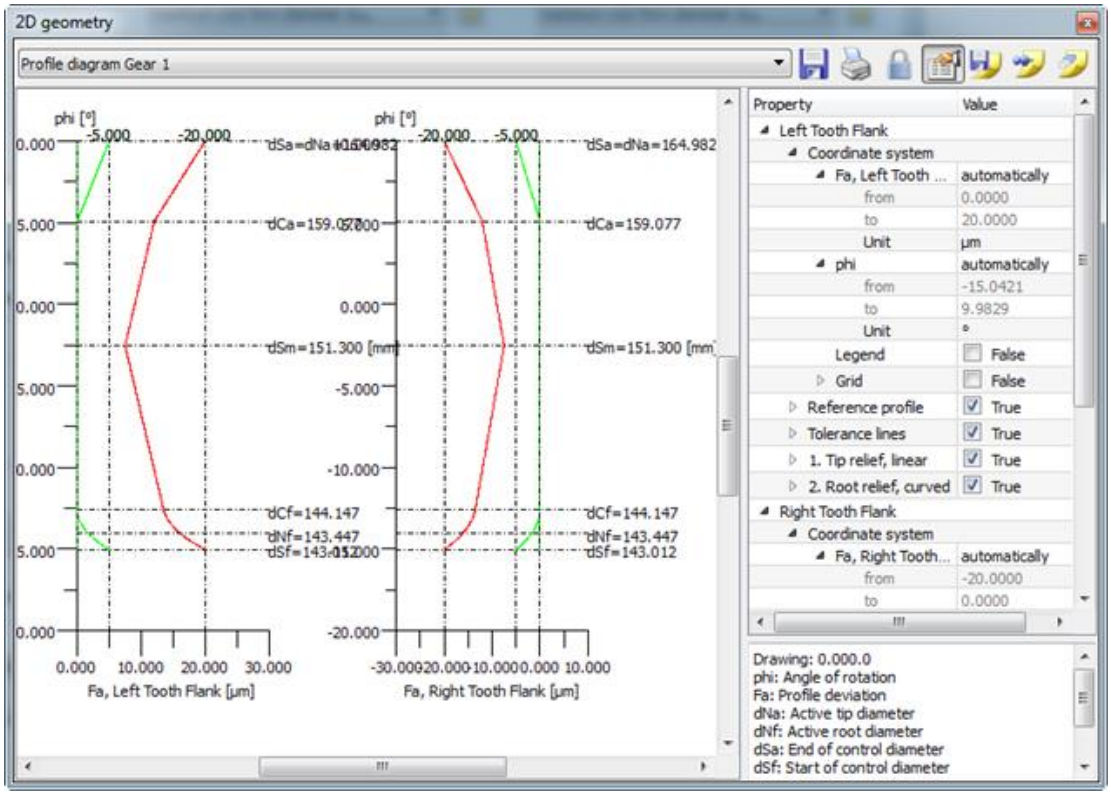


Figure 21.5: Profile diagram for gear 1 in accordance with the predefined modifications

The horizontal axis of the profile diagram shows the profile deviation values and the vertical axis shows the coordinates along the profile. You can select different values for the left-hand vertical axis (roll angle or path of contact length) (`Calculation→Settings→Contact analysis`). The values for the right-hand flank are always given as the diameter. You can also specify the tolerance type by clicking on `Calculation→Settings→General`. If you select the tolerance band type as specified in AGMA 20000-A88, the diagrams are construed in accordance with the method mentioned above. If you set the tolerance band type to constant, the tolerance remains constant along the length or the width of the tooth flank. Click on the "Display profile in the middle of the tolerance band" checkbox to specify whether the central profile (see below) should usually be displayed.

Describe the specific diameter of the right-hand vertical flank:

- dSa: end diameter of the modifications (starting diameter of the modifications at the tip)

- dSf: starting diameter of the modifications (starting diameter of the modifications at the root)
- dCa: active tip diameter (starting diameter of the modification)
- dCf: tip form circle diameter (starting diameter of the modification)
- dCm: center point of the functional profile measured along the path of contact

NOTE:

The profile diagram is in the middle of the facewidth. The **Twist** profile modification is not possible.

Show curves in the diagram:

- green curve: Modifications of "1. Tip relief, linear" and "2. Tip relief, arc-like"
- blue curve: Reference profile (current function profile used for checking and generated from the total of the modified curves)
- red line: Tolerance curve generated by subtracting the profile total deviation from the reference profile. The profile deviation values are listed in the main report.
- green line (middle): Central profile, which can be entered as the target value for processing because it lies in the middle between the reference profile and the tolerance curve.
- gray lines: Tolerance range, which shows the range (as a crosshatched area) in which the actual manufacturing profile can lie.

The manufacturing profile (with tolerance) should lie between the tolerance curve and the reference profile.

You can use the properties to change the colors of the lines or to display or hide the individual curves.

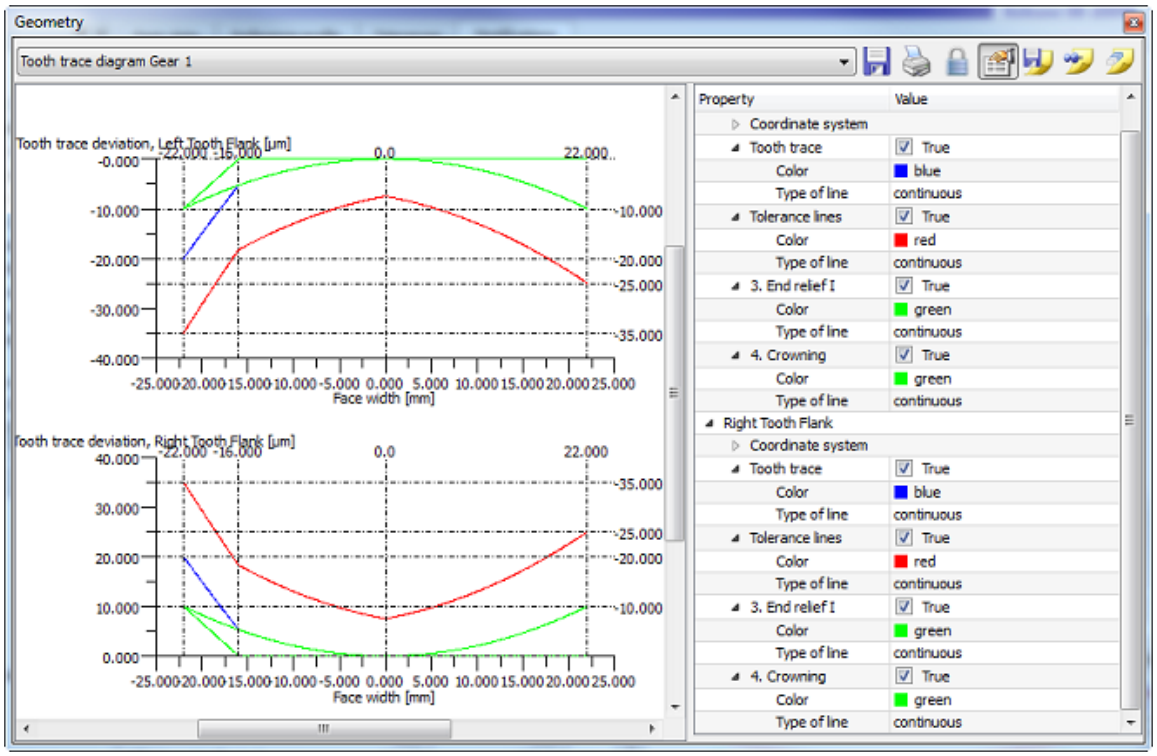


Figure 21.6: Tooth trace diagram for gear 1 with the predefined modifications

In the figure, the reference profile is shown in blue and the tolerance line is shown in red. The horizontal axis shows the coordinates along the tooth trace (facewidth) and the vertical axis shows the flank allowance as specified in the usual industrial conventions. The value of the total tooth trace deviation Fb is shown in the main report.

The manufacturing tooth trace (with tolerances) should lie between the tolerance curve and the reference tooth trace.

21.2.6 Flank curvature radii

In this graphic you see the flank curvature radii along the tooth flank. Along with the normal force, these are critical values for Hertzian pressure.

21.2.7 Angle of flank normal

The normal angle to the flank is shown in this graphic. Every point on the tooth form has a normal.

21.2.8 Drawing

Use this menu to display gears in diagram form. The gears are shown in transverse and axial section.

This option is primarily used for bevel gears and worms.

21.2.9 Assembly

Use this menu to create a diagram of how gears are assembled. The buildup (pair) of the gears is shown in transverse and axial section.

Two views, section and overview, are given for bevel gears with a shaft angle of 90°. For shaft angles <>90° only the section of the bevel gear pair is displayed.

21.2.10 Manufacturing drawing

21.2.10.1 General

Manufacturing drawings are designed to display a number of graphics on the same surface and therefore create a print-ready image that can be used to manufacture a gear. You can also display the drawing data report at the same time. Use a control file to tailor the display to suit specific requirements. The control file is stored in the template directory (usually under KISSDIR\template). It has the module name and the file extension .grc (e.g. Z012gear1.grc).

You can also save the graphic generated here as a .dxf file in the usual way.

21.2.10.2 Editing the control file

By making changes to the control file you can modify the manufacturing drawing to suit your own requirements. The commands used to control the manufacturing drawing are described in the following table.

papersize: A4 papersize: A4 portrait	Specifies the required paper format. This refers to the standard terms used to describe commonly used paper sizes (A3, A4, A5, B4, B5, Letter, Legal and Ledger), and also enables you to input your own dimensions for width and

papersize: 297, 210	height. The default setting is for landscape format. However, you can switch to portrait format by entering the key word "portrait".
fontsize: 5	Specifies the required font size. The font size influences the size of the report and also th diagram titles.
units: inch	The default setting is that input values are assumed to be in mm. The system can handle these units: inch, mm and cm.

You can now add graphics that have specific characteristics. The table below gives an overview of the correct inputs.

draw 2DDiaProfileChart1	"draw" is the key word used to specify that a graphic is to be added. It is followed by the ID of the graphic you want to insert. The number at the end i part of the ID and refers to the gear.
window: 160, 285, 0, 85	"window" identifies the window in which the graphic is displayed. The values show the limits on the left, right, bottom and top.
scaletofit	This optional command forces the graphic to distort so that it fills the windov in every direction. We recommend you do this for diagrams, but not for geometric figures. If this term is not used, the original size and shape of the graphic is retained.

You can insert these graphics:

Tooth form	2DGeoToothDrawing
Drawing	2DGeoGearDrawing
Assembly	2DGeoAssemblyDrawing
Tool	2DGeoToolDrawing
Profile diagram	2DDiaProfileChart
Tooth trace diagram	2DDiaFlankLineChart
Angle of flank normal	2DDiaNormal

Finally, you can now display the report in the required location:

write report1	"write" is the key word used to create a gear data report. Enter report1 to select the gear data of gear 1, report2 to select the gear data from gear 2 etc.
topright: 297, 218	Unlike graphics, you must specify an alignment here. You define this with the first word. The correct commands are topright:, topleft:, bottomright: and bottomleft:. They represent the alignment (top right, top left, bottom right and bottom left). The next two values represent the particular reference point.

21.3 3D geometry

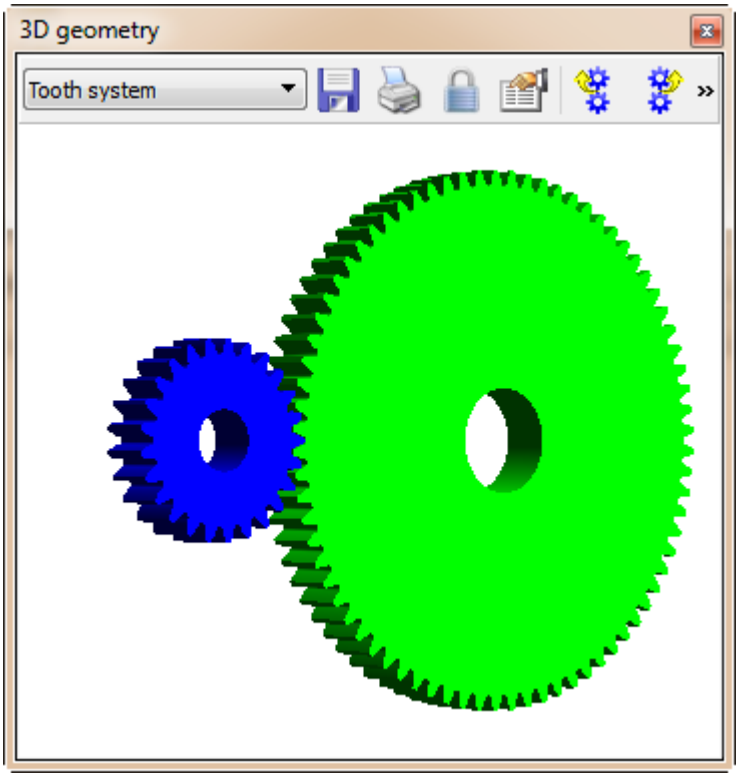


Figure 21.3: Graphics window `Tooth system`

The gears are displayed in the 3D Parasolid viewer.

You can select a number of different output options from the drop-down list in the tool bar of the `Geometry 3D` graphics window (see Figure 21.3). You can store the Parasolid viewer graphics in different file formats such as:

- Windows Bitmap (*.bmp)
- Joint Photographic Experts Group (*.jpg, *.jpeg)
- Portable Network Graphics (*.png)
- Standard for the Exchange of Product Model Data (*.stp, *.step)
- Parasolid Text File Format (*.x_t)
- Parasolid Binary File Format (*.x_b)

21.3.1 Tooth system

The tooth system displays the assembled system of gears in 3D.

You can display these gears in different views.

21.3.2 Tooth form

In the `Tooth form` menu, an individual gear is shown in 3D in the Parasolid viewer. There are the following restrictions on how these gears are generated. Only forms ZI and ZA can be generated for worm gears.

21.4 Evaluation

21.4.1 Specific sliding

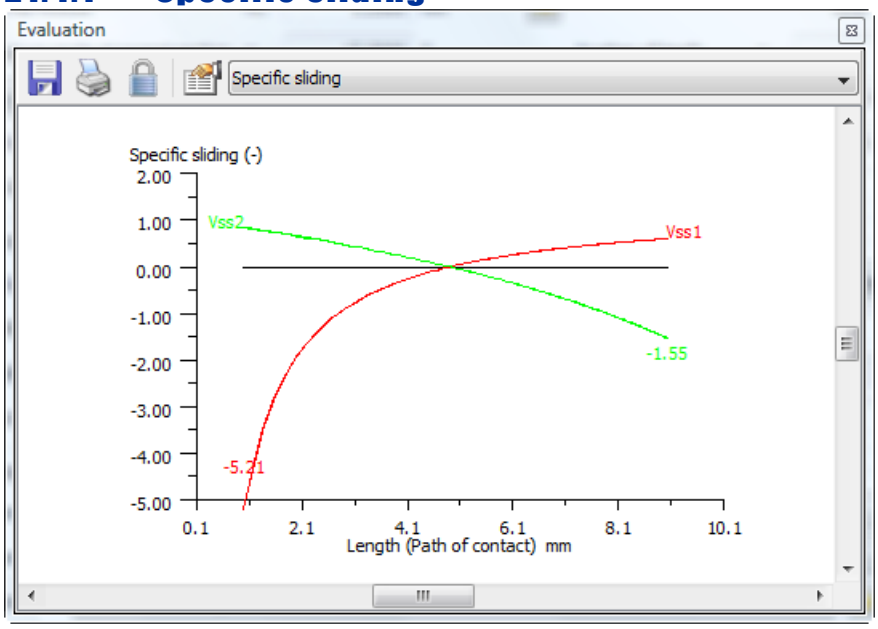


Figure 21.4: Display of `Specific sliding` in the `Evaluation` graphics window

The graphic shows the progression of specific sliding (ratio between the sliding velocity and the tangential speed) for the pinion and the gear over the length of path of contact. This takes into account two situations: maximum tooth thickness - minimum center distance and minimum tooth thickness - maximum center distance.

When you specify the profile shift (see section "Profile shift coefficient" on page II-253), click the button to see a suggested value for balanced specific sliding.

21.4.2 Flash temperature

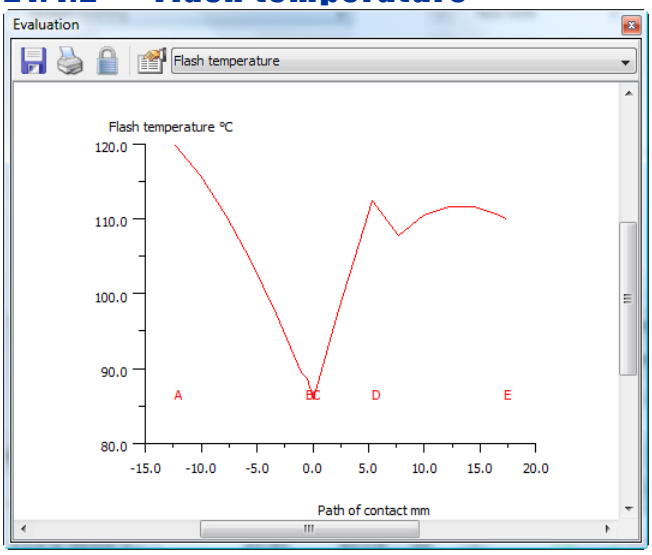


Figure 21.5: `Flash temperature` option in the `Evaluation` graphics window

The flash temperature is the local temperature on the tooth flank at the moment of contact is displayed over the meshing cycle. The point that has the highest temperature can be seen. Therefore it can be decided which action (i.e. a profile modification) can be taken to reduce this value.

21.4.3 Hardening depth

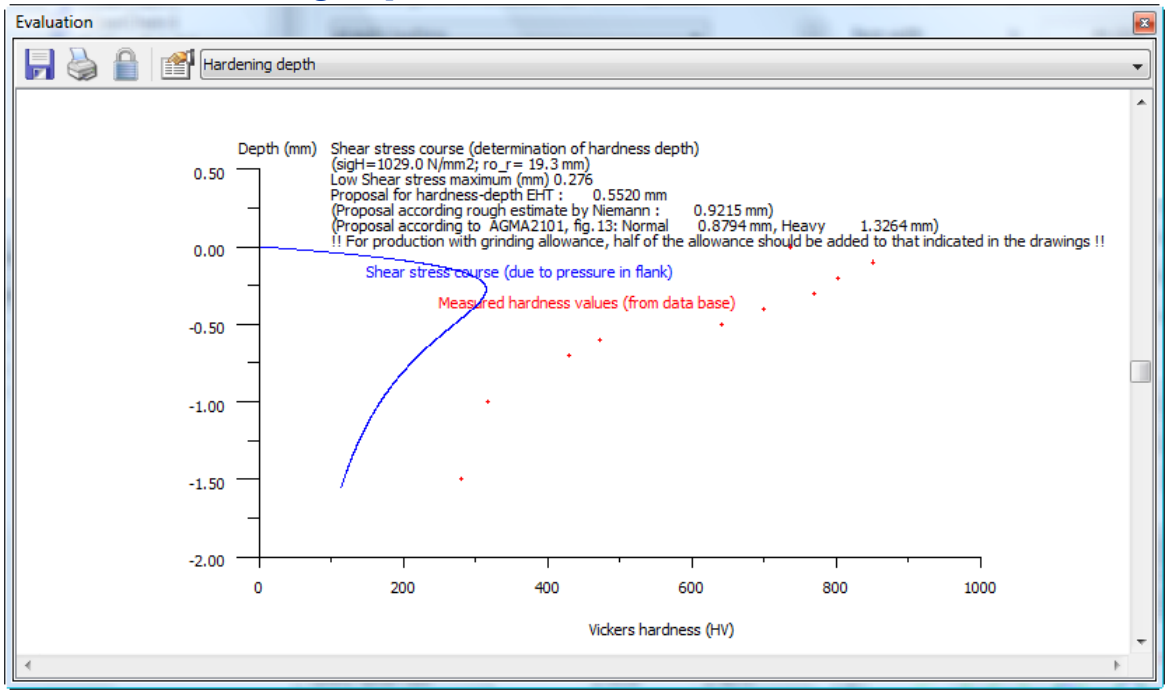


Figure 21.6: `Hardening depth` option in the `Evaluation` graphics window

This calculates the optimum hardening depth (for case hardened or nitrided gears). It shows the stress progression in the depth vertical to the flank surface. This value is displayed directly in the HV values, because HV or HRC values are always used when specifying hardening depth and hardening measurements. If the materials database already contains values for a measured hardening progression, the hardening progression is displayed, accompanied by a warning message if the existing hardening is insufficient.

Proposed values for the recommended hardening depth are displayed in a special report, classified by calculation method, selected material and heat treatment process.

The various different methods are:

- The shear stress progression in the depth of the gear pair is calculated in accordance with Hertzian law. The shear stress is multiplied by a safety factor (enter this under "Settings". The default setting is 1.63). This defines the depth of the maximum shear stress (hmax). The program suggests the value 2*hmax as the hardening depth (EHT).

- For each individual gear in accordance with the proposals given in Niemann/Winter, Vol.II [65] (page 188)
- For each individual gear in accordance with the proposals given in AGMA 2101-D [1] (pages 32-34)
- For each individual gear in accordance with the proposals given ISO 6336 Part 5 [44] (pages 21-23) (to avoid pitting and breaking up of the hard surface layer)

21.4.4 Woehler line for material

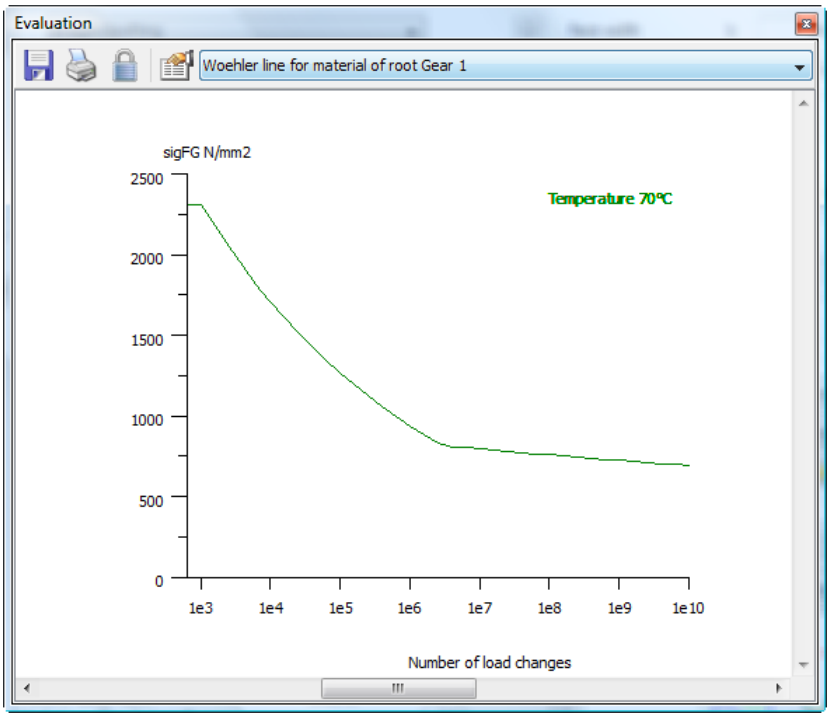


Figure 21.7: `Woehler line` option in the `Evaluation` graphics window.

Displays the Woehler line for the tooth root and flank. This calculation is performed in accordance with the selected calculation standard.

21.4.5 Safety factor curves

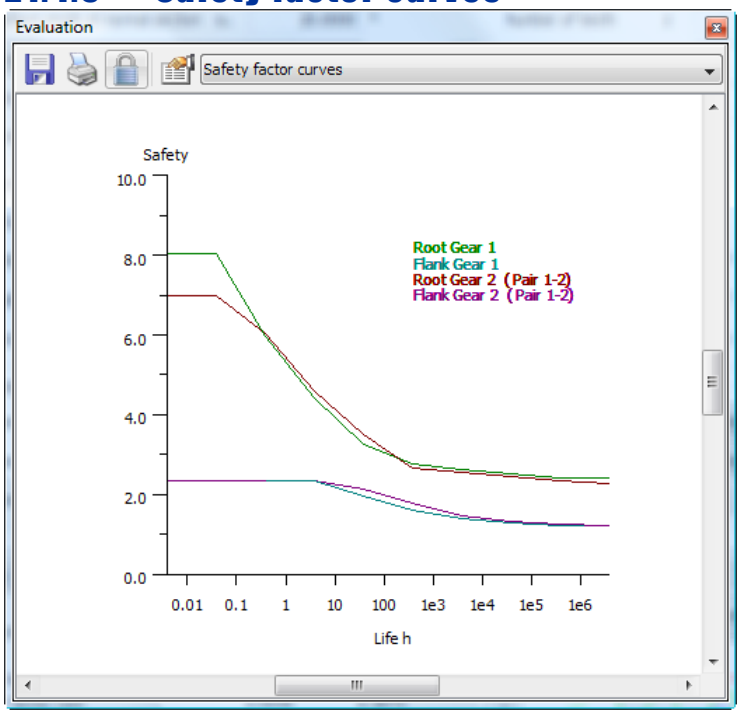


Figure 21.8: `Safety factor curves` option in the `Evaluation` graphics window

The graphic displays the progression of safety depending on the service life.

21.4.6 Oil viscosity, depending on temperature

This displays the course of kinematic viscosity over the operating temperature range of the oil.

21.4.7 Theoretical contact stiffness

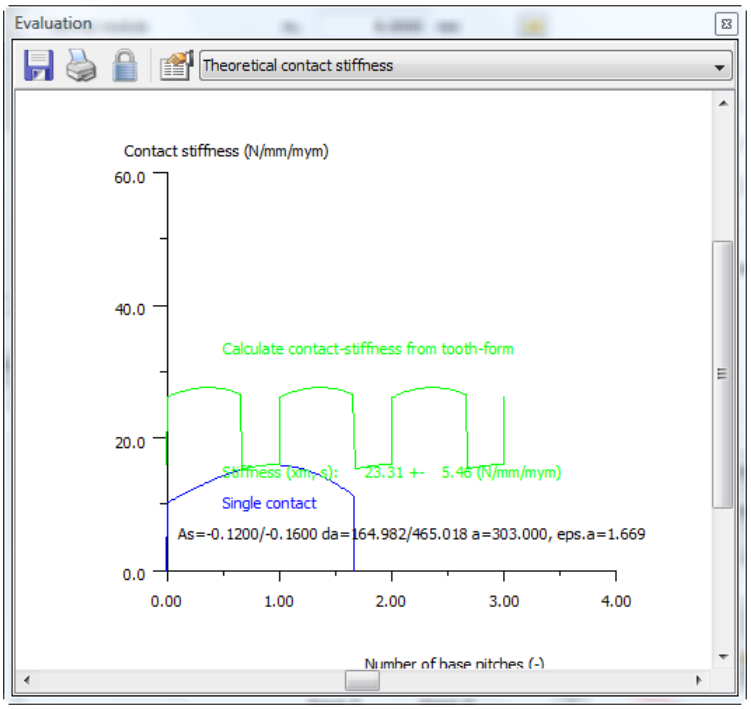


Figure 21.9: `Theoretical contact stiffness` option in the `Evaluation` graphics window

Displays the contact stiffness as a graphic. The contact stiffness is calculated on the basis of the real tooth forms. The calculation takes into account tooth deformation, gear body deformation, and flattening due to Hertzian pressure. Calculation as defined in Weber/Banaschek [69].

For helical gear teeth the overall stiffness is calculated with the section model (the facewidth is split into 100 sections and stiffness added over all sections), see also [58], page 203. The transmission error is defined in accordance with [65], and the transmission variation in the peripheral direction is Δ:

$$\Delta s = \left(\frac{F_{bt}}{b}\right) \cdot \left(\frac{q}{c'}\right) \tag{21.5}$$

$$F_{bt} = \frac{F_t}{\cos(\alpha_t)} \tag{21.6}$$

where (q/c') is replaced by c_{gam}.

NOTE:

The theoretical contact stiffness and the contact stiffness of the effective toothing under load can be quite different.

21.4.8 Contact line (face gear)

To display the contact line on the pinion and on the face gear, select `Graphics > Evaluation > Contact line pinion` or `Contact line face gear`, see Figure 21.10:

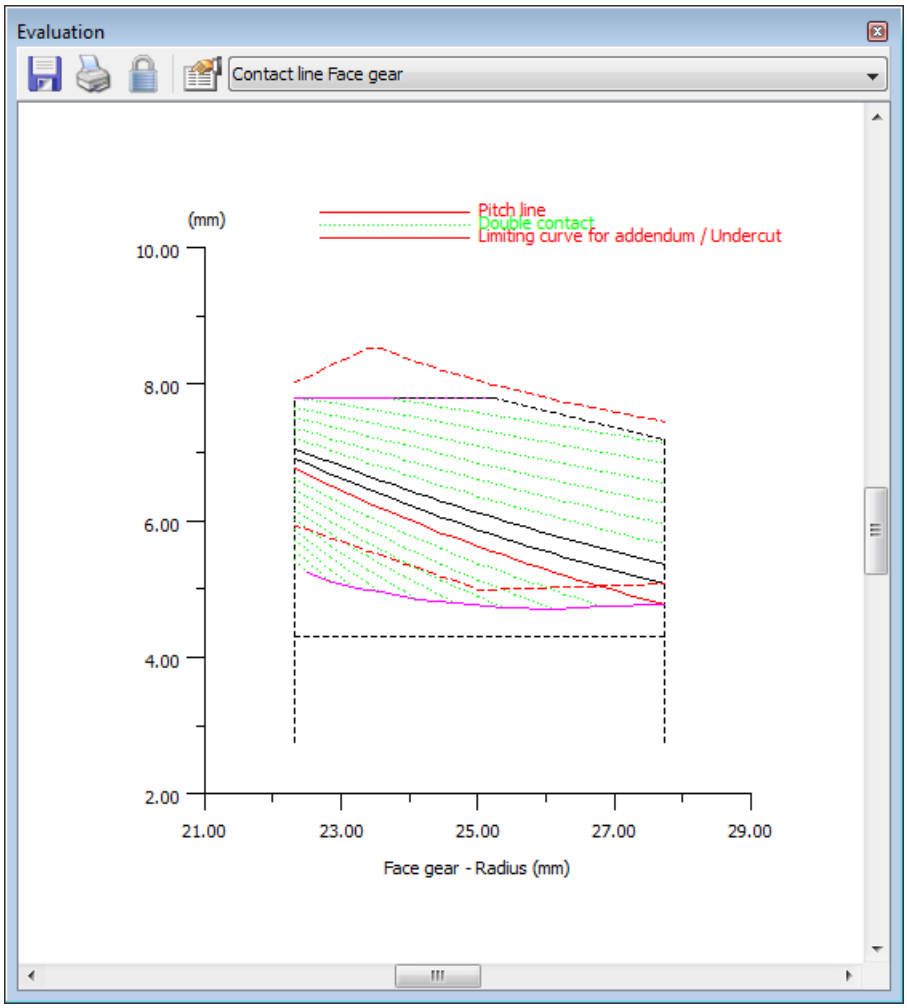


Figure 21.10: Graphics window `Face gear contact line`

21.4.9 Stress curve (face gear)

Select `Graphics > Evaluation > Stress curve` to calculate and display the progression of stress across the facewidth of the gear (see Figure 21.11). This splits the facewidth into segments which you can then calculate as pairs of racks as specified either in ISO 6336, DIN 3990, or AGMA2001. The calculation assumes a constant line load (which results in a slightly different torque for each segment due to the different pitch circle).

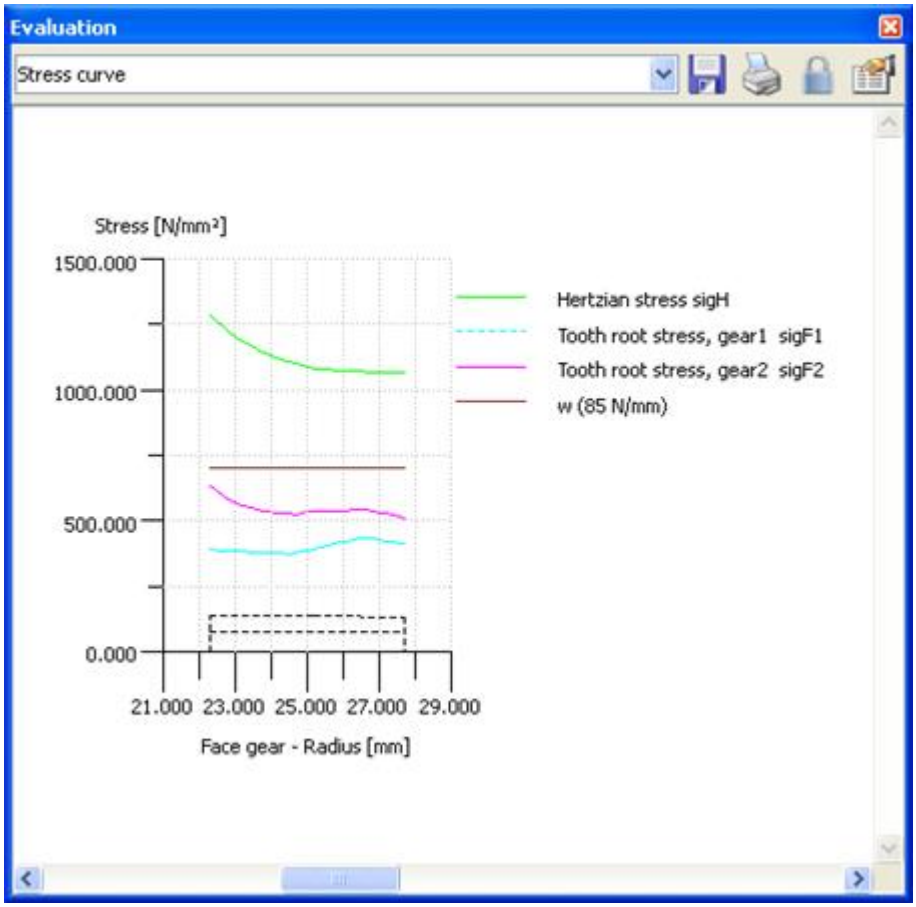


Figure 21.11: `Stress curve` graphics window

When you calculate data in order to represent the contact line and the stress curve, the most important values are calculated in separate sections and saved to two tables. This data is stored in the Z60-H1.TMP and Z60-H2.TMP files.

21.4.10 Scuffing and sliding velocity (face gear)

To display scuffing safety, select `Graphics > Evaluation > Safety scuffing` (see Figure 21.12). However, due to the very different sliding velocities and the changing flank pressure across the tooth flank, calculating the scuffing safety is actually very difficult. Akahori [2] reports massive problems with scuffing at high sliding velocities. For this reason it is appropriate to think about way in which you can calculate the risk of scuffing. One sensible option, as described above for stress distribution, is to calculate scuffing safety in separate sections. Figure 21.12 shows the progression of scuffing safety as defined by the flash and integral temperature criterion along the tooth flank. To achieve realistic results from this calculation, it must be ensured that every section is calculated with the same mass temperature. However, when you work through the calculation you will see there are significant changes in safety when the calculation is performed on the basis of the integral temperature. In particular, this happens as point E on the path of contact gets closer to the pitch point. If you then use the formulae in DIN 3990 to convert the flank temperature at point E to the average flank temperature the results you get will not be particularly precise. For this reason, we recommend you use the flash temperature as the criterion when you perform this calculation for face gears.

Select `Graphics > Evaluation > Sliding speed` to display the sliding velocity. The sliding velocities are important for a number of different applications (for example, plastic, dry run).

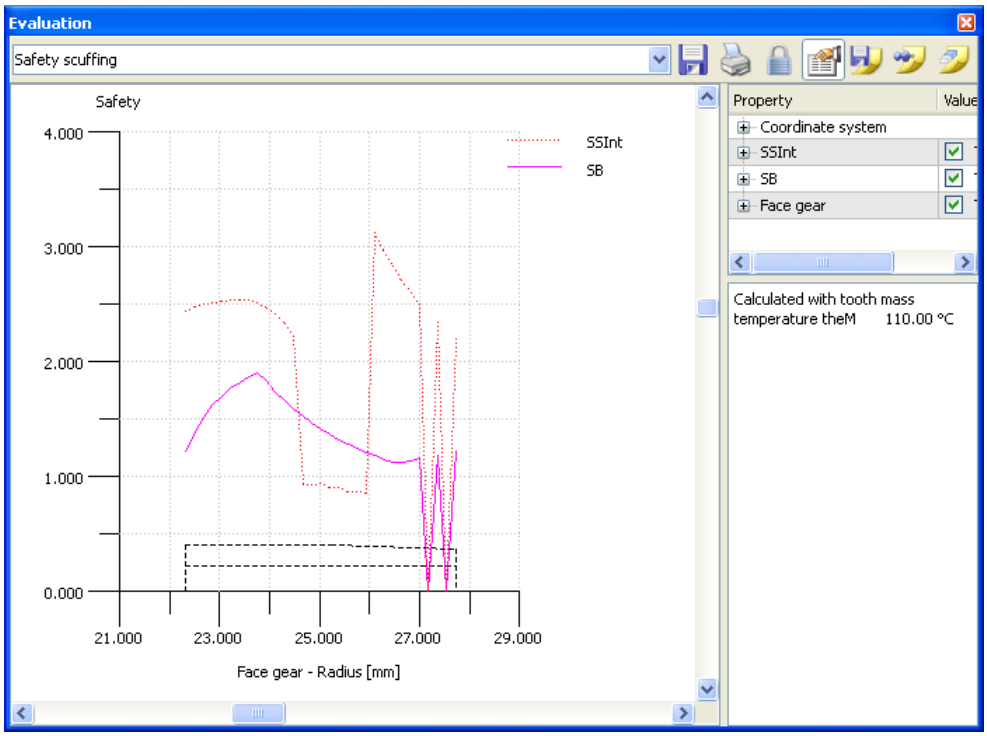


Figure 21.12: Graphics window: `Scuffing safety`

Select `Graphics > Evaluation > Sliding speed` to display the sliding velocity. The sliding velocities are important for a number of different applications (for example, plastic, dry run).

21.5 Contact analysis

NOTES:

The usual strength and speed calculations performed on gears assume that an involute tooth form is being used. However, if you use this program module, you can calculate and evaluate any type of toothing, such as cycloid toothing, just as accurately as involute tooth forms.

All the graphics can be exported:

- 2D diagrams as:
  - BMP
  - JPG
  - PNG
  - DXF
  - IGES
  - TXT
- 2D curves as:
  - TXT
- 3D diagrams as:
  - BMP
  - JPG
  - PNG
  - DAT (for the contact analysis, the Y axis is only output if the "Draw data for path of contact" option is selected in the module-specific settings)

(→see Figure 13.3 on page II-253)

21.5.1 Axis alignment

Display the axis alignment of gear B relative to the axis of gear A. This display is a very useful way of checking the deviation error and inclination error of axis.

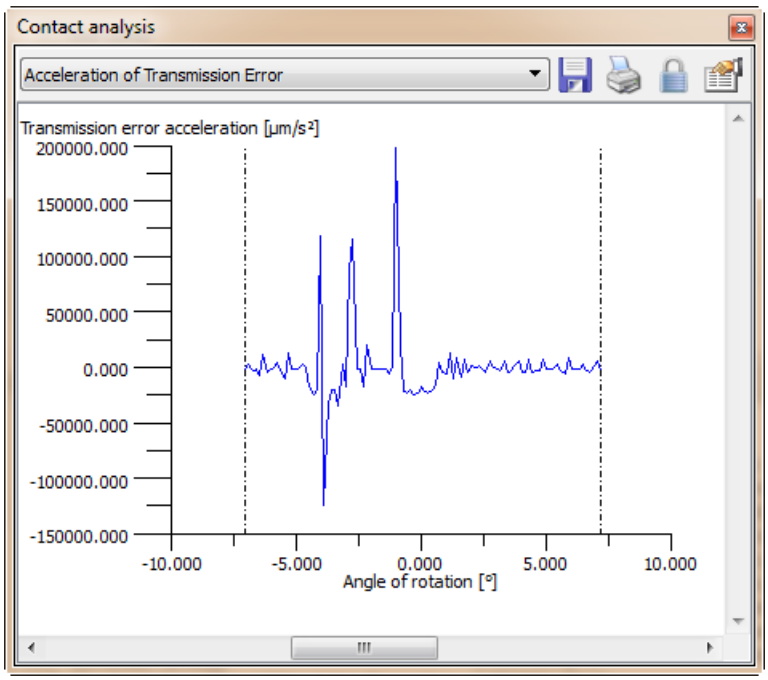
21.5.2 Transmission error

The path of contact under load is used to calculate transmission errors. The diagram shows the displacement of the contact point (μ) of the second gear on the length of the path of contact or the torsional angle (°) of the driven gear.

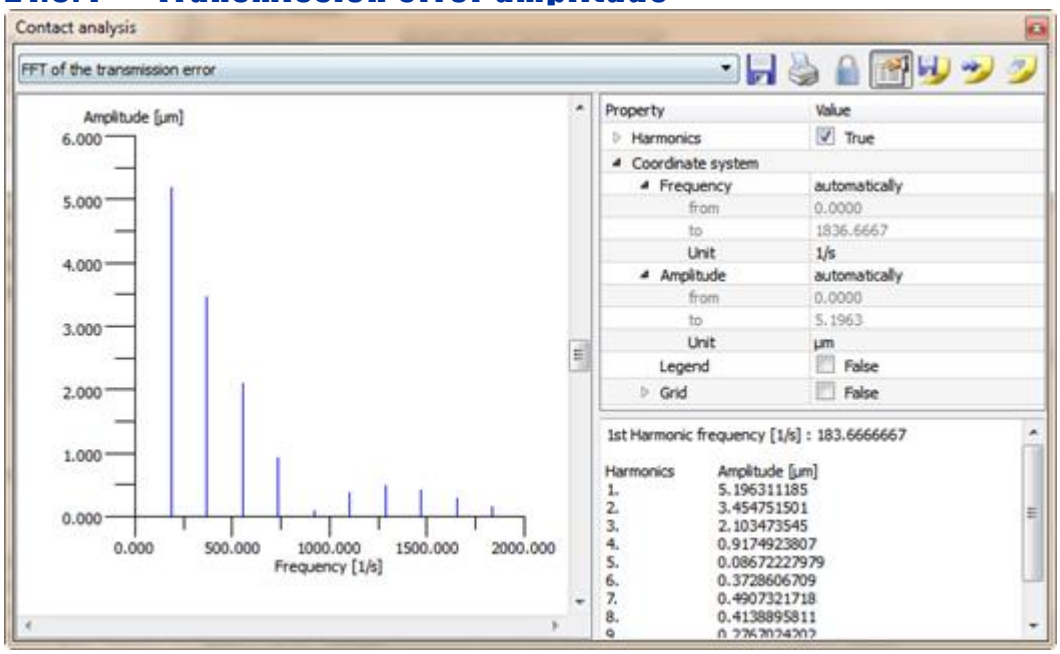
The amplitude of the transmission error plays a role in how much noise is generated but, despite this, you should not ignore the pitch, because high speeds also generate high additional loads.

21.5.3 Transmission error acceleration

The acceleration of transmission error (second derivative with reference to time) is available as a graphic.



21.5.4 Transmission error amplitude



The graphic shows the results of the spectroscopic analysis of the transmission error by fast Fourier transformation.

The user can compare the amplitudes of the spectra with the harmonic frequencies of transmission error in the comment window.

21.5.5 Contact lines on tooth flank

In this graphic you can examine the contact line along the facewidth. All the gear pairs in the contact are shown at the same time in a contact position.

21.5.6 Normal force curve

The normal force curve represents the line load per width for each tooth face in the middle of the cylindrical gear. In a well arranged profile modification, the normal force should increase steadily from zero. If you do not have a profile modification, an overlap length in the normal force curve shows the corner contact.

21.5.7 Normal force distribution

This graphic shows the normal force curve along the tooth flank and facewidth on a 3D gear.

21.5.8 Torque curve

The default value for torque defined in the main screen is kept constant during the calculation. The graphic then shows the torque for gear 1 and the torque for gear 2 divided by the transmission ratio. If these two torque values are different, it means that torque has been lost. The loss is due to friction in the tooth contact.

Variations in the displayed moment course depend on the level of accuracy you have specified and are caused by the accuracy of the iteration.

21.5.9 Single contact stiffness

This graphic shows the individual elements of single tooth contact stiffness. These are the stiffness of both gears and the single tooth contact stiffness of the gear pair.

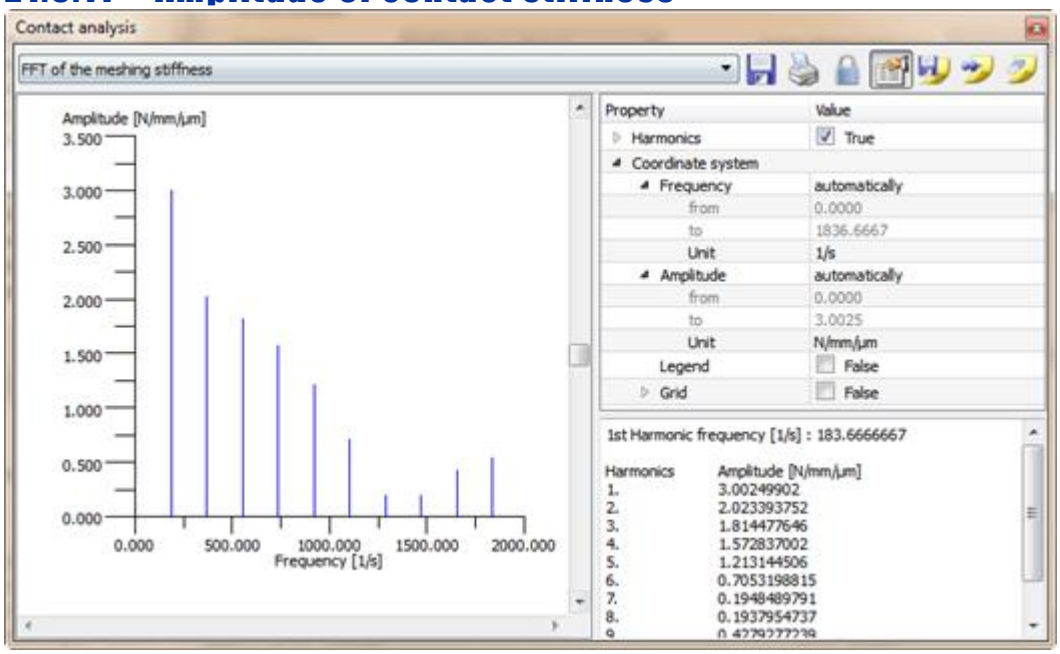
As this is a series-connected spring system, the following applies:

$$\frac{1}{C_{Pair}} = \frac{1}{C_{Gear\ 1}} + \frac{1}{C_{Gear\ 2}}$$

21.5.10 Stiffness curve

The stiffness curve shows the local stiffness at the operating point. It is calculated from the torsion under load at every point of contact. The stiffness value for gears is usually specified per mm facewidth. To calculate the stiffness of the toothing of two gears, multiply the value you specify (c_γ) with the tooth facewidth.

21.5.11 Amplitude of contact stiffness



The graphic displays the results of the spectroscopic analysis of the contact stiffness by fast Fourier transformation.

The user can compare the amplitudes of the spectra with the harmonic frequencies of contact stiffness in the comment window.

21.5.12 Bearing force curve and direction of the bearing forces

The bearing force curve assumes that the support is mounted with a symmetrical bearing position. The value given for the face load factor calculation is used as the distance between the bearings. The purpose of this graphic is not to display the correct bearing forces, but to represent the variations in these forces.

Variations in the bearing forces cause vibrations in the shafts and changes in gear case deformations.

21.5.13 Kinematics

The effective tooth form and the effective path of contact are used to calculate a wide range of kinematic values which are then displayed along the length of path of contact:

- specific sliding

- sliding coefficients K_g
- sliding velocity
- variation in transmission ratio

21.5.14 Specific sliding

You can display specific sliding either alongside the meshing cycle under `Kinematics` or alongside the tooth depth. You can also see it clearly in the area of the tooth flank with contact.

21.5.15 Power loss

This calculates the power loss for a pair of teeth. Power loss is usually greatest at the start and at the end of the mesh because this is where the highest sliding velocities are generated. However, with a profile modification, you can reduce the load at these points so that the maximum value is shifted to the width between start of mesh and the operating pitch point and to the width between end of mesh and the operating pitch point.

21.5.16 Heat development

Heat development links power loss with specific sliding. If the contact point of a gear moves slowly, it creates a higher heat value per length than if the contact point moves more quickly.

High temperatures generated on the tooth flank should be in correlation with the tendency to scuffing. However, this is not directly attributable to temperature.

21.5.17 Stress curve

The effective tooth form is used to calculate and display the exact Hertzian pressure during generating. The same applies to calculating tooth root stress, as defined in the Obsieger process (see page II-275), where the maximum stress in the tooth root area is shown by the angle of rotation.

Stresses are calculated with $K_{H\beta}$ = 1.0, $K_{H\alpha}$= 1.0, $K_{F\beta}$= 1.0, $K_{F\alpha}$= 1.0; only K_A, $K_v$and K_γ are included.

21.5.18 Flash temperature

The effective local temperature shown in the diagram at each point in the path of contact is defined by the gear base temperature (the tooth mass temperature) plus additional local warming (the flash temperature)

Use this data at each contact point from the path of contact calculation to calculate the flash temperature on the tooth flank:

- Sliding velocity
- Speed in a tangential direction to the pinion and gear
- Bevel radii on the tooth flanks
- Hertzian pressure

The friction value introduced to the calculation of the path of contact is used as the coefficient of friction μ. The tooth mass temperature is calculated as specified in ISO TR 15144.

Flash temperature is calculated for:

- ISO as defined in ISO TR 15144

$$\theta_{fl,Y} = \frac{\sqrt{\pi}}{2} \cdot \frac{\mu_m \cdot p_{dyn,Y} \cdot 10^6 \cdot |v_{g,Y}|}{\sqrt{B_{M1} \cdot w_{Y1}} + \sqrt{B_{M2} \cdot w_{Y2}}} \cdot \sqrt{8 \cdot \rho_{rel,Y} \cdot \frac{p_{H,Y}}{1000 \cdot E_r}}$$

- AGMA in accordance with AGMA925 with equation 84

21.5.19 Safety against micropitting

Calculation method

The calculation is performed in accordance with ISO 15144, Method A. All the required data is taken from the contact analysis.

Lubrication gap thickness h and specific thickness of lubrication film λ_{GFP}

The calculation of the progression of the effective lubrication gap thickness h and the effective specific lubrication gap thickness λ_{GF} across the meshing is precisely defined in the ISO TR 15144 proposal. The lubrication gap can vary significantly depending on local sliding velocity, load and thermal conditions. The location with the smallest specific lubrication gap thickness is the decisive factor in evaluating the risk of micropitting.

Permitted specific thickness of lubrication film λ_{GFP}

To evaluate the risk of micropitting it is vital that you know how large the required smallest specific lubrication film λ_{GFmin} is to be. The calculation rule states that:

λ_{GFmin} >= λ_{GFP} to prevent frosting (micropitting), or to ensure safety against frosting Sl = λ_{GFminP}/ λ_{GFP}.

If the lubricant's micropitting load stage is known, the permitted specific thickness of lubrication film is calculated in accordance with ISO TR 15144.

Otherwise, reference values for λ_{GFP} can be derived from the appropriate technical literature.

In [81] you will see a diagram that shows the permitted specific lubrication gap thickness λ_{GFP} for mineral oils, depending on oil viscosity and the frosting damage level SKS.

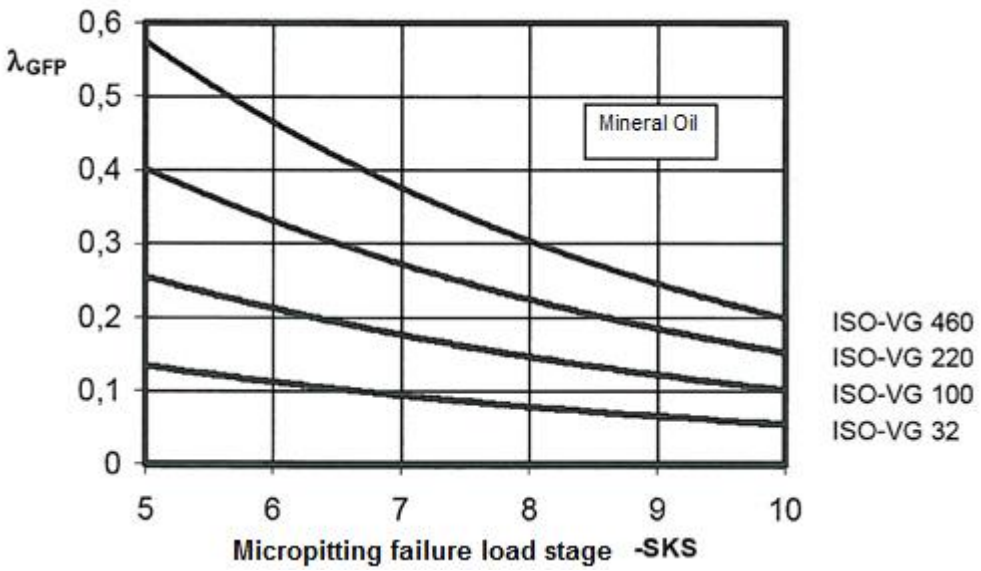


Figure 21.13: Minimum necessary specific thickness of lubrication film λ_{GFP}

The frosting damage level SKS, determined in accordance with the FVA information sheet [82], is nowadays also stated in data sheets produced by various lubricant manufacturers. The data in the diagram applies to mineral oils. However, synthetic oils with the same viscosity and frosting damage level show a lower permitted specific lubrication film λ_{GFP} [81]. Unfortunately, as no systematic research has been carried out on its effects, no properly qualified values are available.

Furthermore, you must be aware that the predefined values λ_{GFP} only apply to case-hardened materials. As specified in ISO TR 15144, for other materials, the

permitted specific lubrication gap thickness λ_{GFP} can be multiplied by the coefficient Ww.

	Ww
Case hardening steel, with austenite content <= 25%	1.00
Case hardening steel, with austenite content > 25%	0.95
Gas-nitrided (HV > 850)	1.50
Induction or flame-hardened	0.65
Heat treatable steel	0.50

Table 21.1: `Material coefficient`

It is interesting to note that, according at least to the table shown above, when the same lubrication gap is used, nitrided materials are more prone to micropitting than case-hardened materials. In contrast, through hardened materials that are not surface hardened are much more resistant.

You should be aware that the data shown here must be used with caution because information about the micropitting process is still incomplete and even technical publications will sometimes present contradictory data.

Safety against micropitting

If the load stage against micropitting as defined in FVA C-GF/8.3/90[82] is specified for the lubricant, the minimum required thickness of lubrication film λ_{GFP} is calculated. The safety against micropitting can therefore be defined as $S\lambda = \lambda_{GFmin} / \lambda_{GFP}$.

21.5.20 Wear

To calculate local wear on the tooth flank, you must first determine the wear factor of the material Jw. This coefficient can be measured using gear testing apparatus or by implementing a simple test procedure (for example, pin/disc test gear) to determine the appropriate value. Investigations are currently being carried out to see how the Jw coefficients determined using a simpler measurement method can be applied to gears. For exact forecasts, you will also need to determine the coefficient Jw for the material pairing. For example, POM paired with POM does not supply the same results as POM paired with steel.

Plastics

You can input the wear factor Jw, in the polymer data file, for plastics, depending on the temperature (for example, Z014-100.DAT for POM). The data is input in 10^{-6} mm^3/Nm.

As an example:

```
-- Data for wear (PROVISORY! From Dissertation R.Feulner, 2008) Dependency from Temperature unknown
-- in 10^-6 mm3/Nm
:TABLE FUNCTION KwearDry
      INPUT X ZahnTempFlanke TREAT LINEAR
DATA
    -20     23     40     90    120
   1.03   1.03   1.03   1.03   1.03
END
```

Steel

Plewe's investigations have revealed that a rough approximation of the wear factors for steel materials can be defined. See also the calculation of wear factors for steel (Calculation of wear factor kw for steel) (see page II-264)

Calculation

Wear is calculated in accordance with the following base equation:

$$\delta_w = J_w \cdot P \cdot V \cdot T$$

(δw [mm], Jw [mm^3/Nm], P: Pressure [N/mm^2], V:speed [mm/s], T:time[s])

As modified to suit gear conditions, local wear results from:

$$\delta_{w_i} = NL \cdot J_W \cdot w \cdot \zeta_{_i}$$

(i = 1.2)

(δw_i [mm], Jw [mm^3/Nm], NL: Number of load cycles, w:Line load [N/mm], ζ_i: specific sliding)

This equation also corresponds to the data in [83], Equation 6.1.

The calculation to determine wear on the tooth flank uses the following data at each point of contact taken from the calculation of the path of contact:

- Specific sliding
- Line load

For POM against steel (at 23°C), [83] gives a Jw of 1.03 * 10^{-6} mm^3/Nm. For PBT against steel it gives a Jw of 3.69 * 10^{-6} mm^3/Nm.

When you interpret the results, you must note that the increasing wear on the tooth flank to some extent changes local conditions (line load, sliding velocity) and therefore also changes the increase in wear itself. For this reason, after a number of load changes, you must select the worn flank (red line in the figure) and use it to recalculate the path of contact.

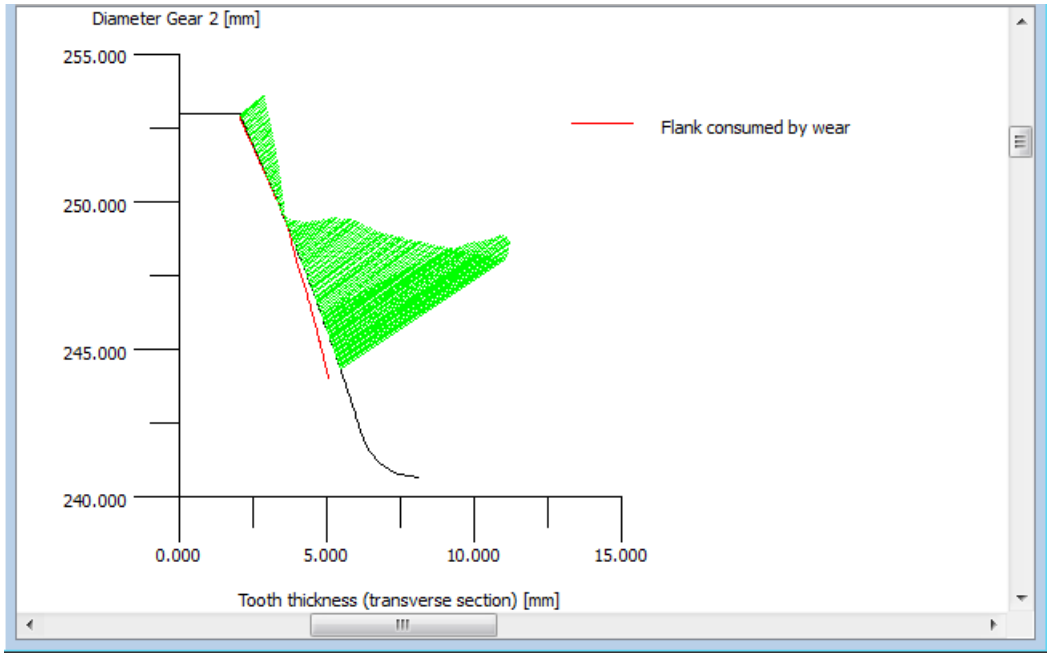


Figure 21.14: Graphics window `Wear`

21.6 Gear pump

Eleven different diagrams document in detail the progressions of the characteristic values in a gear pump when it is generated. You will find more detailed information about how to calculate gear pumps (see section "Gear pump" on page II-393) in KISSsoft-anl-035-E- GearPumpInstructions.doc [77] (available on request).

21.7 3D export

Click `Graphics > 3D Export` to export the geometry of the gears you have just designed to a specified CAD system. The next section (see page II-609) provides more detailed information about which CAD system you should use, and its interface.

NOTE:

Before you call this function for the first time, make sure you are using a suitable CAD system. If you have specified a CAD program that has not yet been installed you may cause a problem when you call this function.

21.8 Settings

Click `Graphics > Settings` to define the background for 3D graphics and select your preferred CAD system. Here you can select any of the interfaces for which you have the appropriate licenses.

The graphics list is where you save the graphics with , as in all the other toothing modules. This is attached to the end of the report, unless otherwise specified in the report template. In the graphics list you can open every graphic with , , and , depending on the graphic type, and then modify enable/disable its properties or delete it with .

Chapter 22
Answers to Frequently Asked Questions

23.1 Answers concerning geometry calculation

23.1.1 Precision mechanics

KISSsoft is an ideal tool for calculating the gears for precision mechanics.

The reference profile and the geometry are calculated as defined in DIN 54800 etc. The strength calculation is performed in accordance with ISO 6336, VDI 2545 or DIN 3990, since no special strength calculation exists for precision gears. For this reason, "Defining required safeties for gear calculation (see section "Required safeties for cylindrical gears" on page II-622)" is important when you are interpreting the results.

If gears are manufactured using topping tools, the tip circle can be used to measure the tooth thickness. In this situation, it is critical that you specify precise value of the addendum in the reference profile to match the corresponding cutter or tool. This is because this value is used to calculate the tip circle. The tip alteration k*mn is not taken into account in the calculation of the manufactured tip circle. The following formula is used:

$$d_{aeff} = d + 2 \cdot m_n \cdot x_{eff} + 2 \cdot m_n \cdot h_{aP} \tag{22.1}$$

23.1.2 Deep toothing or cylindrical gears with a high transverse contact ratio

Using deep toothed gears is recommended for some specific applications (for example, for spur gears that should not generate a lot of noise).

In KISSsoft, you can easily calculate all aspects of deep toothed gears. To calculate the geometry, you must select a profile of a suitable height when you select the reference profile:

Normal profile height: e.g. m_n * (1.25 + 1.0)
For deep tooth form: e.g. m_n * (1.45 + 1.25)

You must be aware that this type of gear is more prone to errors such as undercut or pointed teeth. Experience has shown that you must select a value of 20 or higher as the number of pinion teeth to ensure that you can create a functionally reliable pair of gears. KISSsoft also has very effective and easy to use strength calculation

functionality; as specified in DIN 3990, Part 3, calculation of gears with a transverse contact ratio greater than 2.0 tends to be on the conservative side.

The Geometry-Variants calculation (Modules Z04 and Z04a) is very good at sizing optimum arrangements of deep toothed pairs of gears!

See also section 13.16.

23.1.3 Pairing an external gear to an inside gear that has a slightly different number of teeth

When you pair a pinion (for example, with 39 teeth) with an internal gear (for example, with 40 teeth) that has a slightly different number of teeth, the teeth may have a collision ("topping") outside the meshing area. This effect is checked and an error message is displayed if it occurs.

To size a functioning pairing of this type, select this strategy:

- Reference profile: Short cut toothing
- Pressure angle: the bigger the better
- Sum of profile shift coefficients: select a negative value
- Pinion profile shift coefficient: approximately 0.4 to 0.7

23.1.4 Undercut or insufficient effective involute

(this triggers frequent error messages when you calculate the geometry of cylindrical gears.)

An insufficient effective involute occurs if the tip of the other gear in the pair meshes so deeply with the root of the other gear that it reaches a point where the involute has already passed into the root rounding. These areas are subject to greater wear and tear. Some gear calculation programs do not check this effect and suffer recurrent problems as a consequence.

To keep a close eye on the undercut and effective involute, you should always work with the `Calculate form circle from tooth form` (see page II-436) option. This function checks the tooth form every time a calculation is performed. It defines any undercut it discovers and takes it into account in the calculation.
The tooth form calculation takes into account all aspects of the manufacturing process. In contrast, calculating geometry in accordance with DIN 3960 uses simplified assumptions.

23.1.5 Tooth thickness at tip

The tooth thickness at the tip circle is calculated for a zero clearance status. In addition, the maximum and minimum value is calculated using all tolerances.

When you check the tooth geometry, the tooth thickness at the tip must usually be at least 0.2 * module (in accordance with DIN 3960). If this limit is not reached, KISSsoft displays the appropriate warning message. Click on `Calculation > Settings > General` to change this coefficient if required.

23.1.6 Special toothing

The term "special toothing" is used to describe toothing with non-involute flanks. The reference profile (or the normal section through the hobbing cutter or rack-shaped cutter) of special toothing is not straight (unlike involute toothing). However, the same generating process is used to manufacture both toothing types. As part of the tooth form calculation, special toothing can either be imported from CAD or defined directly (cycloid, arc of circle toothing). In addition, a suitable counter gear can then be generated by clicking `Generate tooth form from counter gear`.

By simulating the generation process, the tooth form and, from this, the geometry can then be defined for special toothing. As no standards or documentation are available for strength calculations, analogies for these tooth form types must be drawn from the calculations used for the cylindrical gear process. For more information see the Path of contact (see section "Contact analysis" on page II-596) section.

23.1.7 Calculating cylindrical gears manufactured using tools specified in DIN 3972

Profiles I and II are profiles for the final treatment. They can all be handled easily by KISSsoft. Simply select the tool you require from the selection list (Reference profiles).

Profiles III and IV belong to tools used in preliminary treatment. However, you should always use a finished contour to calculation the strength of a gear, these profiles should therefore only be used as a premachining cutter.

The reference profiles are dependent on the module as defined in the following formulae:

Profile III	$h_{fP} = 1.25 + 0.25\ m_n^{-2/3}$	$h_{aP} = 1.0$	$\rho_{fP} = 0.2$
Profile IV	$h_{fP} = 1.25 + 0.60\ m_n^{-2/3}$	$h_{aP} = 1.0$	$\rho_{fP} = 0.2$

In the `Reference profile` tab, if the configuration is set to `Tool: Hobbing cutter`, you can click the plus button in the hobbing cutter row to see a selection list that includes Profiles III and IV in accordance with DIN 3972. Remember that the data you enter here depends on the module. If you want to change the module, you must select a the correct reference profile again.

Use the recommendations in the standard to select the correct allowances for preliminary treatment:

Profile III	Grinding allowance = $+0.5\ m_n^{1/3} \tan(\alpha_n)$
Profile IV	Grinding allowance = $+1.2\ m_n^{1/3} \tan(\alpha_n)$

If Preliminary treatment has been selected (in the Reference profile tab), you can set the appropriate Grinding allowance for Profile III or IV in the list in the Grinding allowance field.

Click on the +button next to Grinding allowance q to input a tolerance interval for the grinding allowance qTol (=qmax-qmin). The grinding allowance for preliminary treatment then lies in the range qmin ... qmax, where qmin = q - qTol/2; qmax = q + qTol/2 applies.

The control masses (base tangent length etc.) for preliminary treatment are then calculated with the following allowances:

Maximum grinding allowance with As.e + qmin*2 / cos(an)

Minimum grinding allowance with As.i + qmax*2 / cos(an)

Note:

If you want customer-specific tolerances to be processed automatically, you can define them in a file called "GrindingTolerance.DAT". The \dat directory has an example of this type of file, which is called "GrindingToleranceExemple.DAT". When this file is renamed to "GrindingTolerance.DAT" its tolerance values are used in the calculation.

23.1.8 Variations in rolling as defined in DIN 58405

DIN 58405 specifies the base tangent length allowances and permitted composite errors for toothing used in precision mechanics. In this case, the reference profile specified in DIN 58400 assumes a pressure angle of α_n=20°. If you use a operating

pressure angle that is not 20°, DIN 58405 Sheet 3, sections 1.2.10 and 1.2.11, state that the permitted composite error and the permitted rolling deviations must be multiplied with a coefficient L = tan(20°)/tan(abs). This must be performed because the base tangent length allowances are standardized and the center distance error increases as the pressure angle is reduced. KISSsoft takes coefficient L into account when calculating tolerances to comply with DIN 58405, because it is specified in the standard.

However, the tolerances specified in ISO 1328 and DIN 3961 do not include this coefficient because it is not listed in the standard.

23.1.9 Automatic change of reference profiles

Some calculations have revealed the problem that the reference profile changes automatically when the center distance changes. In the Reference profiles tab, the factors for the tool tip and dedendums change automatically. Why?

This is because the "Retain tip circle or dedendum when the profile shift changes" checkbox is active in the General tab in the module-specific settings.
If you change the center distance, the profile shift coefficient also changes. The setting you make automatically changes the factors for the reference profile.

23.1.10 Non-identical (mirrored symmetry) tooth flanks

If the tooth flanks (left, right) are not identical, will this cause an error when the tooth contour is exported?

The tooth flanks used in the calculation (sizing) are identical.

The export function used in the system not only exports the involutes but also the entire tooth form. This is an approximated curve.

With the export precision (permitted variation ε) you can define how closely you want to approximate the calculated tooth form.

In each case, an approximate curve in the specified level of accuracy is given for either half of the tooth or the whole tooth. You can only use mirror symmetry with approximation accuracy.

This is the error you specified as the permitted variation.

The smaller the selected variation, the more detailed the curve.

23.1.11 Internal teeth - differences in the reference profile if you select different configurations

A gear pair with internal teeth has been calculated in the KISSsoft system. A pinion type cutter is then to be used to manufacture this internal gear. The tool is manufactured to suit particular customer requirements and is influenced by the particular tooth form which is used. This must reflect the reference profile geometry of the internal gear. How can you then determine the pinion type cutter geometry?

A gear's reference profile is the corresponding rack profile. regular hobbing cutter for an outside gear has this rack geometry, and therefore makes it easy to define the hobbing cutter profile. However, you must reverse the gear profile to achieve the hobbing cutter profile (the addendum of the gear reference profile becomes the dedendum of the hobbing cutter and so on).

If the manufacturing tool is a pinion type cutter, the limited number of teeth on the pinion type cutter result in a different situation. Basically speaking, the inverse gear reference profile corresponds to that of the pinion type cutter. However after this, you must change the addendum of the cutter in such a way that you can achieve the necessary root diameter on the internal gear.

First of all, you must define the number of teeth on the pinion type cutter. Depending on the type of machine tool used to manufacture the gear, the reference diameter of the pinion type cutter is already predefined to some extent. This reference diameter must be greater than the diameter of the main shaft of the machine tool that is to be inserted in the pinion type cutter tool. However, if this diameter is too large in comparison with the size of the pinion type cutter, the shaft diameter will be too small. This will cause powerful vibrations during the production process and result in a poor accuracy grade. To prevent this, you must know the approximate pinion type cutter diameter. The reference diameter is then divided by the module to determine the number of teeth on the pinion cutter.

If you want to use the KISSsoft system to design the pinion type cutter geometry, you must first input the number of teeth on the pinion type cutter. You can start with 0.0 for the profile shift coefficient of the pinion type cutter. A pinion type cutter's profile shift changes as it is used. Every time the pinion type cutter is resharpened, the profile shift is reduced slightly. A new pinion type cutter usually has a positive profile shift (for example +0.2), a worn tool therefore has a negative profile shift.

After you have input the data for a pinion type cutter, you must first check all the entries, i.e. whether the required root form diameter has been achieved. If not, you must reduce the tip fillet radius of the pinion type cutter. If that does not help, you must increase the addendum of the tool reference profile. However this also changes the active root diameter.

The same problem can also happen with the tip form diameter dFa. It often happens that you cannot generate the entire involute part up to the tooth tip. In this situation, you must either increase the number of teeth on the pinion cutter tool or reduce the tip diameter of the gear.

If you develop a gear that is manufactured by a pinion type cutter, it is always critically important that you investigate the production process early on in the development process. This is because not every gear geometry can be created with this production process.

23.1.12 Effect of profile modifications

Profile modifications are a popular topic of discussion. Where should these modifications start, and which values should be used to make these modifications?

Linear tip relief is a type of profile modification. It has the following characteristics: Starting from a particular point, ever increasing amounts of material are removed from the involute toothing part up to the tip diameter.

The tooth contact in the modified area is disrupted. This is only a benefit when subject to the corresponding load. This entire area is taken into account when calculating the contact length to determine the transverse contact ratio ε_a. Shouldn't this be different?

If you use profile modifications you "delete" the real involute. Why is this a good idea?

This is a complex problem that must be taken into consideration when you design profile modifications. The amount of material removed (tip relief C_a is the reduction of tooth thickness at the tip due to the profile modification) and must be applied according to the tooth bending.

For example, if the tooth had infinite stiffness, and you ignore any of the possible effects of compensating for production errors, the profile modification would simply reduce the transverse contact ratio. If you did not take this profile modification into account, you would make an error in the geometry calculation. This is basically correct for a gear that is subject to a lower load. However, you will usually need to design gears for optimum performance at operating torque and the strain that this places on the teeth.

If the tip relief C_a is well arranged, the profile modification then compensates for the tooth deformation, so that the tooth contact across the entire tooth depth is not compromised. In this case, the transverse contact ratio is not reduced. Here you have, when compared to a gear without profile modification, a changed normal force curve over the meshing.

However, the maximum force (in the operating pitch diameter), where only one gear pair is in contact, is not changed. For this reason, the maximum root and flank strains, which determine the service life of the drive, remain unchanged. This profile modification reduces the normal force at the start and at the end of the meshing. This also leads to a significant reduction in the risk of scuffing. The risk of scuffing is due to flank pressure and sliding velocity. Sliding is greatest at the start and the end of the tooth contact and therefore, by reducing the flank pressure in this area, you can also reduce the risk of scuffing. A profile modification can reduce the influence of tooth strain on stiffness fluctuations across the meshing and therefore limit the number of transmission errors. This also lowers the levels of vibration and noise.

This clearly illustrates that a profile modification does not reduce the transverse contact ratio, as long as this has been properly arranged, i.e. for the operating torque of the drive. However, where lower loads are involved, the contact of gears where the profile has been modified is not as good as those without profile modification. This is because the transverse contact ratio has been significantly reduced. In this case, although the load would increase, it would do so by a comparatively small amount and can therefore be ignored.

23.1.13 Number of teeth with common multiples

A toothing with 15:55 teeth has been sized. Different documents state that you should avoid gear reductions (like 11:22) that are whole numbers. Furthermore, you will also discover that you should also avoid using numbers of teeth that are common multiples (in this case the 5 in 3*5 to 11*5). Is that true and is it displayed in KISSsoft?

Let's assume we have a gear which has a fault on one of its teeth. In a whole number gear reduction, this tooth will always come into contact with the same tooth in the counter gear. The error is then transmitted to the counter tooth. However, if the tooth with the fault comes into contact with a different counter tooth in every rotation, this error will be reduced as the gears wear in.

Nowadays, most gears are surface hardened. Unlike weak gears, they hardly ever wear in. As a result, this problem is now less critical than it used to be, where it was important that whole number gear reductions (such as 11:22) were avoided even when hardened gears were used. In contrast, whole number toothing combinations with common multiples (such as 15:55) are quite unobjectionable for surface hardened gears.

In KISSsoft you will find notes about whole number combinations with common multiples in both fine sizing and rough sizing under the keyword "hunting". If you see YES in the hunting table, this means: no common multiple is present.

23.1.14 Allowances for racks

From Release 10/2003 onwards, allowances for racks are defined in conjunction with the paired gear.

This conforms to DIN 3961.

"The tolerances for the toothing of a rack should not be greater than the tolerances of its counter gear. If the counter gear's manufacturer is not known, the rack length should be the same as the counter gear circumference."

23.2 Answers to questions about strength calculation

23.2.1 Differences between different gear calculation programs

You will always discover differences in the results when you compare calculations performed with different gear calculation programs. Many of these differences are due to the different data entered. However, even if all the data entered is the same, you will still get different results.

One of the questions our users often ask is whether the results calculated by KISSsoft are correct.

The main calculation process used in the KISSsoft cylindrical gear calculation functions is based on DIN 3990, ISO 6336, and AGMA, It faithfully follows the procedure described in Method B. However, as DIN 3990, or ISO 6336 offer various different Methods (B, C, D) and sub methods, it is no surprise that the results they supply are slightly different from other calculation programs. Most programs do not perform calculations that consistently use method B, instead they partially use Method C or even D, which are easier to program.

To give our users an additional safety, we have therefore integrated the `FVA program` calculation variant into KISSsoft. This variant supplies exactly the same results as the FVA program ST+, that was developed by the Technical University in Munich and which can be used as a reference program.
The minor differences between KISSsoft's calculations in accordance with DIN 3990 and the FVA programs are due to the slight (permissible) deviations of the FVA program from the standard process defined in DIN 3990.

23.2.2 Difference between cylindrical gear calculation following ISO 6336 or DIN 3990

The strength calculation method used in ISO 6336 is virtually the same as that defined in DIN 3990. The majority of the differences only affect minor details which have very little effect on the safeties calculated for tooth root, flank and scuffing.

The only significant difference happens to be the life factor (Z_{NT} and Y_{NT}). In the endurance area (in accordance with DIN, depending on material type and calculation method 10^7 to 10^9 load cycles) this coefficient in ISO 6336 decreases

from 1.0 to 0.85 at 10^{10} load cycles. Only with "optimum material treatment and experience" does the coefficient remain 1.0.

As a result, gears in the range of endurance limit range supply much smaller safeties (15% lower) when calculated in accordance with ISO 6336 for root and flank! In the case of optimum material treatment or for the number of load cycles in the limited fatigue strength range, the safeties are practically identical.

23.2.3 Calculation using Methods B or C (DIN 3990, 3991)

Cylindrical gears:

Calculation using Method B or C is described in DIN 3990. Method B is much more detailed and is therefore the method we recommend KISSsoft usually uses Method B. However, we do not consider Method B to be precise enough to calculate the form factors for internal teeth, which is why we recommend Method C.

Converting to using Method C means that most of the calculation is performed in accordance with Method B and only the tooth form factor is calculated as defined in Method C.

Note: The most precise way of calculating internal teeth is to take the exact tooth form into account (see "Tooth form factor using graphical method", chapter 13.3.16.3).

Bevel gears:

Tooth form factors are calculated in accordance with standard Method C.

23.2.4 Required safeties for cylindrical gears

Defining the necessary safeties (for tooth root, flank, scuffing) for gears in a particular application, for example, in industry standard drives, vehicles, presses etc., is a very important step in the gear calculation process.

The (DIN 3990 or ISO 6336) standards give hardly any information about this; DIN 3990, Part 11 (industrial gears) has this data:

Minimum safety for root:	1.4
Minimum safety for flank:	1.0

AGMA2001 does not specify minimum safeties. The AGMA 6006 guideline (for gearboxes in wind power installations) has a note that SFmin = 1.56 is specified for root safety for calculation in accordance with ISO6336. In contrast, SFmin = 1.0 is

sufficient for calculations in accordance with AGMA. This matches our findings that calculations performed in accordance with AGMA give much lower root safeties.
We therefore recommend a minimum safety of 1.4*1.0/1.56 = 0.90 for industrial gears calculated in accordance with AGMA.

Scuffing is calculated in accordance with DIN 3990, Part 4:

Minimum safety for scuffing (integral temperature): 1.8

Minimum safety for scuffing (flash temperature): 2.0

The standards do not specify this value for precision mechanics (module under 1.5). Despite this, in accordance with empirical values the required safeties are much smaller than for gears with a larger module (root 0.8; flank 0.6)! The reason for this: The formulae and methods used in strength calculation are all taken from tests with larger gears and only supply very conservative factors (values that err on the side of safety) for small modules.

Defining required safeties for gear calculation

You can use the simple method described here to obtain the required safeties:

1. Examine and define the basic settings of the calculation (e.g. application factor, lubricant, accuracy grade, processing etc.).
2. Then apply the gear calculation method (without changing the basic settings unless you absolutely have to!) on known set of gears. You should select gears that run reliably under operating conditions and also such that have failed.
3. You can then use the resulting safeties calculated with these gear sets to define the point up to which minimum service reliability can be guaranteed.
4. You can then use these parameters to calculate the sizing of new gears. You can, of course, change these minimum safeties to reflect the results of your own tests and examinations.

23.2.5 Insufficient scuffing safety

You can increase scuffing safety by:

- Oil selection (higher viscosity at high temperatures)
- Tip relief (profile modification)
- Different distribution of the profile shift

The methods used to calculate scuffing safety (unlike those used to determine the tooth root and flank) is still a matter of controversy. For this reason, you should not pay too much attention to it, especially if the results of scuffing safety at flash temperature and the integral temperature process are very different.

23.2.6 Material pairing factor (strengthening an unhardened gear)

When pairing a hardened gear with an unhardened gear (e.g. pinion made of 17CrNiMo6 and gear made of 42CrMo4) you get the positive effect of increased load capability on the flank of the unhardened gear. This effect is taken into account by the material pairing coefficient (coefficient in the range 1.0 to 1.2). As stated in ISO 6336, the surface roughness of the hardened gear should be low (polished surface), otherwise the load capability will not increase; on the contrary, the tooth of the weaker gear may actually be ground off.

23.2.7 Defining the scoring load level (oil specification)

In accordance with Niemann [65], page 166, on a test rig the torque on the test gear is gradually increased until scuffing occurs. This torque level is then entered in the oil specification parameters (example: no scuffing at load 10; scuffing at load 11: scuffing load level of the oil is therefore 11).

To calculate the resistance to scoring you must then enter this load level (for the oil specification). In the example described above this is the value 11 (in accordance with Niemann [65], page 341). The scuffing safety calculation defines the safety against scuffing with predefined safeties greater than 1.0. This creates a necessary reserve, because the gradual increase in torque used in the test only approximates the effective scuffing torque.

23.2.8 The influence of the face load factor KHß for tooth trace deviation fma is due to a manufacturing error.

When calculating a cylindrical gear in accordance with ISO 6336, a higher value was determined for the tooth trace deviation f_{ma} when calculating the face load factor $K_{H\beta}$. This was due to a manufacturing error. The value for $K_{H\beta}$ does not change. Why then, does this value for $K_{H\beta}$ not change if a higher value for f_{ma} is used ?

Before you can calculate $K_{H\beta}$ you must input the position of the contact pattern. If the contact pattern has been defined as "economical" or "optimum", $K_{H\beta}$ is calculated in accordance with the formulae in ISO 6336 or DIN 3990. f_{ma} has no influence on the calculation of $K_{H\beta}$ and is therefore ignored.

See formulae: (53) or (55) in ISO 6336:2006.

The reason for this is that a well designed contact pattern can compensate for manufacturing variations and variations due to deformation. If a higher value of f_{ma} is to be used in the calculation, this means, in reality, that a good contact pattern can never be present. That is why, in this situation, you should select the contact pattern position "not verified or inappropriate" when calculating the face load factor.

23.2.9 Load spectrum with changing torque

You can also enter load spectrum elements with negative torque, but then the prefix operator is NOT taken into account.

The problem:

Until now, no calculation guidelines have been drawn up to describe how to calculate gears with changing load spectrums.

The only unambiguous data is that, during every cycle (and in each element of the collective) a change in torque takes place. At this point, the load change corresponds exactly to a double-load with +torque and then with –torque. This instance can be calculated correctly by entering the load spectrum of the +moments and the alternating bending factor YM for the tooth root. The flank is also calculated correctly, because the +moments always apply to the same flank.

If, in contrast, the drive runs forwards for a specific period of time and then runs backwards, the experts agree that the tooth root is not subjected purely to an alternating load (and possibly this is the only point at which an alternating load change takes place). However, discussions are still raging as to how this case can be evaluated mathematically. It is even more difficult to define how mixed load spectra with unequal + moments and –moments for the tooth root are to be handled. For this type of case, only the +moments are observed for the flank (with the prerequisite that the +moments are equal to or larger than the –moments).

Note on the handling of load spectra with reversing torque:

A load progression as represented in Figure 13.10 below, where the tooth is subjected to a load a few times on the left flank, and then a few times on the right

flank, can be converted into a load spectrum as shown below. This is represented in an example here.

Load progression (example):

13 loads with 100% of the nominal load (100 Nm) on the left flank, then

9 loads with 80% of the nominal load (80 Nm) on the right flank, etc.

This results in the following process:

11 load cycles with 100% load, positive torque, pulsating; then

1 load cycle with 100% load on the left and 80% load on the right; then

7 load cycles with 80% load, negative torque, pulsating; then

1 load cycle with 80% load on the right and 100% load on the left;

then repeated again from the start.

This can be represented as a load spectrum as follows:

Frequency	Torque	Load left flank	Load right flank
11/20 = 0.55	100 Nm	100%	0%
7/20 = 0.35	80 Nm	0%	100%
2/20 = 0.10	100 Nm	100%	80%

23.2.10 Strength calculation with several geometries on one gear

How can you take several simultaneous meshing points on a motor pinion into account in the calculation?

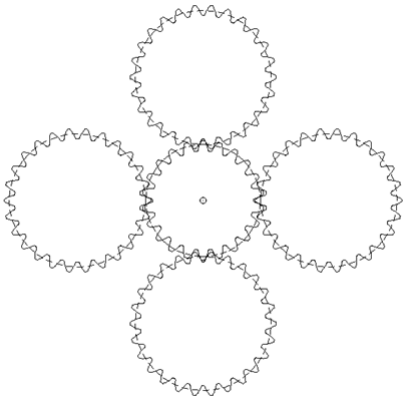


Figure 22.1: Fourfold meshing

You can solve this problem with a normal face gear pair calculation (Z12).

Simply divide the power by a factor of 4 (reduce by 25%)

Then press the "Details" button in the Strength area behind the reference gear.

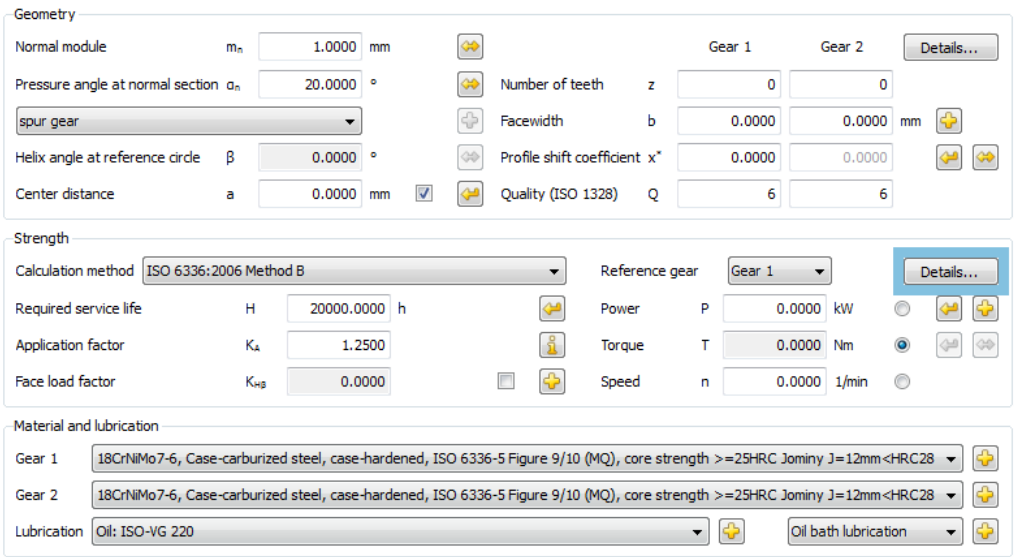


Figure 22.2: Details Strength

Then press the plus button after the number of load cycles to perform the next change. The number of load cycles for gear 1 changes "Automatically" to 4 load cycles per revolution.

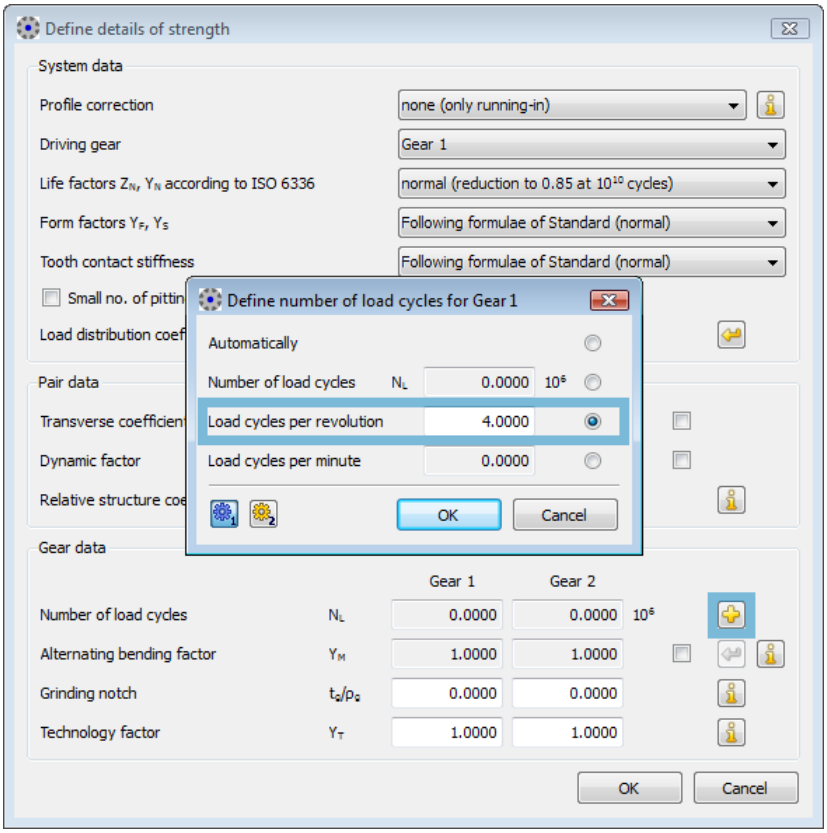


Figure 22.3: Define number of load cycles for gear 1

23.2.11 Bevel gears: – Determine permitted overloads

Can maximum overloads be taken into account when calculating bevel gears in accordance with ISO standards?

AGMA norms have definitions that allow for a standard overload of 250%. This overload is defined as being present for less than 1 second, not more than 4 times in an 8 hour time period. Does the ISO standard have comparable regulations with regard to overloads (shock)? No references could be found about this subject in the ISO standard.

ISO 10300 does not give any information about permitted overloads. However, ISO has a different Woehler curve (for YNT and ZNT factors) than AGMA. Therefore, in principle if ISO 10300 is strictly adhered to, the total number of load

changes including the overload must be input. The application factor is 2.5 (which corresponds to 250% overload). After this, you must calculate and check the safety factors.

If the load only occurs very infrequently, (less than 1000 times during the entire service life), this can be handled in a static calculation. KISSsoft has a simplified version of the strength calculation process, specifically to cover this situation. This is based on the ISO method, but only takes into account the nominal stress in the tooth root (without stress correction factor YS). Here you must note, that in this case, you must maintain a minimum safety level of 1.5 with regard to the material's yield point!

23.2.12 Take shot peening data into account in calculating the strength of toothed gears

On page 47 of AGMA 2004-B89 you will see a note about shot peening. This states that shot peening improves tooth root strength by 25%.

If you are using KISSsoft to perform calculations in accordance with DIN or ISO, you can achieve the increase in strength due to shot peening by inputting the corresponding technology factor. To do this, go to the Factors tab and click on "Z-Y factors..." in the General factors group.

You will find the details of useful entries as specified in Linke, Bureau Veritas/RINA or ISO 6336 in the manual. If you want to perform the calculation in accordance with AGMA, you do not have the option of inputting the technology factor. In this case, you must increase the foot endurance limit by inputting the corresponding percentage rate directly when you enter the material data. To do this, go to the Basic data tab and then click the plus button behind the material selection. In the dialog window, then activate "Own input". Input the endurance limit as shown in the following figure.

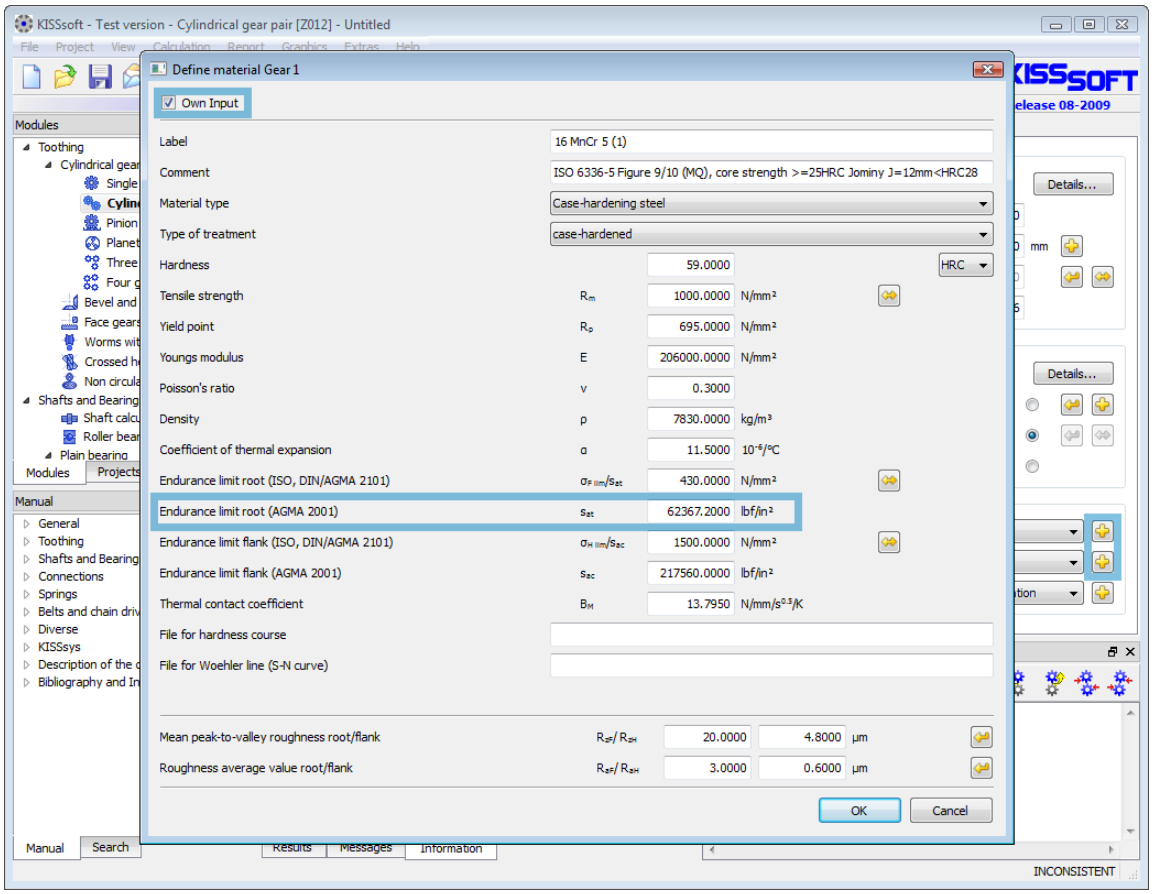


Figure 22.5: Material own input

23.2.13 Calculation in accordance with AGMA 421.06 (High Speed Gears)

In the KISSsoft system, you perform calculations as specified by AGMA 421.06 for high speed gears in the following way.

AGMA 421 is an old, well-established standard (1968), and has since been replaced by AGMA 6011-I03 (2003)

Please note the following points in section 52.6.14.

23.2.14 Comparison of a FEM calculation with spiral-toothed gear wheel calculation

The accepted wisdom is that the differing results in the tooth root stress were primarily due to the lower value of the "Reference Facewidth" in the KISSSOFT calculation.

The effective contact of spiral-toothed gear wheels is included in our calculation of the "Reference Facewidth". This results from the pressure ellipse (flattening of the point of contact) In addition, if sufficient facewidth is present, 1x module per facewidth is added to each side, as specified in ISO 6336-3.

23.2.15 Estimate the strength of asymmetrical spur gear toothings

At present, KISSsoft does not have any algorithms that can be used to perform a direct strength calculation for asymmetrical gears. Safeties are determined using the calculation methods in ISO10300 for hypoid gears (hypoid teeth are asymmetrical and have an unequal pressure angle on the right-hand and left-hand flank).

This procedure is described below:

The calculation is run twice, each time with a symmetrical tooth, once with a high pressure angle (calculation I), once with small pressure angle (calculation II).

The safety factor for the required safety against pitting that corresponds to the calculation with the flank under load is applied here. Therefore, if the load flank is the one with the small pressure angle, the safety against pitting from the calculation with the smaller angle (SHII) is used.

Root safety is determined with the nominal stress (tooth form factor YF), which is derived from the loaded flank. The tooth thickness at root sFn is determined from both these calculations, so therefore:

sFn = (sFnI + sFnII)/2

The stress concentration (factor YS) is calculated with the formula given above, and using the root radius and the application of force lever arm of the flank under load, and also sFn. All the remaining factors for defining the root fracture safety SF are the same.

23.2.16 Determine the equivalent torque (for load spectra)

Some calculation guidelines require you to determine the equivalent torque of a load spectrum and therefore perform sizing. How can I define the equivalent torque in KISSsoft?

The fundamental issue here is that the verification of a toothing with equivalent torque must give the same safeties as the verification with the actual load spectrum. For this reason, you can follow this procedure:

1. Input the load spectrum and calculate the toothing.

2. Make a note of the lowest root safety and the lowest flank safety for each gear.

3. In the `Module specific Settings` window, which you access from `Calculation` -> `Settings`, input the safeties you have noted as required safeties in the "Required safeties" tab. At this stage we recommend you deactivate the "`Securities depend on size`" tab.

4. Delete the load spectrum by setting "Individual load".

5. Then click the Sizing button next to the torque input field. This field is now filled with the equivalent torque.

6. Now run the calculation to check the data. The safeties you have now defined for the root or flank of a particular gear must be exactly equal to the previous smallest value (as in step 2). None of the gears can have a safety that is less than the safeties you recorded in step 2.

23.2.17 Check changes in safeties if the center distance changes

Is it possible to check how the safeties change when gears are mounted with a different center distance?

Select `Calculation`-> `Settings` ->`Module specific settings` in the `Calculations` tab and select `Calculation with operating center distance and profile shift according to manufacture`. You can then input the profile shift coefficients and center distance independently of

each other. The calculation then uses the circumferential forces in the operating pitch diameter instead of the circumferential forces in the reference circle.

23.2.18 Warning: "Notch parameter qs .... outside RANGE (1.0 to 8.0) ..."

Stress modification factor Y_S is calculated with a formula that complies with ISO 6336, Part 3 or DIN 3990, Part 3. This formula uses a notch parameter q_s, which is also documented in these standards:

$$q_s = \frac{s_{Fn}}{\frac{r_{OF}}{2}} \tag{22.4}$$

The validity area for the formula for Y_S in accordance with the standard lies in the range 1.0 ...q_s... 8.0. This formula should not be used outside this range.

If $q_s < 1$, Y_S (calculated with $q_s=1$), should be somewhat too large. In this case, the calculation results will fall in the validity area.

If $q_s > 8$, Y_S, (calculated with $q_s=8$) is rather too small. The calculation results in this case then fall outside the validity area. However, you should ensure that the calculation is not too imprecise.

23.3 Abbreviations used in gear calculation

Abb. in standards etc.	Abb. in KISSsoft	
a	a	Center distance (mm)
a_d	a.d	Reference center distance (mm)
A_a	A.a	Center distance allowance (mm)
As_e	As.e	Tooth thickness allowance at the normal section (mm)
α_{en}	alf.en	Angle at which force is applied (degree)
α_n	alf.n	Pressure angle at the normal section (degrees)
α_{Pro}	alf.Pro	Protuberance angle (degrees)
α_t	alf.t	Pressure angle on the reference circle (degrees)
α_{wt}	alf.wt	Operating pressure angle (degrees)
b	b	Facewidth (mm)
B_M	B.M	Thermal contact coefficient (N/mm/s.5/K)
β	beta	Helix angle at reference circle (degree)
β_b	beta.b	Base helix angle (degree)
c	c	Tip clearance (mm)
c'	c'	Single spring stiffness (N/(mm*µm))
c_γ	c.g	Contact spring stiffness (N/(mm*µm))
d	d	Reference diameter (mm)
d_a	d.a	Tip diameter (mm)
d_b	d.b	Base diameter (mm)
d_f	d.f	Root diameter (mm)
$d_f(x_E)$	d.f(x.E)	Root circle with profile shift for A_{se} (mm)
d_i	d.i	Inside diameter gear (mm)
d_{Na}	d.Na	Tip active circle diameter (mm)
d_{Nf}	d.Nf	Active root diameter(mm)
$d_{Ff(0)}$	d.Ff(0)	Root form diameter (mm)
d_{sh}	d.sh	External diameter of pinion shaft (mm)

d_w	`d.w`	Operating pitch diameter (mm)
D_M	`D.M`	Theoretical ball/pin diameter (mm)
	`D.M eff`	Effective ball/pin diameter (mm)
e_{fn}	`e.fn`	Normal gap width on the root cylinder (mm)
η_{tot}	`eta.tot`	Total efficiency
ε_α	`eps.a`	Transverse contact ratio
ε_β	`eps.b`	Overlap ratio
ε_γ	`eps.g`	Total contact ratio
f_f	`f.f`	Profile form deviation (mm)
$f_{H\beta}$	`f.Hb`	Helix slope deviation (mm)
f_{ma}	`f.ma`	Tooth trace deviation due to manufacturing tolerances (mm)
f_{pe}	`f.pe`	Single pitch deviation (mm)
f_{sh}	`f.sh`	Tooth trace deviation due to deformation of the shafts (mm)
F_a	`F.a`	Axial force (N)
$F_{\beta y}$	`F.by`	Actual tooth trace deviation (mm)
F_n	`F.n`	Normal force (N)
F_r	`F.r`	Radial force (N)
F_t	`F.t`	Nominal circumferential force in the reference circle (N)
	`Fase.d`	Tip chamfer (mm)
g_α	`g.a`	Length of path of contact (mm)
Γ	`Gamma`	Gamma coordinates (point of highest temperature)
h	`h`	Tooth depth (mm)
h_{aP}	`h.aP`	Addendum reference profile (in module)
h_F	`h.F`	Bending lever arm (mm)
h_{fP}	`h.fP`	Dedendum reference profile (in module)
h_k	`h.k`	Protuberance height (in module)
ha	`ha`	Chordal height (mm)

H	`H`	Service life in hours
I	`I`	AGMA: Geometry factor for pitting resistance
Impulse	`Impulse`	Gear driving (+) / driven (-)
j_n	`j.n`	Normal backlash (mm)
j_t	`j.t`	Rotational backlash (transverse section) (mm)
j_{tSys}	`j.tSys`	Backlash of the entire system (mm); for planetary stages
k	`k`	Number of teeth spanned
$k * m_n$	`k * m.n`	Tip alteration (mm)
K_A	`K.A`	Application factor
$K_{B\alpha}$	`K.Ba`	Transverse coefficient - scuffing
$K_{B\beta}$	`K.Bb`	Width factor - scuffing
$K_{B\gamma}$	`K.Bg`	Pitch factor - scuffing
K_f	`K.f`	AGMA: Stress correction factor
$K_{F\alpha}$	`K.Fa`	Transverse coefficient - tooth root
$K_{F\beta}$	`K.Fb`	Width factor - tooth root
$K_{H\alpha}$	`K.Ha`	Transverse coefficient - flank
$K_{H\beta}$	`K.Hb`	Width factor - flank
$K_{H\beta be}$	`K.Hbbe`	Bearing application factor
K_V	`K.V`	Dynamic factor
K_{wb}	`K.wb`	Alternating bending coefficient
l	`l`	Bearing distance l on pinion shaft (mm)
m_n	`m.n`	Normal module (mm)
m_{Red}	`m.Red`	Reduced mass (kg/mm)
m_t	`m.t`	Transverse module (mm)
M_{dK}	`M.dK`	Diametral measurement over two balls without backlash (mm)
M_{dKeff}	`M.dKeff`	Effective diametral measurement over two balls (mm)
M_{dReff}	`M.dReff`	Effective diametral roller mass (mm)
M_{rK}	`M.rK`	Radial measurement over one ball without

		backlash (mm)
M_{rKeff}	`M.rKeff`	Effective radial measurement over one ball (mm)
μ_m	`mu.m`	Average friction factor (as defined in Niemann)
μ_m	`my.m`	Averaged friction factor
μ_m	`my.my`	Friction factor
n	`n`	Speed (RpM)
ν_{E1}	`n.E1`	Resonance speed (min^{-1})
N	`N`	Nominal speed
N_L	`N.L`	Number of load changes (in mio.)
ν_{100}	`nu.100`	Kinematic nominal viscosity of oil at 100 degrees (mm^2/s)
ν_{40}	`nu.40`	Kinematic nominal viscosity of oil at 40 degrees (mm^2/s)
p_{bt}	`p.bt`	Base circle pitch (mm)
p_{et}	`p.et`	Transverse pitch on path of contact (mm)
p_t	`p.t`	Pitch on reference circle (mm)
P	`P`	Nominal power (kW)
$P_{V\,Z}$	`P.VZ`	Loss of power due to tooth load (kW)
$P_{V\,Ztot}$	`P.VZtot`	Total power loss (kW)
P_{WaelzL}	`P.WaelzL`	Meshing power (kW)
R_Z	`R.Z`	Average total height (mm)
ρ_F	`ro.F`	Tooth root radius (mm)
ρ_{fP}	`ro.fP`	Root radius reference profile (in module)
ρ_{Oil}	`ro.Oil`	Specific Oil density at 15 degrees (kg/dm^3)
s	`s`	Distance on pinion shaft (mm)
s_{an}	`s.an`	Normal tooth thickness on the tip cylinder (mm)
s_{Fn}	`s.Fn`	Tooth root thickness (mm)
s_{mn}	`s.mn`	Normal tooth thickness chord, without backlash (mm)
	`s.mn`	Effective normal tooth thickness chord with

	e/i	clearance (mm) (e: upper, i: lower)
S_B	S.B	Safety factor for scuffing (flash temperature)
S_F	S.F	Safety for tooth root stress
S_H	S.H	Safety for pressure at single tooth contact
S_{Hw}	S.Hw	Safety for flank pressure on operating pitch circle
S_{Sint}	S.Sint	Safety factor for scuffing (integral temperature)
S_{SL}	S.SL	Safety for transmitted torque (integral temperature)
σ_F	sig.F	(Effective) tooth root stress (N/mm^2)
σ_{F0}	sig.F0	Nominal tooth root stress (N/mm^2)
σ_{Flim}	sig.Flim	Endurance limit tooth root stress (N/mm^2)
σ_{FP}	sig.FP	Permitted tooth root stress (N/mm^2)
σ_H	sig.H	Flank pressure on the operating pitch circle (N/mm^2)
σ_{H0}	sig.H0	Nominal flank pressure on the pitch circle (N/mm^2)
$\sigma_{HB/D}$	sig.HB/D	Flank pressure HPSTC (N/mm^2)
σ_{Hlim}	sig.Hlim	Endurance limit Hertzian pressure (N/mm^2)
σ_{HP}	sig.HP	Permitted flank pressure (N/mm^2)
σ_s	sig.s	Yield point (N/mm^2)
Σx_i	Total x.i	Total profile shift coefficients
T	T	Torque (Nm)
θ_B	the.B	Highest contact temperature (^{o}C)
θ_{int}	the.int	Integral flank temperature (^{o}C)
θ_m	the.m	Tooth mass temperature (^{o}C)
θ_{M-C}	the.M-C	Tooth mass temperature (^{o}C)
θ_{Oil}	the.Oil	Oil temperature (^{o}C)
θ_s	the.s	Scuffing temperature (^{o}C)
θ_{Sint}	the.Sint	Scuffing integral temperature (^{o}C)

u	`u`	Gear ratio
v	`v`	Circumferential speed reference circle (m/s)
v_{ga}	`v.ga`	Maximum sliding velocity on tip (m/s)
	`Vqual`	Accuracy grade in accordance with DIN 3962 or ISO 1328
w	`w`	Nominal circumferential force reference circle per mm (N/mm)
W_k	`W.k`	Base tangent length (no backlash) (mm)
	`W.k e/i`	Effective base tangent length (mm) (e: upper, i: lower)
x	`x`	Profile shift coefficient
x_E	`x.E`	Profile shift coefficient at manufacturing for A_{se}
$X_{\alpha\beta}$	`X.alfbet`	Angle factor
X_B	`X.B`	Geometry factor
X_{BE}	`X.BE`	Geometry factor
X_{Ca}	`X.Ca`	Tip relief factor
X_e	`X.e`	Contact ratio factor
X_Γ	`X.Gam`	Distribution factor
X_M	`X.M`	Flash factor
X_Q	`X.Q`	Meshing factor
X_S	`X.S`	Lubrication factor (scuffing)
X_{WrelT}	`X.WrelT`	Relative structure coefficient (scuffing)
y_a	`y.a`	Run-in amount (µm)
y_b	`y.b`	Run-in amount (µm)
Y	`Y`	AGMA: Tooth form factor
Y_b	`Y.b`	Helix angle factor
Y_{drel}	`Y.drel`	Support factor
Y_e	`Y.e`	Contact ratio factor
Y_F	`Y.F`	Tooth form factor
Y_{NT}	`Y.NT`	Lifetime factor

Y_R	`Y.R`	Surface-factor
Y_S	`Y.S`	Stress correction factor
Y_{st}	`Y.st`	Stress correction factor test gear
Y_X	`Y.X`	Size coefficient (tooth root)
z	`z`	Number of teeth
z_n	`z.n`	Virtual gear no. of teeth
Z_β	`Z.b`	Helix angle factor
$Z_{B/D}$	`Z.B/D`	Single contact point factor
Z_E	`Z.E`	Elasticity factor ($N^{1/2}$/mm)
Z_ε	`Z.e`	Contact ratio factor
Z_H	`Z.H`	Zone factor
Z_L	`Z.L`	Lubricant factor
Z_{NT}	`Z.NT`	Lifetime factor
Z_R	`Z.R`	Roughness factor
Z_V	`Z.V`	Speed factor
Z_W	`Z.W`	Material hardening factor
Z_X	`Z.X`	Size factor (flank)
ζ_w	`zet.W`	Wear sliding coef. by Niemann
ζ_a	`zet.a`	Specific sliding at the tip
ζ_f	`zet.f`	Specific sliding at the root

Part III

Shafts and Bearings

Chapter 23
Defining Shafts

This program consists of a basic package and different expert add-ins. The following calculations are available here:

- Deformation, force, torque, and stress diagrams
- Eigenfrequencies (bending, torsion and axial movements)
- Buckling loads
- Static and fatigue strength
- Roller bearing calculation
- Sliding bearing calculation (hydrodynamic)
- Necessary width modification of pinion

Base package

In this module you can input and correct geometry and material data, shaft specifications, drawing numbers, support, peripheral conditions, external forces and moments (simplified input for couplings, spur and bevel gears, worms, worm gears, belt pulleys etc.).

A shaft with the machine elements mounted on it (for example, gears or bearings) is defined in the graphical shaft editor.
The properties required to define a shaft in this editor are:

- Any dimensions (cylindrical and conical), axially symmetric cross section, solid and hollow shafts, beams (H, I, L profiles etc.).
- Integrated drawing tool that allows simple corrections to be made to the shaft contour (diameter, lengths). You can change any of these elements by simply clicking on them with the mouse.
- Definition of notch geometries for the automatic calculation of notch factors. The following notch geometries are available here:
  - Radius
  - Chamfer
  - Relief groove
  - Interference fit

- Longitudinal keyway
- Circumferential keyway
- Square keyway
- V-notch
- Spline
- Cross hole

- You can enter these values for force and moment in any spatial positions. However, the following values are already predefined:
  - Cylindrical gear
  - Bevel gear
  - Worm
  - Worm wheel
  - Coupling
  - Rope sheave/V-belt
  - Centrical force
  - Eccentric force
  - External masses with moment of inertia (additional mass)
  - Power loss
- Calculation of:
  - Shaft weight
  - Moment of inertia
  - Axial force
  - Static torsion of the shaft

- Clear representation of geometry data and the calculated bearing and peripheral forces both on screen and on paper.

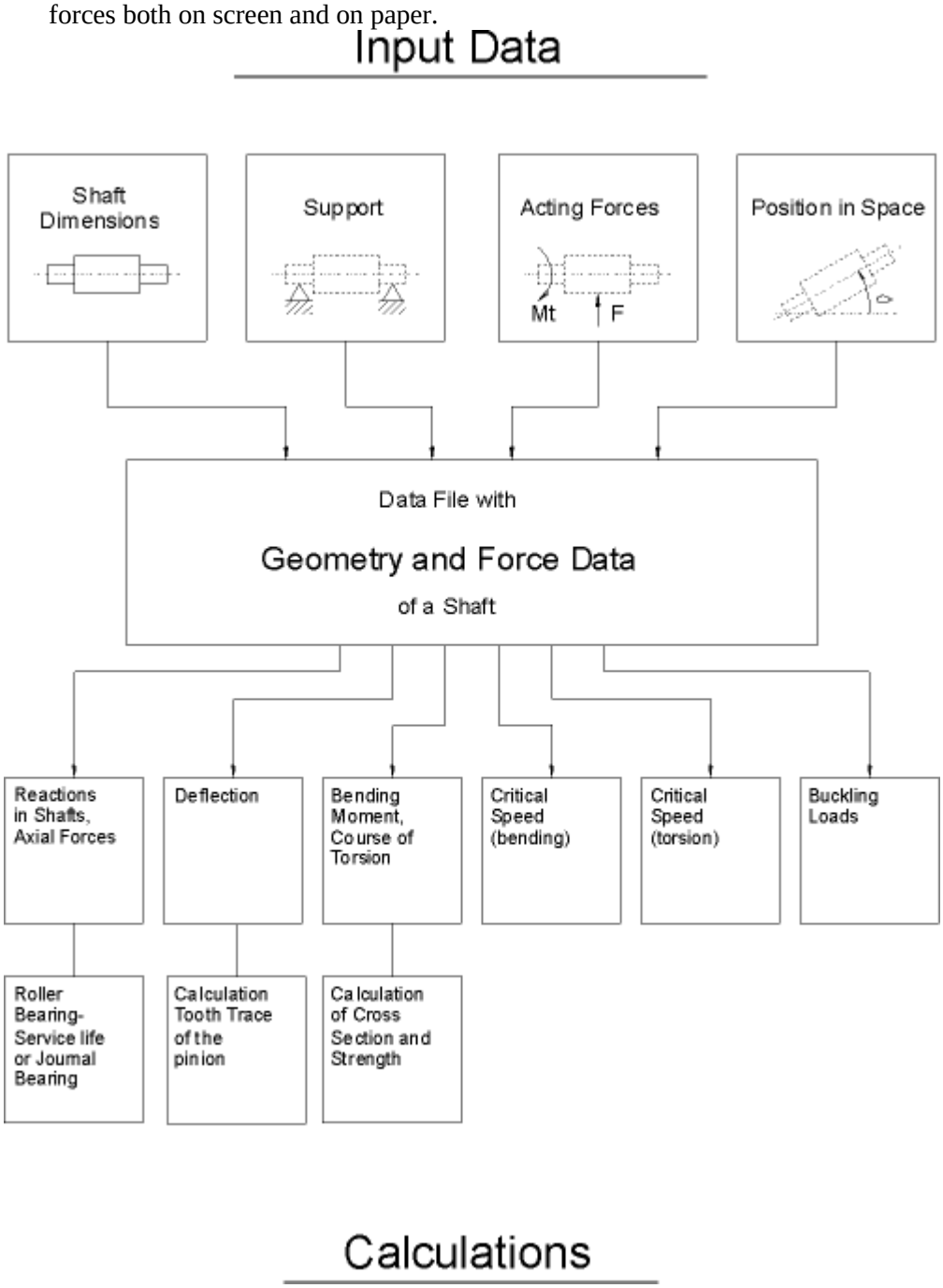


Figure 23.1: Flow-chart of the modules for shaft and bearing calculation in KISSsoft.

24.1 Input window

The KISSsoft system offers a range of different input windows in which you can define shafts. The Shaft editor (see page III-645) shows a graphical representation of the shaft system. The Elements tree (see page III-647) illustrates the structure of the shaft system in a tree structure. Outer contour (see page III-655), Inner contour (see page III-662), Forces (see page III-662), Bearing (see page III-667) and Cross sections (see page III-672) values for a shaft are shown as a table in the Elements list (see page III-648). You define the parameter of an element in the Elements editor (see page III-649).

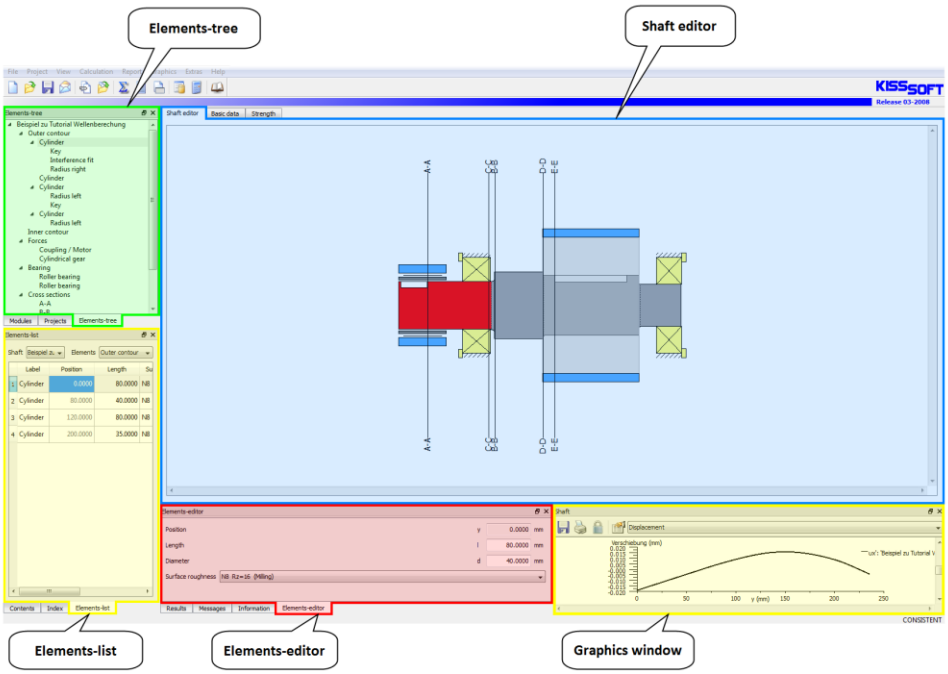


Figure: The different input windows where you can define shafts

24.1.1 Shaft editor

The shaft editor shows a graphical representation of the shaft system. Use the vertical tool bar on the right-hand edge of the shaft editor to add the most frequently used elements. If your system has several shafts, the new element is always added to the active shaft. A shaft becomes active when one of its elements is selected. If no element has been selected, the last shaft is the active one. The active shaft is also displayed in the Elements list (see page III-648).

Via the Context menu you can print the graphics in the shaft editor and save them as picture files. Each of the different elements also have interactive Context menus.

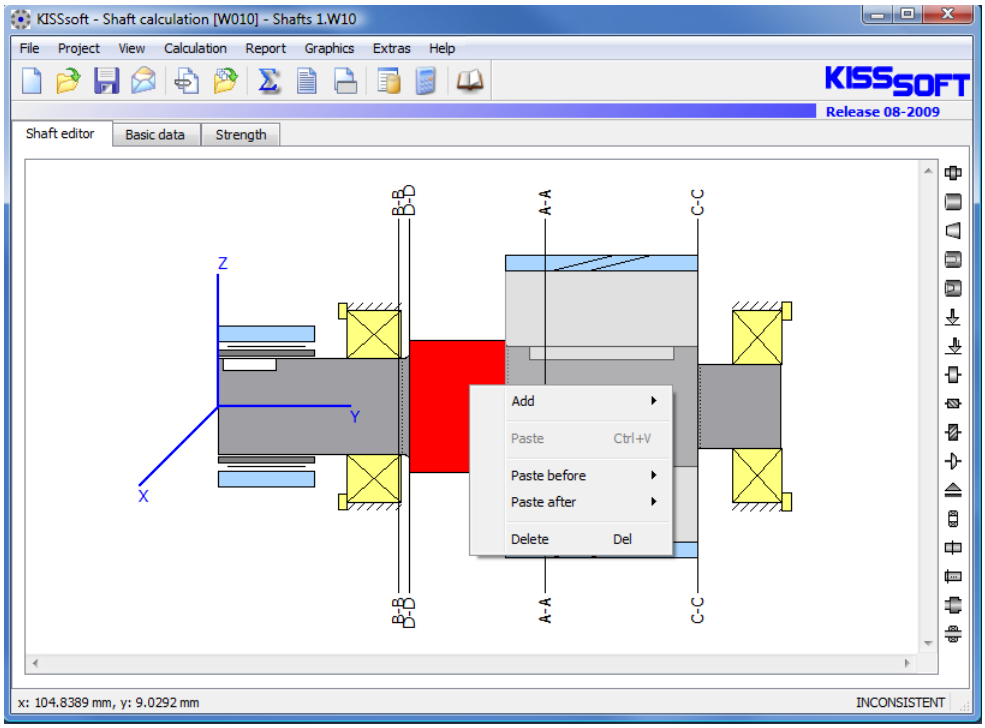


Figure: Context menu in the Shaft editor

24.1.2 Elements tree

The Elements tree illustrates the structure of the shaft system in a tree structure. Shafts are at the highest level. The connecting elements in systems with several shafts are also shown here. Each shaft groups its main elements by Outer contour (see page III-655), Inner contour (see page III-662), by Strength (see page III-656), Bearings (see page III-667)and Cross sections (see page III-672). For the cylinder and conus main elements, the sub-elements are located on a further sub-level.

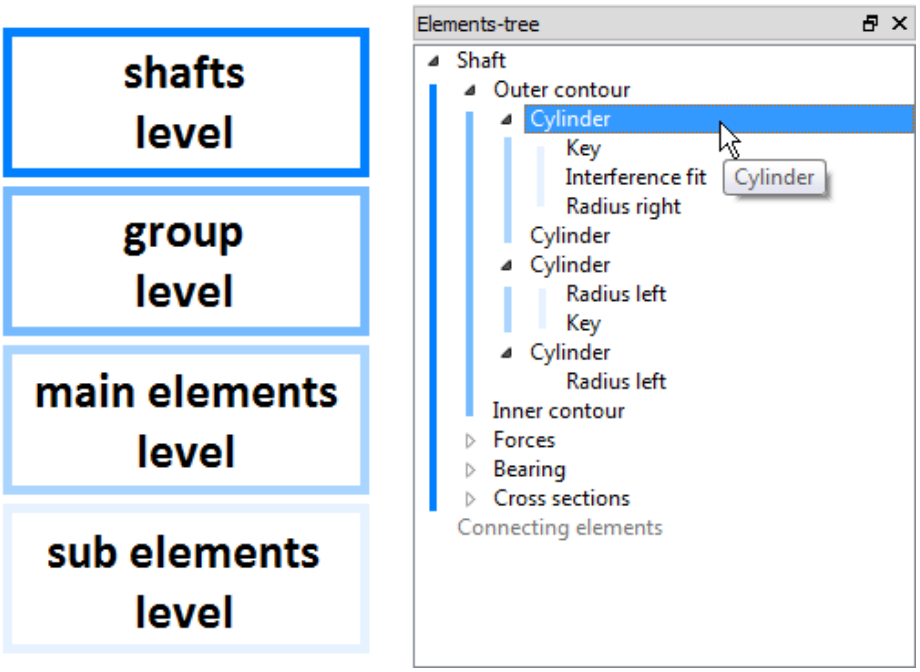


Figure: Levels in the Elements tree

You can select, copy, insert and delete elements via the Elements tree. In a Context menu you see which actions are available for each element. Special actions are available, depending on the element type. You can also arrange shafts, roller bearings and cross sections. You can also import (see page III-660)/export (see page III-662) outer and inner contours to DXF.

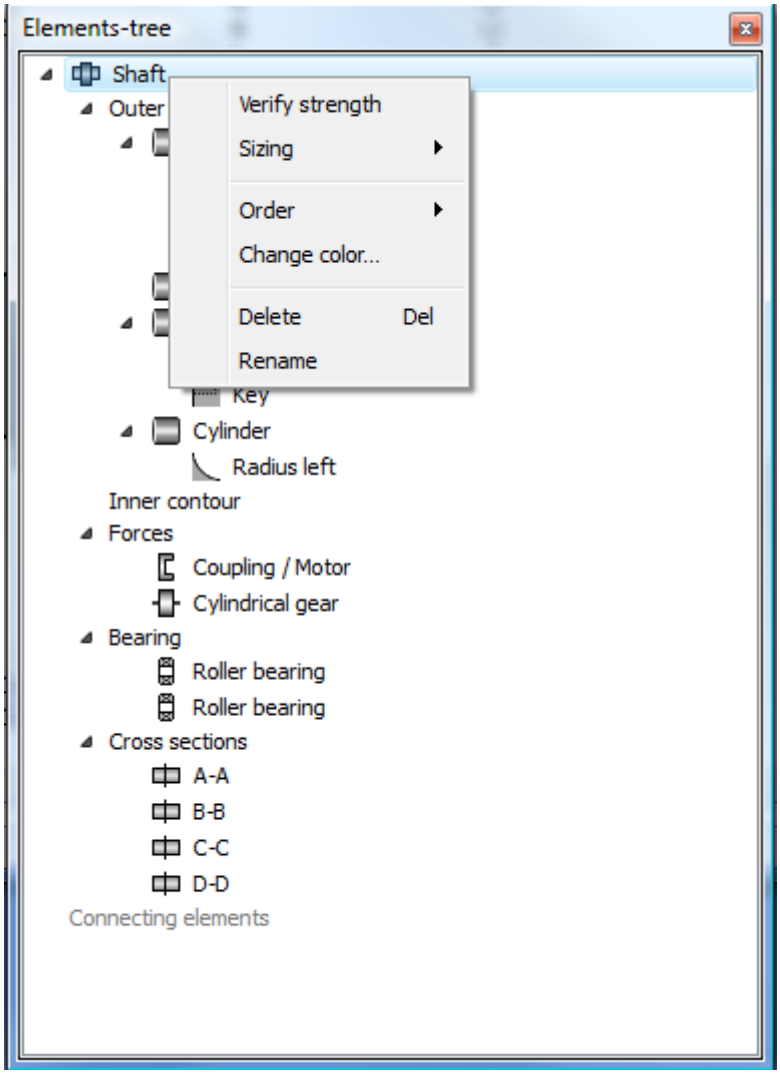


Figure: Context menu in the Elements tree

24.1.3 Elements list

The Elements list lists groups of elements in table format. Two selection lists show the active shaft and the currently displayed elements. You can edit the parameter

listed in the table directly in the Elements list. The context menu allows you to insert elements quickly and easily.

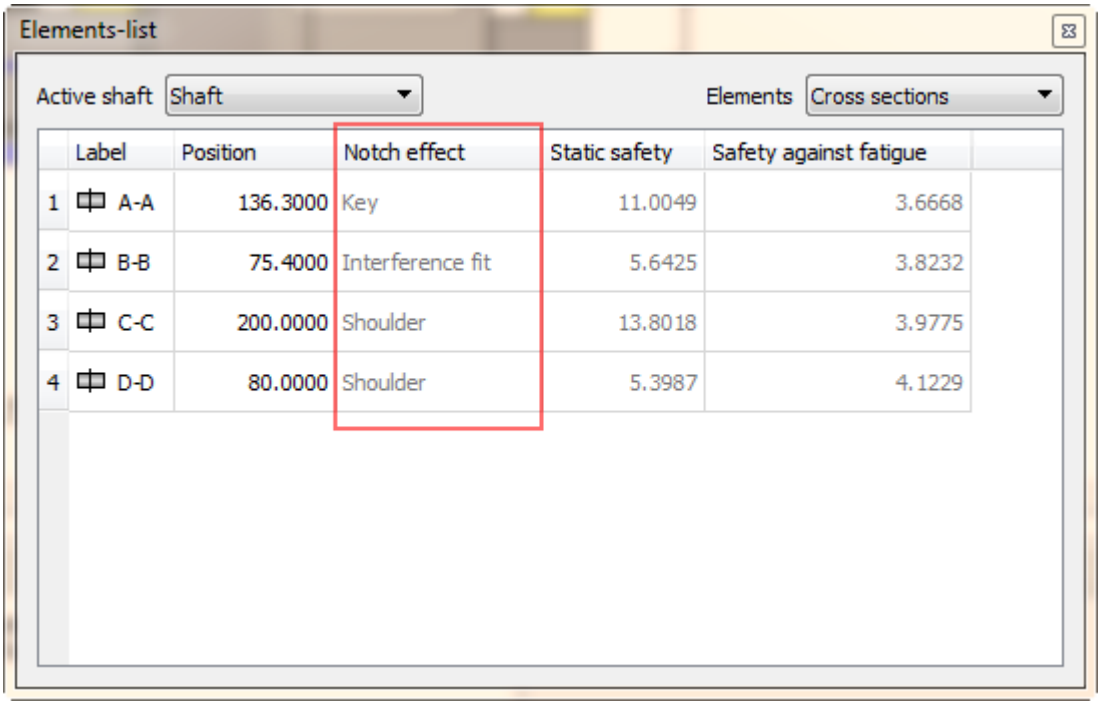


Figure: Context menu in the Elements list

24.1.4 Elements editor

In the Elements editor you can edit any of the parameters of the selected element.

24.2 Element overview

24.2.1 The Shaft element

To input a shaft, click on the first icon in the vertical tool bar in the Shaft editor (see page III-645). You will also find the `Add shaft` option in the context menu of the Elements tree (see page III-647). A new entry appears at the end of the Elements tree. Single click on the shaft element in the Elements tree to input parameters for the shaft in the Elements editor (see page III-649), as shown in Figure 23.4).

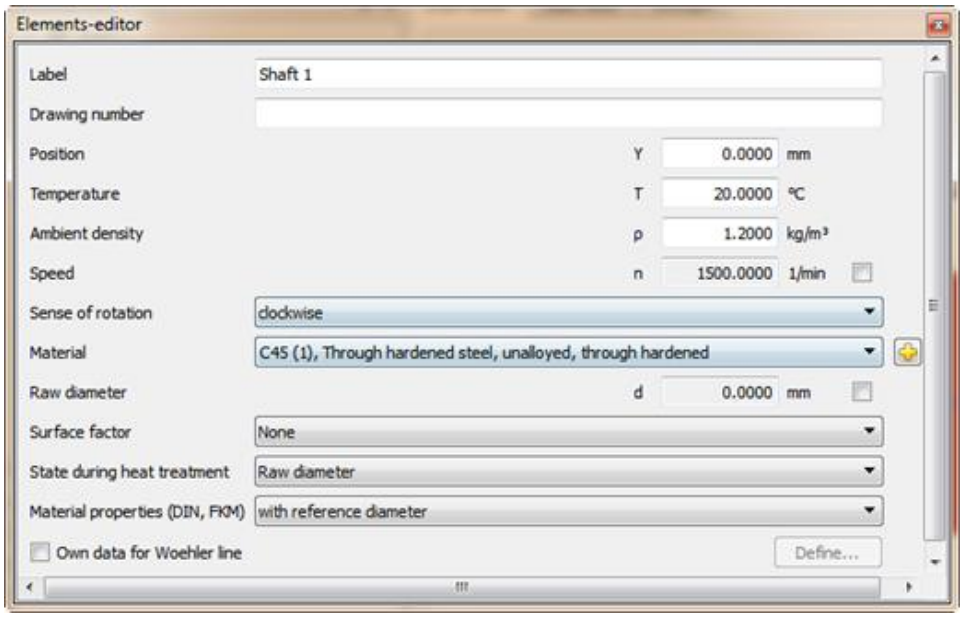


Figure 23.4: `Elements editor` for inputting shaft parameters

The next section describes the individual input fields in which you enter parameters for a specific shaft.

24.2.1.1 Drawing number

In the `Drawing number` input field you can enter a string of any characters apart from ";" (semicolon). The drawing number you enter here does not affect the calculation.

24.2.1.2 Position

The `Position` input field is where you enter the `Y` coordinate of the starting point of the shaft with regard to the global co-ordinates system.

NOTE

Global coordinates are indicated by upper case letters. Lower case letters indicate a shaft's local coordinate system.

24.2.1.3 Temperature

The shaft may undergo thermal expansion if the shaft's temperature is not the same as the Reference temperature (on page III-675). In addition to the thermal expansion of the shaft, the thermal expansion of the gear case can also be taken into account by the Housing temperature (see page III-676).

24.2.1.4 Ambient density

Bodies placed in hydrostatic fluids experience buoyancy. The value here is the same as the weight of the displaced medium, and is defined by the volume and the density of the displaced medium. KISSsoft takes the buoyancy effect into account, if you enter the appropriate ambient density value. The default setting is for air density. The next table lists technical values for other media.

Medium	**Air**	**Water**	**Oil**
Density ρ	1.2	998	772

Table 23.1: Densities [kg/m^3] of a few important fluids where ϑ = 20°C and p = 1016 mbar

NOTE

If a shaft is operated in different ambient media, for example, as is the case for input shafts in ships, you can combine two individual shafts, each of which has different ambient density data, by using the `Connections` element in the Elements tree and calculate them as a single shaft.

24.2.1.5 Speed

Shaft speed [1/min] around its longitudinal axis. If you click the checkbox to the right of the input field, you can change the speed independently of other shafts.

However, if this checkbox is not active, the value is taken from the `Speed (see page III-674)` input field in the `Basic data` input window.

24.2.1.6 Sense of rotation

The sense of rotation can influence the way loads are distributed along the shaft, for example, as the result of helical toothed gears, and therefore affect the service life of the bearing. Click the checkbox to the right of the `Speed` input field to view and select these entries from the drop-down list. However, if this checkbox is not active, the value is taken from the `Sense of rotation (see page III-675)` input field in the `Basic data` input window.

24.2.1.7 Material

You can select a shaft material from this drop-down list and therefore assign a specific material to each individual shaft. If you use this function together with the `Connections` element in the Elements tree you can generate shafts made of different materials.

24.2.1.8 Base size

The `Base size` input field is decisive for strength calculation. However, if you select the `Pre-turned to actual diameter` option in the `Strength` input window in the `State during heat treatment` drop-down list, the setting of the raw measure value has no effect on the calculation. In contrast, if the selection is set to `Raw diameter`, the largest, rounded shaft diameter will be selected and the strength calculation will be performed using this value. Click the checkbox to the right of the input field to specify your own diameter for the blank before it is turned.

24.2.1.9 Surface factor

In this selection list, you can define if an additional surface factor should be applied or not. Here you can select either `Rollers` or `shot peening`.

24.2.1.10 State during heat treatment

To define the technological size coefficient $K_{1,\mathrm{deff}}$, select one of these two options:

- Pre-turned to actual diameter. The raw diameter has no influence on the technological size coefficient. The value $K_{1,deff}$ is recalculated for each cross section based on the actual diameter size.
- Raw diameter. $K_{1,deff}$ is determined once from the raw diameter and applied cross section.

NOTE

You can also define the `Base size` field in the `Elements editor` of the corresponding shaft. To do this, input the dimension of the raw material which was used to generate the final material characteristics during the last heat treatment. If this involves a solid shaft, enter the external diameter of the unworked part. For a pipe, enter the wall thickness and, for a cast part, enter the greatest wall thickness.

24.2.1.11 Material properties

From the `Material characteristic values` drop-down list, specify how KISSsoft is to define the material characteristic values that are relevant to strength.

1. `at reference diameter` Values are taken from the database and multiplied by K_1
2. R_p, R_m `as stated in database, SW for reference diameter` The values R_p and R_m are determined according to size (excluding K_1), and the fatigue strength σ_W is determined for the reference diameter entered in the database and then it is multiplied with K_1.
3. R_p, R_m `as stated in database, σW constant` The values R_p and R_m are determined according to size, and the fatigue strength σ_W is taken from the database without being influenced by the geometric size factor. The size coefficient K_1 is not taken into account here.
4. R_p, R_M `according to the database, σW is calculated from` R_m The values R_p and R_m are taken from the database, and σ_W is determined from the tensile strength R_m in accordance with the standard.

The data of the material used to calculate the shaft strength is derived from the values in the database as follows:

- Fatigue limit factors (for tension/compression, bending, etc.) are taken directly from the material database. There, these values are defined for every calculation method. If data for these materials has been specified in the calculation method, it is these values that are used.
- Tensile strength values are stored in the database according to their diameter as defined in the specific EN standard. The raw diameter is used to fetch the

tensile strength value from the database and use this in the calculation. This method of defining the actual tensile strength is very reliable and can be used for every calculation method. It has the effect that the same values are used for each calculation method.

When you specify a calculation method, you can decide to use the material database on the basis of the requirements given in the corresponding standard. Then, the actual tensile strength is defined using the thickness factor taken from the base tensile strength of the sample diameter (normally 10 mm), in accordance with the standards (either FKM or DIN: if you use Hänchen this triggers an error message).

- The yield point or strain limits are taken either from the database or from the standard, in the same way as for the tensile strength.

24.2.1.12 Own data for Woehler line (S-N curve)

Click the `Own data for Woehler line` checkbox to define your own Woehler line. You can also enter values for the sustainable damage or Miner total here. If you do not activate this checkbox, the program will define the Woehler line in accordance with either DIN 743 or FKM. You should specify your own Woehler line, or modify the sustainable damage value if you are modifying your calculation to suite the results of specific tests.

24.2.1.13 Taking the results into account in the report

If this flag is set, the corresponding shaft is output in the main shaft report, along with all its elements (external/internal contour, force elements, bearing). However, this is only valid for inputs and does not affect the results of the calculation. The default setting is that this flag is set.

24.2.2 Outer contour

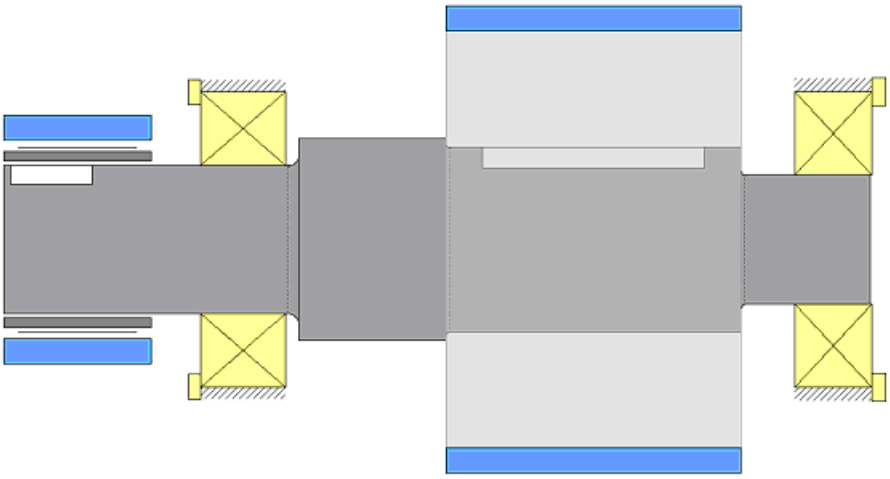
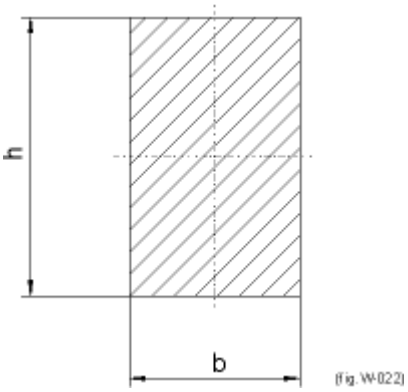


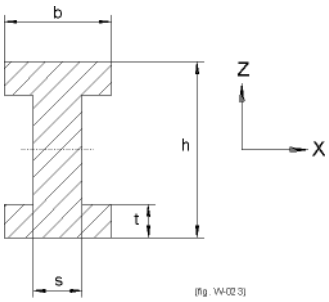
Figure 23.5: Display the outer contour in the `Shaft editor`

You can use (hollow) cylinders, (hollow) cones and beams to define the shaft geometry. To enter a new element, select the element you want at group level in the Elements tree, e.g.`Outer contour`. Click the right-hand mouse button on this element to add it to the group at the right-hand end of the shaft. Alternatively, you can select an existing element at element level (e.g. `cylinder`) and then right-click with the mouse to open a context menu. The `Add element before(after)` option opens another sub menu in which you select an element to be inserted at a position relative to the existing element.

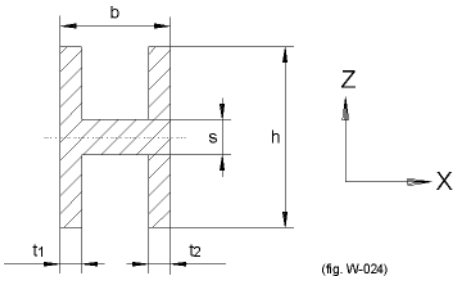
Possible profiles for beams are:



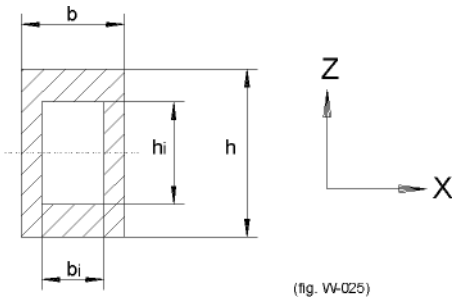
Rectangular profile



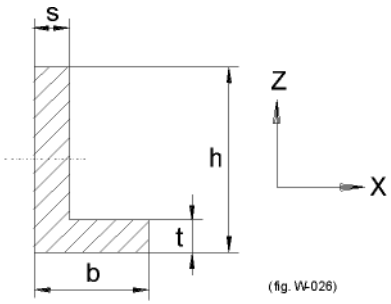
Double T profile



H profile



Rectangular profile (hollow)



L profile

24.2.2.1 Defining sub elements

Before you can define a sub element, first select the main element to which you want to add this sub element in the Elements tree. Then right-hand mouse click to select the sub element you require. The inserted sub element now appears in the Shaft editor and its corresponding notch factors are defined in the strength calculation. Once you have defined a sub element, you can activate it in the same way as a main element (see Activate).

Adding sub elements:

- **Radius right/ left**
  Input values:
  - Radius: Size of the radius
  - Surface roughness: Radius surface
- **Chamfer right/left**
  Input values:
  - Length: Chamfer length
  - Angle: Chamfer angle

- **Relief groove right/left**
  Input values:
  - Relief groove form: Select the relief groove form in accordance with DIN 509 or FKM
  - Series (DIN 509): (Selection: series 1, radii as defined in DIN 250; Series 2, special radii)
  - Stress (DIN 509): (with conventional stress; with increased fatigue strength)
  - relief groove length: Length of the relief groove in the direction of the axis
  - Transition radius: Radius between the end of the relief groove and the next element
  - Depth of recess: Recess depth
  - Surface roughness: Recess surface
- **Interference fit**
  Input values:
  - Interference fit length: Interference fit length
  - Type of interference fit: (Selection: Slight interference fit, interference fit and interference fit with end relief)
  - Reference measure: this specifies the measurement from the left-hand end of the selected element up to the start of the thread
- **Longitudinal groove**
  Input values:
  - Groove length: Groove length
  - Surface roughness: Groove length surface
  - Reference measure: this specifies the measurement from the left end of the selected element up to the start of the groove
- **Circumferential groove**
  Input values:
  - Depth: Depth of the circumferential groove
  - Rounding in the groove bottom: Radius of the circumferential groove
  - Surface roughness: Surface of circumferential groove
  - Reference measure: this specifies the measurement from the left end of the selected element up to the middle of the circumferential groove

- **Square groove**
  Input values:
  - Width: Width of the square groove
  - Depth: Depth of the square groove
  - Radius: Radius of the square groove
  - Surface roughness: Surface of the square groove
  - Reference measure: this specifies the measurement from the left end of the selected element up to the middle of the square groove
- **V-notch**
  Input values:
  - Depth: Depth of the V-notch
  - Surface roughness: Surface of the V-notch
  - Reference measure: this specifies the measurement from the left end of the selected element up to the middle of the V-notch
- **Spline**
  Input values:
  - Standard: Normal range of the spline (click the button to select the required size from a list)
  - Tip circle: you can either select this from a list of standards or input your own value
  - Root circle: you can either select this from a list of standards or input your own value
  - Number of teeth: you can either select this from a list of standards or input your own value
  - Module: you can either select this from a list of standards or input your own value
  - Surface quality: Spline surface quality
  - Length: Spline length
  - Reference measure: this specifies the measurement from the left end of the selected element up to the start of the spline
- **Spline shaft**
  Input values:
  - Tip circle: Tip circle of the spline shaft

- Root circle: Root circle of the spline shaft
- Number of keys: Number of keys
- Spline shaft root rounding: (Selection: Shape A, Shape B and Shape C)
- Length: Length of the spline shaft
- Reference measure: this specifies the measurement from the left end of the selected element up to the start of the spline shaft
- Surface quality: Spline shaft surface

Cross hole
Input values:

- Bore diameter: Diameter of bore
- Surface roughness: Axial boring surface
- Reference measure: this specifies the measurement from the left end of the selected element up to the position of the cross hole

Thread
Input values:

- Label: Thread label
- Thread depth: Thread depth
- Rounding: Rounding in the notch bottom of the thread
- Length: Thread length
- Reference measure: this specifies the measurement from the left-hand end of the selected element up to the start of the thread
- Surface roughness: Thread surface

General notch effect
Input values:

- Width: Width of the overall sub element
- Notch factor bending/ torsion/tension-compression/shearing force: you can enter the notch factors directly here.
- Surface roughness: Surface of the overall sub element
- Reference measure: this specifies the measurement from the left end of the selected element up to the middle of the overall sub element

You can enable the "Conical shoulder" notch type directly in the Strength calculation (see section "Cross section types" on page III-712).

24.2.2.2 Importing the shaft geometry

Right-hand mouse click next to the outside or inner contour to open a pop-up menu (see figure). Click `Import` to import a .ktx or a .dxf file.

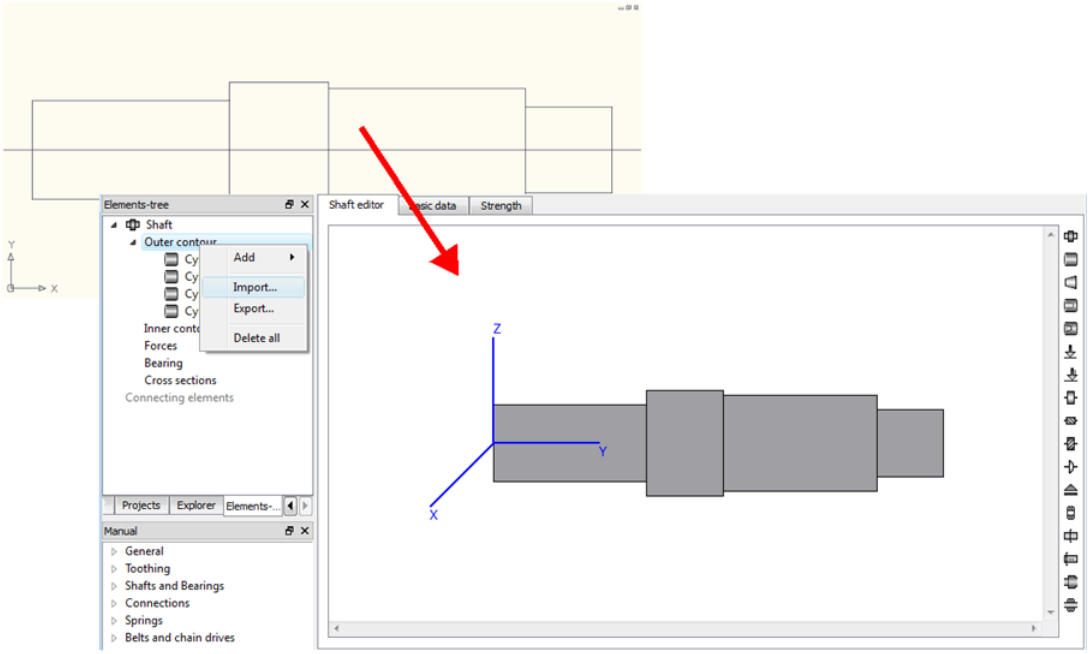


Figure 23.6: Importing the shaft geometry from a dxf file

Reading (importing) a ktx file:

In KISSsoft, go to the Shaft calculation Elements tree and right-hand click on the `Outer contour` element to open a pop-up menu in which you select the `Import` option. Select the required *.ktx file and click on `Open`. The shaft contour is now uploaded into KISSsoft.

Reading (importing) a dxf file:

The outside and inner contour (if present) of the shaft should be output individually by the CAD system.

NOTE:

You can use the default value **ALL** for the layer name so that all layers are imported. You can also import the contours as variants in different layers. To do this, enter the layer name in the appropriate input field. If you don't know the exact

layer name, you can input an invalid name as a test (for example, xxx) If you then try to read (import) this, the resulting error message will list the valid layer names.

- Draw the shaft contour with a mid line in a CAD system. Use the x, y plane as the coordinates system (X-axis as rotational axis) to ensure the contour is interpreted correctly after it has been read imported and so that the shaft is drawn in KISSsoft in the Y, Z plane (rotational axis Y-axis). Save the shaft geometry as a *.dxf file.
- In KISSsoft, go to the Shaft calculation Elements tree and right-hand click on the `Outer contour` element to open a pop-up menu in which you select the `Import` option. Now select the *.dxf file you require and click `Open`.
- This opens another dialog in which you can define the layer, the point of origin (x/y) and the angle of the symmetry axis. After you have input this data, click OK to close this dialog. The shaft contour is then loaded with these details.

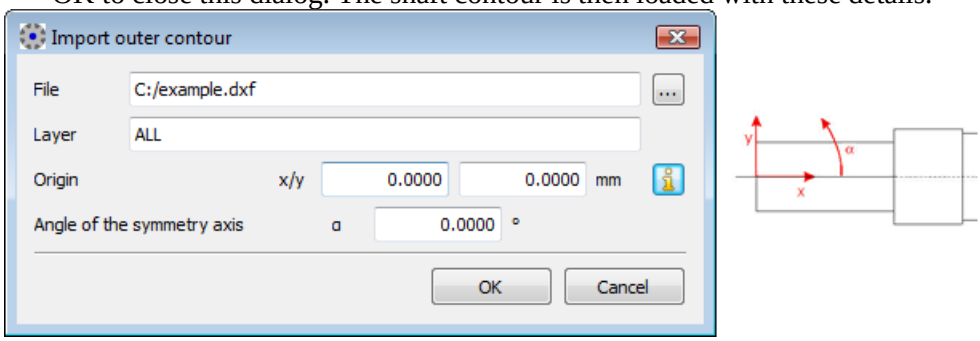


Figure 23.7: Import dialog for loading dxf files

24.2.2.3 Export shaft geometry

Right-hand mouse click next to the outside/ or inner contour to open a pop-up menu (see figure). If you select `Export`, you can create either a *.ktx or *.dxf file.

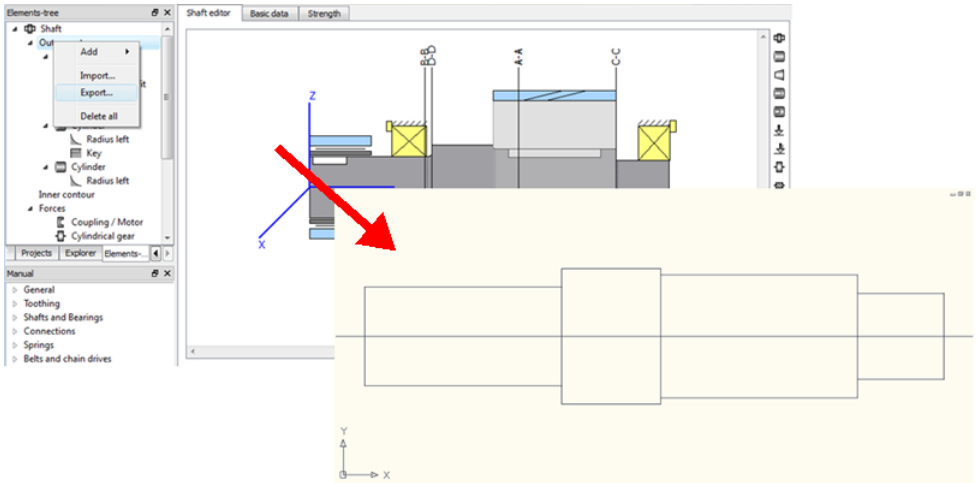


Figure 23.8: Export shaft geometry in a dxf file

Procedure for importing in a file:

- You can export previously-defined shafts from the Shaft editor. In the KISSsoft Elements tree for shaft calculation, right-hand mouse click on the required element e.g. `Outer contour`, start the Popup menu start and select `Export`. You can export inside or outer contours of the different shafts.
- After you select a contour, a dialog opens in which you can define the name of the *.ktx or *.dxf file.

24.2.3 Inner contour

The inner contour is generated from left to right (just like the outer contour). For example, if you want to generate a shaft with an axial hole from the right-hand side, you must first input data for an inside cylinder starting from the left-hand side with a diameter of 0 that extends up to the point where the bore begins.

24.2.4 Forces

24.2.4.1 Forces

Forces can be applied arbitrarily to any point on the shaft and even outside(!) the shaft. Different methods are available for defining force transmitting elements

(such as gears) or even individual forces. In most forces, the direction of the torque is defined as "driving"/"driven". "Driving" means that the shaft is the driving element or that the torque of the sense of rotation is counter to the sense of rotation. See also 23.3.5 (see page III-675).

Comments about special elements:

- Cylindrical gear
  Position of contact: specify the location of the point of contact with the paired gear in accordance with Figure 23.3 on page III-673 (this point is where the forces apply).
  Instead of simply entering the reference diameter, you get a more accurate result if you enter the operating pitch diameter and the operating pressure angle instead of the nominal pressure angle.
  Click the Convert button to calculate these values.
- Bevel gear
  Position of contact: refer to the data for cylindrical gears.
  An additional force component due to friction ($\mu = 0.05$) is taken into account when calculating hypoid gears.
- Face gear
  For face gears, the reference cone angle is always set to 90° (this input cannot be changed).
- The worm
  is usually a driving element. Its efficiency is included in the calculation of force components. Position of contact: refer to the data for cylindrical gears.
- The worm wheel
  is usually "driven". Its efficiency is included in the calculation of force components. Position of contact: refer to the data for cylindrical gears.
- Rope sheave
  Direction of rope pulley: Input the direction of the resulting belt forces as shown in Figure 23.3 on page III-673.

The direction of the helix angles and the positions of the elements are defined in Figure 23.9.

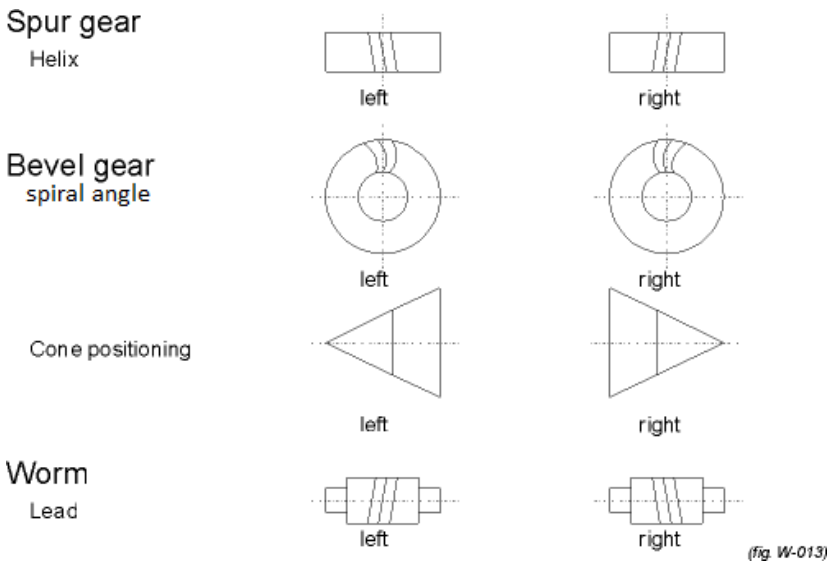


Figure 23.9: Defining the direction of force elements.

Eccentric force

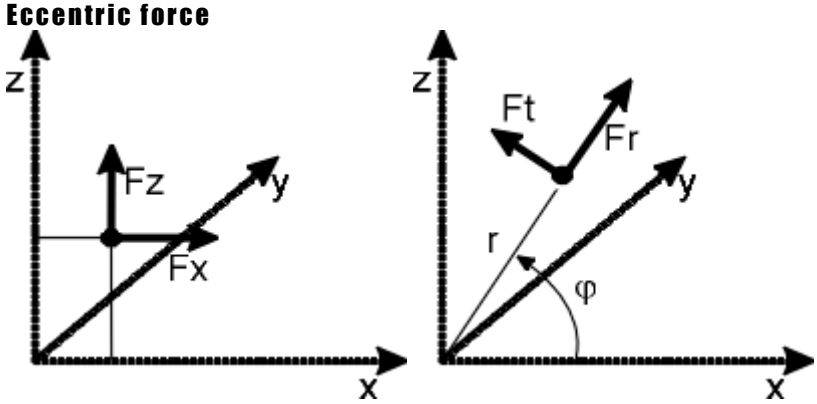


Figure 23.10: Cartesian/polar coordinates for eccentric force

You can enter values for eccentric force either in Cartesian or polar coordinates (see Figure23.10). You can change the coordinates system in the `Drawings/Settings` tab in the Shaft editor.

Transferring data from gear calculation

In the `Elements editor`, you can import the data used to define spur and bevel gears from a gear calculation file. Select the element you require in the Elements tree and then click on the `Read data from file` checkbox. Then select the gear number (1 to 4). The data relevant to this gear pair is then imported directly. In this situation, the data at the pitch point is used, and not the data at the reference circle.

Important: If the Read data from file option in this input window remains active, data will be imported again from the gear calculation every time you call the shaft calculation function. If you then change the gear data later on, the new data will automatically be transferred along with it! If the flag is not set, the data is only copied once from the gear calculation and not updated later on. For this reason, in the input screen with the gear data, when there are linked files, you cannot change the contents of most input fields, except the Position of contact and the Y-coordinate.

24.2.4.2 Coupling

A coupling transmits torque and can also be subject to radial and axial forces. From the torque (or the specified power and speed) you can calculate the circumferential force to

$$F_t = \frac{M_t \cdot 2}{d} \tag{23.2}$$

F_t = Circumferential force

M_t = Torque

d = Effective diameter

Calculating radial force for a coupling:

$$F_r = K_2 \cdot F_t \tag{23.3}$$

F_t = Circumferential force

K_2 = Radial force factor

Define the direction of the force in the input window. You are also prompted to enter the mass of the coupling so it can be included in the calculation as a gravitational force.

Calculating axial force for a coupling:

$$F_a = K_3 \cdot F_t \tag{23.4}$$

F_t = Circumferential force

K_3 = Axial force factor

The axial force acts along the center line of the shaft.

24.2.4.3 Mass

Masses placed on the shaft are used as moments of inertia to determine the critical speeds. They are to be considered as a gravitational force.

24.2.4.4 Magnetic tension

Radial and axial forces produced by electromagnetic windings are included in the calculations.

Calculating radial force:

$$F_{mr} = K_1 \cdot D \cdot L \cdot v \cdot \frac{f}{del0} \tag{23.5}$$

K_1	=	0.1 for three-phase motors where the number of poles is 2 0.2 otherwise
D	=	(mm) inside diameter of the stator of three-phase motors or external diameter of the rotor of direct current motors
L	=	(mm) Length of the active "packet of plates" (excluding the cooling slits)
v	=	Damping factor: Three phase current asynchronous motor: Squirrel cage: v = 0.25

		Three phase current asynchronous motor: Wound rotor: v = 0.7 Three-phase current synchronous motor v = 0.5 DC machine with wave formation v = 1.3
f/del0	=	Ratio of the mean eccentricity and the nominal air gap = 0.2 for AC machines = 0.1 for DC motors

Calculating axial force

K_3	=	35 . α/D
K_3	=	Axial force factor
T	=	Torque (Nm)
α	=	Axial groove lead (deg)
D	=	(mm) inside diameter of the stator of three-phase motors
		or external diameter of the rotor of direct current motors

24.2.5 Bearing

In addition to calculating the shaft, you can export roller bearings and general bearings as separate roller bearing or plain bearing files (`File > Export`).

24.2.5.1 General bearing

All elements of a bearing (rigid or elastic) are considered to be a bearing. Input a fixed bearing, right mounted, left mounted, or thrust, bearing to determine the point on the shaft at which axial force is transmitted. This information is also used in the roller bearing calculation. In taper roller bearings (or similar configurations) it is not always obvious which bearing is subject to the axial force. In this case, you must enter the mounting data for the bearings. You can also specify a radial or axial offset in the bearing alignment. This enables you to take into account other factors such as the simulation of assembly error.

24.2.5.2 Roller bearings

In addition to general bearings, you can also select specific roller bearings. The bearing data is then taken from the bearing database. This means the bearing's

geometry data is already available and you can draw the bearing using the width and external diameter values. In addition, for a bearing with an inclined contact angle, the direction of the force can be taken into account in the calculation. You can either use the bearing stiffness value taken from the bearing database (if available) or specify your own value here. You can also define a bearing clearance for each roller bearing (according to DIN 620 C2, C0, C3, C4 or Own input).

A preload force, applied on the outer ring, can be used instead of the offset to define the preload force on the bearing. This is only taken into account for bearings with inner geometry, and only if the corresponding bearing can accept an axial preload force. Additionally, for bearings with inner geometry, a rotation around axis X and Z of the outer ring can be specified by the user. This could for example be used to model the housing deformation, and enables the user to enter the FEM results directly.

The diametral pitch defines the diameter-related clearance of a bearing. For a deep groove ball bearing, the diametral pitch clearance is defined as:

Pd = do – di – 2 * Dw

Where Pd is the diametral pitch bearing clearance, do is the external race diameter, di is the internal race diameter and Dw is the rolling body diameter. Similar definitions are used for other bearing types, depending on which particular type is involved.

24.2.5.3 Constraints on various bearings

Options for selecting a `roller bearing` with displacement and rotation options:

Roller bearing selection list	**ux**	**uy**	**uz**	**rx**	**ry**	**rz**
Non-locating bearing	fixed	non-locating	fixed	non-locating	non-locating	non-locating
Fixed bearing adjusted on both sides <->	fixed	fixed	fixed	non-locating	non-locating	non-locating
Fixed bearing adjusted on right side ->	fixed	right	fixed	non-locating	non-locating	non-locating
Fixed bearing adjusted on left side <-	fixed	left	fixed	non-locating	non-locating	non-locating

Thrust bearing attached on both sides <->	non-locating	fixed	non-locating	non-locating	non-locating	non-locating
Thrust bearing attached on right side ->	non-locating	right	non-locating	non-locating	non-locating	non-locating
Thrust bearing attached on left side <-	non-locating	left	non-locating	non-locating	non-locating	non-locating

Options for selecting a `Bearing (in general)` with displacement and rotation options:

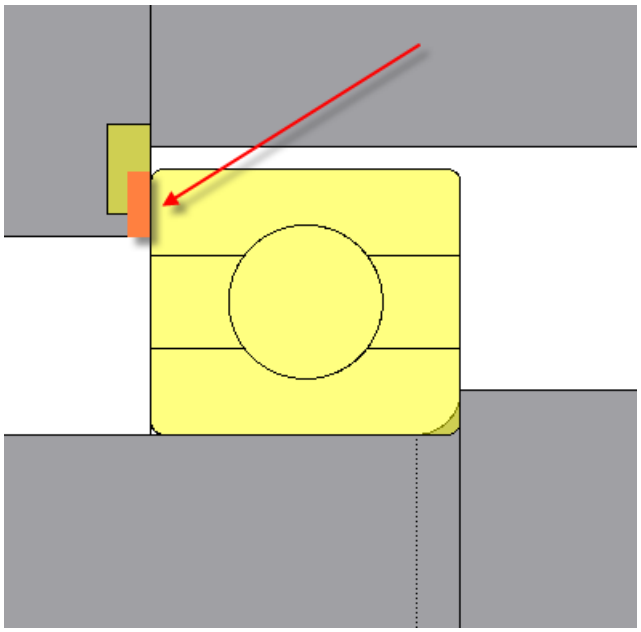
Bearing (in general) selection list	**ux**	**uy**	**uz**	**rx**	**ry**	**rz**
Own Input	Own definition	Own definition	Own definition	Own definition	Own definition	Own definition
Non-locating bearing	fixed	non-locating	fixed	non-locating	non-locating	non-locating
Fixed bearing adjusted on both sides <->	fixed	fixed	fixed	non-locating	non-locating	non-locating
Fixed bearing adjusted on right side ->	fixed	right	fixed	non-locating	non-locating	non-locating
Fixed bearing adjusted on left side <-	fixed	left	fixed	non-locating	non-locating	non-locating
Thrust bearing attached on both sides <->	non-locating	fixed	non-locating	non-locating	non-locating	non-locating
Thrust bearing attached on right side ->	non-locating	right	non-locating	non-locating	non-locating	non-locating
Thrust bearing attached on left side <-	non-locating	left	non-locating	non-locating	non-locating	non-locating
Fixed	fixed	fixed	fixed	fixed	fixed	fixed

ux, uy, uz: Displacement in X-, Y-, Z-direction.

rx, ry, rz: Rotation about X-, Y-, Z-direction.

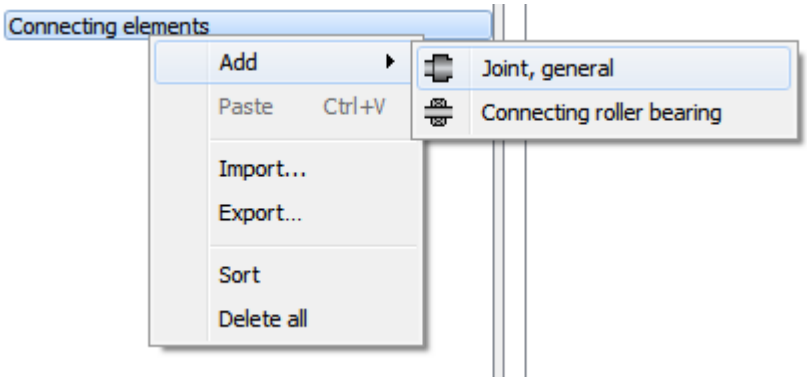
24.2.5.4 Pure thrust bearing

If a bearing is defined as a pure thrust bearing (attached left/right/both sides) no radial forces can be transferred to it. Any radial forces created by the pressure of the outer ring on the housing and the corresponding coefficients of friction will be ignored.



24.2.6 Connection elements

A number of coaxial shafts can be connected by two different connection elements: a general connection or a connecting roller bearing. The connection between these shafts, one defined as "shaft outside" and the other as "shaft inside", defines the constraints on the shafts over all possible degrees of freedom at that connection point, i.e. three relative displacements along axis x, y (the axial direction) and z, and three relative rotations around axis x, y and z.



NOTE

The number of coaxial shafts is limited to 30.

24.2.6.1 Connection, general

In general, you can define three different configurations of a connection element: a joint, a stiff connection or Own input.

Joint

All displacements are prohibited, but all rotations are permitted.

Stiff connection

All degrees of freedom (3 displacements and 3 rotations) are prohibited.

Own Input

The user can define their own constraints on the translational and rotational degrees of freedom (DOF). The options are:

- free: no restrictions on the corresponding degrees of freedom.
- fixed: the associated DOF is constrained in both directions.
- fixed with stiffness: the associated DOF is constrained in both directions with additional stiffness.
- one sided: the associated DOF is constrained on one axis direction (positive or negative, depending on the choice), but free on the opposite direction. The amount of allowable movement on the non-constrained direction can be input.
- one sided with stiffness: similar to above, but an additional stiffness can be input
- double-sided: the associated DOF is constrained on both directions. However, an allowable clearance can be input for both directions
- double-sided with stiffness: as above, but with the option of specifying stiffness.

24.2.6.2 Connecting roller bearing

A roller bearing can be used to connect two shafts. The only additional information needed, compared to above, is to define the inside and outside shaft for that roller bearing.

The bearing inner ring is assumed to be fixed on the inside shaft, and the bearing outer ring is assumed to be fixed on the outside shaft. The type of the bearing

(fixed adjusted on left/right side, etc) defines how the axial forces are transmitted between the shafts through the bearing. Roller bearings never constrain y axis rotation, so different rotation speeds between the connected shafts are permitted.

Due to the fact that moments as well as forces need to be transmitted through a connecting element, we strongly suggest that you perform an inner geometry bearing calculation when using roller bearings as connecting elements.

24.2.7 Cross sections

For information about the significance of cross sections in strength calculation, please refer to the corresponding section (see page III-710) in theCalculating Shafts section (see page III-688).

24.2.7.1 Free cross section

Free cross sections allows you to input the effects of notch, no matter what the actual definition of the shaft geometry.

24.2.7.2 Limited cross section

You should define the restricted cross section as the preferred cross section type in shaft calculations. The effect of notch is determined automatically in accordance with the geometry data at this position in this cross section. If you make changes to the shaft geometry, you do not need to modify the cross section manually. The changes are transferred automatically. However, if you are working with restricted cross sections, you must input shaft geometry in detail.

24.2.7.3 Documentation point

Set a documentation point to document the equivalent stress, displacement, rotation, force and torque at a particular position on the shaft in the report.

24.3 Basic data

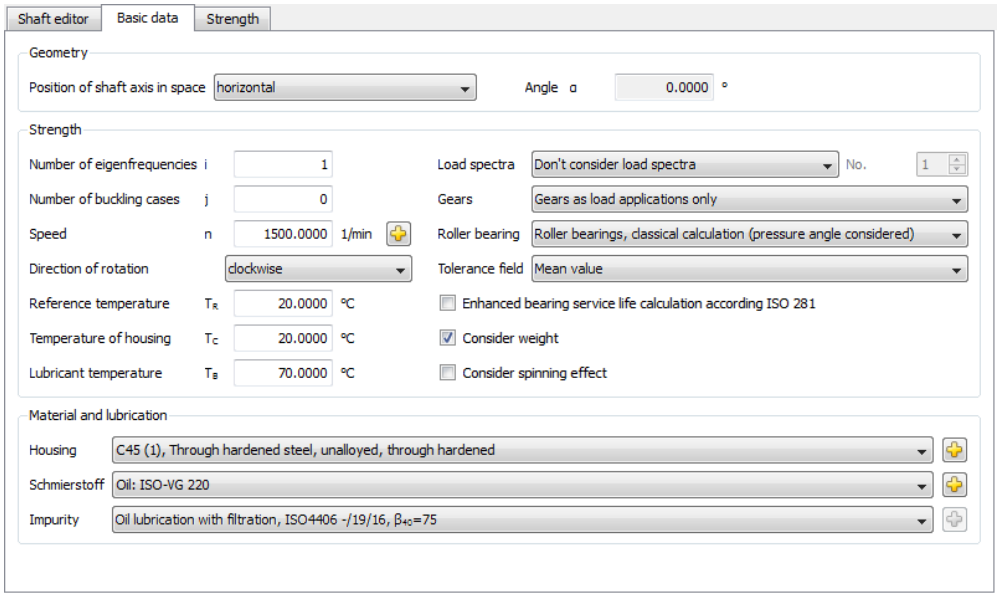


Figure 23.6: `Basic data` input window in `Shaft Calculation module`

In the `Basic data` input window you can control the basic preliminary settings for shaft calculation. You can enter values for these parameters:

24.3.1 Position of shaft axis in space

You define the position of the shaft axis in space as shown in Figure 23.7.

The consequence of the shaft arrangement in space (horizontally, vertically or in a defined angle to the horizontal) is:

The mass of the shaft (in a horizontal position) is considered a gravitational force in the plane ZY when the deflection is being calculated. However, if the shaft is positioned vertically the resulting axial force is, for example, included in roller bearing calculations. If a shaft is positioned at an angle, the corresponding force components are distributed on the ZY plane and as axial force.

Alternatively, you can enter the shaft weight direction vector using the 3-coordinate format.

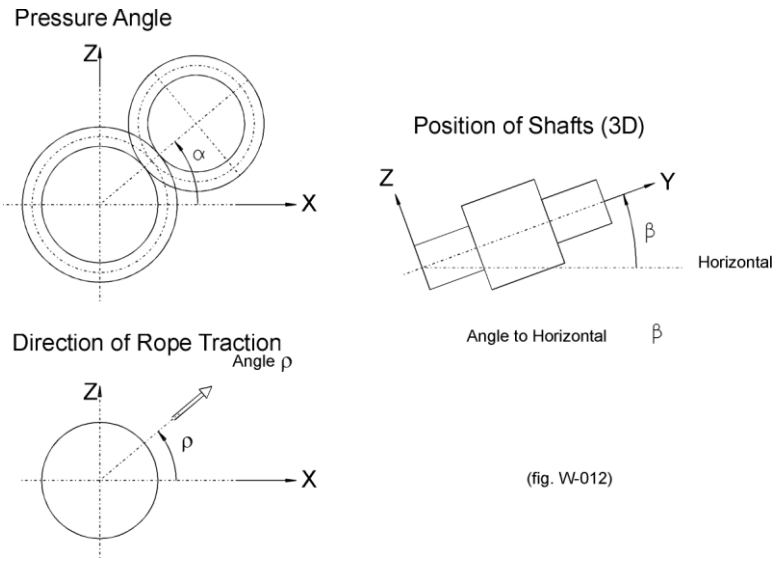


Figure 23.7: Defining the position of the shaft and the position of contact.

24.3.2 Number of eigenfrequencies

In this input field, specify the number of eigenfrequencies (see page III-694) that KISSsoft is to determine.

24.3.3 Number of buckling cases

In this input field, specify the number of buckling cases (see page III-696) that KISSsoft is to determine.

24.3.4 Speed

Enter the speed in revolutions per minute (rpm). Click the button to open the `Define speed` window. You will see the default values for speed and sense of rotation for all the shafts. If you click the checkbox next to the Speed input field, you can overwrite the speed for a particular shaft.

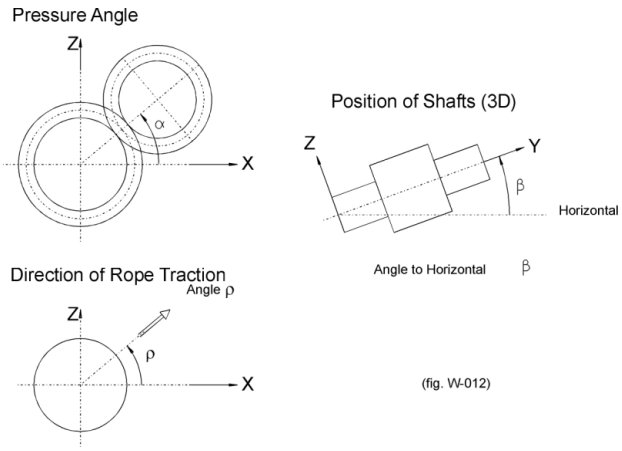
NOTE

If you change the speed, the effective torque and power change accordingly.

24.3.5 Sense of rotation

The shaft axis runs along the positive y–direction (left to right in the graphical Shaft editor). In the Shaft editor, the Z–axis points upwards, the X–axis points towards the user. A right-hand rotation of the shaft around the positive y-axis direction is specified as "clockwise".

The next figure shows the direction of these co-ordinates and the positive direction of forces and moments. Please note that weight has an effect in the negative Z–direction if the shaft is positioned horizontally (see section "Position of shaft axis in space" on page III-673).



In most force elements, the directions of the moments is usually defined by the terms "driving" and "driven". The entry "driving" means either that the shaft drives (an external application) or that the torque runs counter to the sense of rotation (i.e. the shaft loses power). The entry "driven" means either that the shaft is driven from outside (e.g. by a motor) or that the torque runs in the same direction as the sense of rotation (i.e. the shaft is supplied with power).

24.3.6 Reference temperature

The `Reference temperature` is the temperature specified for the shaft dimensions. This is the temperature on which the drawing data or element testing is based.

24.3.7 Temperature of housing

When used together with the thermal expansion coefficient, the housing temperature defines a strain which changes the distance between the bearing points. In addition, the thermal expansion and Young's modulus of the gear case has an effect on the nominal operating clearance of roller bearings.

NOTE

If you want to investigate the influence of thermal expansion in greater detail, you must also take the axial stiffness of bearings into account. If the bearings are assumed to be rigid, the load peaks will be too high.

The bearing's outer ring and the housing have the same temperature. The bearing's inner ring and the shaft also have the same temperature.

24.3.8 Lubricant temperature

The value entered for the `lubricant temperature` is only used to calculate the extended working life of the bearing. The lubricant temperature changes the lubricant's viscosity.

24.3.9 Load spectra

If loads, as defined in the Shaft editor, are assigned a load spectrum, you can calculate the deformation either using the nominal load or with an arbitrary element taken from the load spectrum. To do this, select the `Consider load spectra` option from the `Load spectra` drop-down list. If you only want to take into account one element from the load spectrum, you should select `Consider only one element of the load spectrum`. Enter the appropriate element number in the input field to the right of the drop-down list.

24.3.9.1 Lastkollektiv mit negativen Elementen

Load spectra with negative load spectrum elements (T < 0 and/or n < 0) are handled as follows:

Factor for torque or power	Factor for speed	Shaft direction of rotation	Force element
+	+	-	-
+	-	C	D

-	+	-	D
-	-	C	-

- = unchanged

C = direction of rotation changes clockwise/counterclockwise

P = driving/driven changes

24.3.10 Gears

Select an option from this drop-down list to specify how gears are to be handled in the shaft calculation:

- Gears are only handled as load applications. The masses and stiffness of the gears are not taken into account.
- Consider gears as masses. The gear wheel is handled as a mass in the bending calculation. The mass results from the difference between the reference circle and the outer shaft diameter as well as the gear width (same specific weight as the shaft).
- Consider gears as mass and as stiffness. The gear is handled as part of the shaft contour (for example, pinion shaft).
- Consider gears mounted by interference fit, with stiffness in accordance with ISO 6336-1. The shaft is stiffened at the mid diameter d_m, with $d_m = (d_1+d_2)/2$, d_1 = shaft diameter, d_2 = reference circle of the gear.

NOTE

Gear wheels set on shafts with a firm interference fit always pose the problem of knowing how much they stiffen the shaft. Although KISSsoft cannot solve this problem, it can estimate how much influence the interference fit has: It is sufficient to perform the calculation for `Gear as mass` and for `Gear as mass and stiffness`, and to note the difference in the diagram of bending. If the difference is small, the interference fit has no influence. However, if the difference is significant, you must enter more precise information. To do this you must integrate a part of the gear in the shaft contour in the graphical shaft input.

24.3.11 Roller bearing

The `Roller bearing` drop-down list has four options:

- Roller bearings, classic calculation (contact angle not taken into consideration), calculation using the classic method (as described in manufacturers' catalogs). Roller bearings primarily place constraints on the degree of freedom of

movement found in displacement and/or rotation, which is why they are modeled in this way when you select this option. You can enter any value as the stiffnesses for translation and rotation, no matter what type or size of bearing is involved. Any interrelationships between axial and radial forces (i.e. as in taper roller bearings) are ignored.

- Roller bearings, classical calculation (contact angle considered).
  The information in point one also applies here, with the difference that the correlation between axial and radial forces is included in the calculation, as for taper roller bearings.
- Roller bearing stiffness calculated from inner geometry, calculation using the classic method (as described in manufacturers' catalogs).
- Roller bearing service life in accordance with ISO/TS 16281

You will find more detailed information in the description of Bearing calculation (see page III-724).

24.3.12 Tolerance field

The definition of the bearing clearance class does not yet provide a definitive statement about bearing clearance because only one range of values has been defined for the bearing clearance class. The `Minimum` and `Maximum` options define the upper and lower limits of the range, whereas the `Mean value` is the arithmetical average of the `Maximum` and `Minimum` for (radial) bearing clearance.

NOTE

The selection you make in the Tolerance field has no influence on the general behavior of the bearing.

24.3.13 Modified rating life according ISO 281

Click on this checkbox to include the lubricant state in the bearing life calculation. However, to achieve an accurate result you must first have set the parameters for the `Lubricant` and `Impurity` drop-down lists, and entered a value in the `Lubricant` temperature input field. After the calculation is complete, you see a value for the modified service life L_{mnh} in the `Results` window and/or in the report.

24.3.14 Consider weight

Click this checkbox to include the shaft's dead weight in the section dimension calculation. Depending on the orientation of the shaft arrangement (see section "Position of shaft axis in space" on page III-673) you will see additional axial and shear forces which may have an influence on the diagram of bending and/or axial displacement.

NOTE

In a global coordinates system, gravitational forces act on the shafts in the negative, Z-direction.

24.3.15 Consider gyroscopic effect

Click this checkbox to include the properties of rotating shafts that have weights attached to one end and which rotate either in the same (or opposite) direction around the longitudinal axis. Whereas, in situations that are not technically critical, the eigenfrequency sinks when the speed increases in a counter direction, the eigenfrequency increases when the speed is in the same direction. The number of eigenfrequencies that appear here is double the number that appear when the effect of spinning is not taken into account.

24.3.16 Housing material

The housing material value is only used to calculate the thermal expansion of the housing. The materials available for housings are identical to those used for shafts.

24.3.17 Lubrication

Your choice of lubricant only affects the bearing life calculation. Click the button for your `own input` for the lubricant parameter.

24.3.18 Impurity

As defined in ISO 281, the impurity coefficient e_C depends on the type of oil filter, the bearing size, and the viscosity of the lubricant. This value varies within the range 0 (high level of impurity) ≤ eC ≤ 1 (ideal). Select the `Own Input` option and then click the button to specify your own e_C values.

NOTE

Click the button to enter your own values. You can define new values for `Housing` and `Lubricant` that are based on existing data. However, these values are not stored permanently in the database.

24.4 Module specific settings

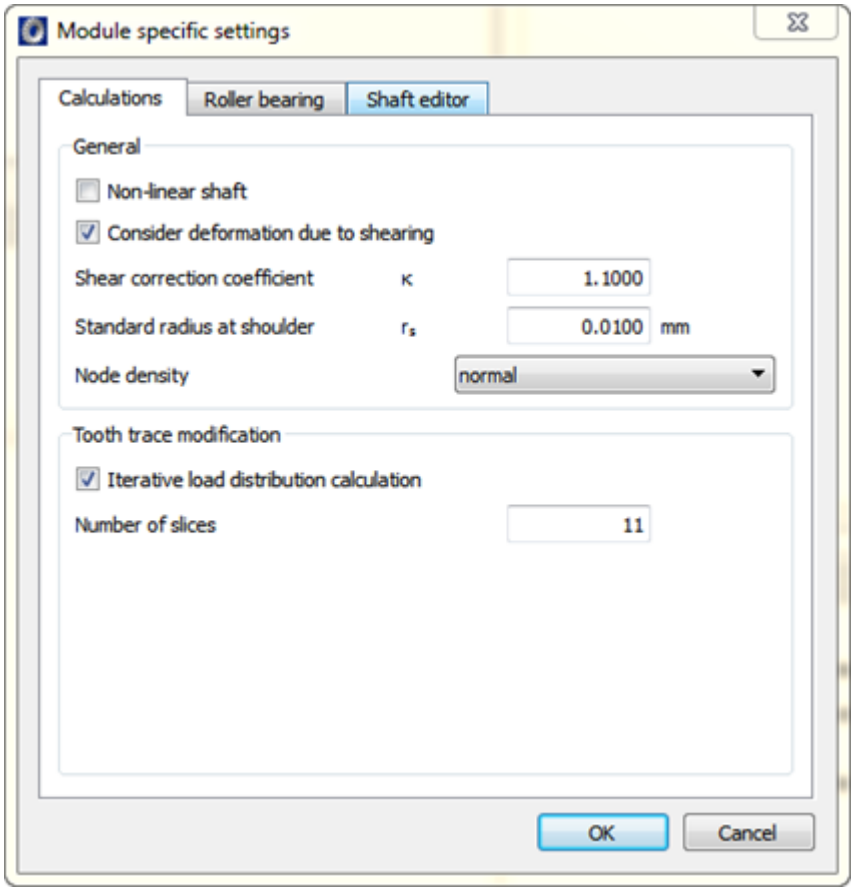


Figure 23.8: `Module specific settings` dialog window

24.4.1 Non-linear shaft

Click this option to perform a calculation using geometric non-linear beam elements. Due to the planet shaft deflection, the results here also show a displacement in the axial direction because the arc length remains constant. In most situations where shafts are used, this non-linear model is irrelevant.

EXAMPLE

A shaft, which is fixed to its mounting on both sides, is subjected to centrical force. The linear beam model does not allow for an elongation of the beam because it ignores axial displacement during shear and moment loads. If you click on the

`Non-linear shaft` field you can select a calculation method that takes into account the bending effect on the shaft and therefore the elongation of the beam. This results in axial forces.

24.4.2 Take into account deformation due to shearing and shear correction coefficient

If this checkbox has not been selected, the shaft is modeled to be infinitely stiff. In this case, shearing forces have no effect on the diagram of bending. However, if you do want to include deformation due to shearing, you can specify your own shear correction coefficient κ:

$$A' = A \cdot \frac{1}{\kappa} \tag{23.1}$$

where

A’ shear section

A Cross-sectional area

The shear correction coefficient $\kappa \geq 1$ includes the irregular distribution of stress across the cross section and applies to the entire shaft system. For circular-shaped cross sections, $\kappa = 1.1$ applies, and $\kappa = 1.2$ applies for rectangular-shaped cross sections.

NOTE

Note the definition of the shear correction coefficient used in KISSsoft, as shown in the previous equation. Some sources also use the reciprocal value for the formula symbol.

24.4.3 Standard radius at shoulder

To calculate the effect of notch on shoulders, you require a radius. This can be input as a sub-element. If no radius has been defined, you can use the standard radius defined for calculating the effect of notch.

Generally, we recommend you define radii for each shoulder.

24.4.4 Node density

The user can influence how many nodes are used to calculate a beam. If you are performing a linear calculation, this has no effect on the result, apart from line moments which are distributed across the existing nodes. The beam elements supply the exact solution in the linear model independently of the length.

Reasons for influencing the density of nodes are, on one hand to speed up calculations (for example, in series calculations in KISSsys) and, on the other hand, to ensure the accuracy of the display of the diagram of bending and the corresponding report.

The density of the nodes affects the accuracy of non-linear beam elements. For this reason, the maximum distance between two nodes for non-linear calculations when compared with a linear calculation is halved, no matter what value is predefined.

24.4.5 Iterative calculation of load distribution

If this selection box is enabled then the load distribution is calculated iteratively for the selected gear in the "Tooth trace modification" tab. The initial gear is replaced by a particular number of identical gears. The number of gears is set in the "Number of slices" field. The load on each replacement gear is set according to the current load distribution and the load on each gear is adjusted iteratively until the quadratic mean value (or "root mean square" - RMS) of the error in the line load difference between two sequential calculations is less than 1%.

You will find details of how KHβ is calculated in "Tooth trace modification" (see III-73).

Note: In the case of bevel gears the selection box must be selected so that the effect of the changeable operating pitch circle of the gear can be taken into account. Otherwise the bevel gear is handled as a cylindrical gear whose pitch circle dw equals the pitch circle in the middle section.

24.4.6 Input different load cycles for bending and torsion (for finite life calculations)

Every time a shaft rotates, the bending load cycle changes. For this reason, the number of bending load cycles is calculated using the service life and the speed. The number of torsional load cycles is often very much lower, because not every rotation causes a torsional load cycle. For example, a gear unit may be started in the morning and run throughout the day with a constant torque; resulting in exactly one torsional load cycle per day. In contrast, a shaft running at 1000 rpm for 8 hours would be subject to 8000 bending load cycles in the same space of time. As a consequence, in this example, the ratio between the number of bending load cycles is: torsion would be 8000: 1. You can enter this ratio here.

24.4.7 Axial clearance

This is where you define the axial clearance for rigid fixed bearings. The clearance applies to both directions. As a result, a bearing that is fixed on both sides may deviate either to the right or to the left by this value. However, this clearance value is not used if the bearing stiffness is taken into account by the inner bearing geometry. Axial clearance only applies to rigid roller bearings. You can either use this clearance value, or enter your own stiffness values for general bearings.

If an axially elastic shaft is mounted on several fixed bearings, for example, two bearings in a face-to-face arrangement, and the shaft is subject to a tension load, relatively high reaction forces are caused in the roller bearings which are not present in elastic bearings in real life. You can prevent this by entering a relatively small axial clearance for the bearings.

24.4.8 Failure probability

The failure probability value n is used to calculate the service life of roller bearings. The default value is 10% but you can overwrite this here. The valid input range is 0.05% < n < 10%.

24.4.9 Required service life

Required service life of roller bearings. However, this value does not actually affect the roller bearing calculation. However, if the calculated bearing service life is less than this value, the program displays a warning message. Click the sizing button next to the input field to size the service life of individual shafts. Sizing can only be performed in the fatigue strength range of the Woehler line.

24.4.10 Maximum service life coefficient

In this input field you define the upper limit for the service life coefficient a_{ISO}:

$$a_{ISO} = a_{ISO} \qquad falls \quad a_{ISO} \leq a_{ISO,\max}$$

$$a_{ISO} = a_{ISO,\max} \qquad falls \quad a_{ISO} \geq a_{ISO,\max}$$

The default value, as defined in ISO 281-2007, is $a_{ISO,max} = 50$.

24.4.11 Display critical bearing

The shaft editor shows the critical roller bearings using colors according to their lifetime. The color "orange" represents the critical bearings (below the minimum service life). The color "dark orange" represents the bearings below the required service life, and the color "blue" represents the bearings above the required service life.

24.4.12 Surface roughness of housing

The value of the surface roughness of the gear case is used to calculate the nominal operating clearance for roller bearings. The pressure is calculated for a housing with an infinitely large external diameter. If different roughnesses are needed for different bearings, or if you want to define the external diameter, you can specify an additional shaft that is then used for that purpose.

24.4.13 Calculation method for friction

From this list you can select whether the calculation method described in SKF catalog 1994 or in SKF catalog 2004 should be used to calculate friction. These methods are described in more detail in the Roller bearing chapter, under Torque of friction (see page III-739).

24.4.14 Type of oil lubrication

The type of oil lubrication used is important if you are using the method described in SKF catalog 2004 to calculate friction. The method differentiates between oil bath and oil injection lubrication. This is described in greater detail in the Torque of friction (see page III-739) section.

24.4.15 Bearing manufacturer

Only bearings made by selected bearing manufacturers are listed in the selection options.

24.4.16 Show coordinate system

This option toggles the coordination system in the Shaft editor on and off.

24.4.17 Show automatic dimensioning

This option toggles the mass line in the Shaft editor on and off.

24.4.18 Equivalent stress for sizings

This is the equivalent stress used to size a shaft for strength.

24.4.19 Maximum deflection for sizings

The maximum permitted deflection for sizing a shaft for deflection.

Chapter 24
Calculating Shafts

Once you have finished defining the shafts, either click the Σ button in the tool bar or press `F5` to calculate all the shaft-specific values. The results are shown either as a graphic or as a table of values. For example: click the `Graphics` menu in the menu bar, to display the shaft's diagram of bending in `Shaft > Displacement` (see Figure 24.1).

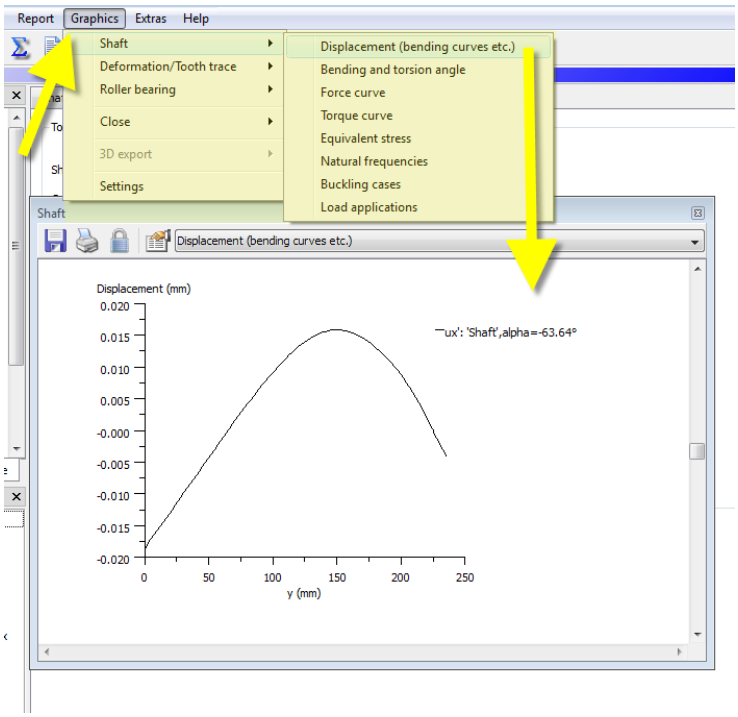
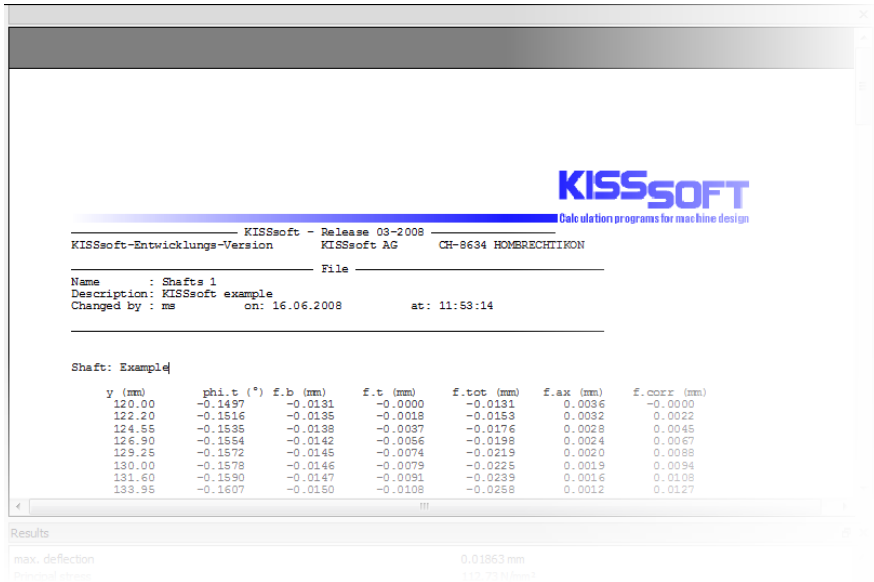


Figure 24.1: Opening the Graphic window via the `Graphic menu`

Alternatively, go to the `Report` menu and select the `Diagram of bending` option to display a list of the calculated values.



KISSsoft - Release 03-2008

KISSsoft-Entwicklungs-Version KISSsoft AG CH-8634 HOMBRECHTIKON

File

Name : Shafts 1
Description: KISSsoft example
Changed by : ms on: 16.06.2008 at: 11:53:14

Shaft: Example

y (mm)	phi.t (°)	f.b (mm)	f.t (mm)	f.tot (mm)	f.ax (mm)	f.corr (mm)
120.00	-0.1497	-0.0131	-0.0000	-0.0131	0.0036	-0.0000
122.20	-0.1516	-0.0135	-0.0018	-0.0153	0.0032	0.0022
124.55	-0.1535	-0.0138	-0.0037	-0.0176	0.0028	0.0045
126.90	-0.1554	-0.0142	-0.0056	-0.0198	0.0024	0.0067
129.25	-0.1572	-0.0145	-0.0074	-0.0219	0.0020	0.0088
130.00	-0.1578	-0.0146	-0.0079	-0.0225	0.0019	0.0094
131.60	-0.1590	-0.0147	-0.0091	-0.0239	0.0016	0.0108
133.95	-0.1607	-0.0150	-0.0108	-0.0258	0.0012	0.0127

Results

max. deflection 0.01863 mm

Figure 24.2: Calculation report for `Diagram of bending`

The sections that follow provide more detailed information about the interim results of the values you are interested in.

25.1 Deflection and Bearing Forces, Distribution and Force of Torque

The stress, displacement, and rotation, calculations are based on the one-dimensional Finite Element Method (FEM). The program determines the diagram of bending by automatically splitting the shaft into 50 to 100 sections and by using as many points for the diagram of bending. Boundary conditions and internal boundary conditions (bearing forces and moments) are found by solving a set of simultaneous equations with the same number of unknown variables.

Elastic bearings are considered by setting stiffness values (displacement and torsional stiffness).

The calculation enables you to:

- Calculate the diagram of bending, course of transverse force, and torque diagram, in the XY and the ZY plane (shaft axes always along the Y-axis) with or without taking into account the dead weight.
- Calculate the axial force taking into account the mass (depending on the length of the shaft).
- Graphical display of all critical dimensions on screen and as a printout: course of deflection, shearing force, bending moment in different planes, torsional moment and static comparative stress.
- Calculate the forces and torques in bearings (and shaft ends) for an unlimited number of bearings and any type of bearing.
- Bearing clearance is always considered. If the bearing calculation method according to inner geometry is selected then the bearing stiffness at the operating point and the static safety are also reported. Static safety is calculated as
  $S_0 = (p_0/p_{max})^n$
  where p_{max} is the maximum Hertzian pressure on the ring races. For ball bearings $p_0 = 4200$ N/mm^2 and n = 3. For roller bearings $p_0 = 4000$ N/mm^2 and n = 2.
- The system calculate and records the deformation and rotation of the inner ring relative to the outer ring.
- Note: the calculation assumes that the inner ring of the bearing is connected to the shaft. If a hollow shaft is connected to the inside of a roller bearing, the bearing displacement and rotation are documented with the reversed sign.

- Calculation of the inclination of the diagram of bending in bearings, e.g. when calculating cylindrical roller bearings. The progression of the angle of inclination can also be displayed on screen and printed out.
- The diagram of bending can be calculated with or without taking shearing deflection into account.

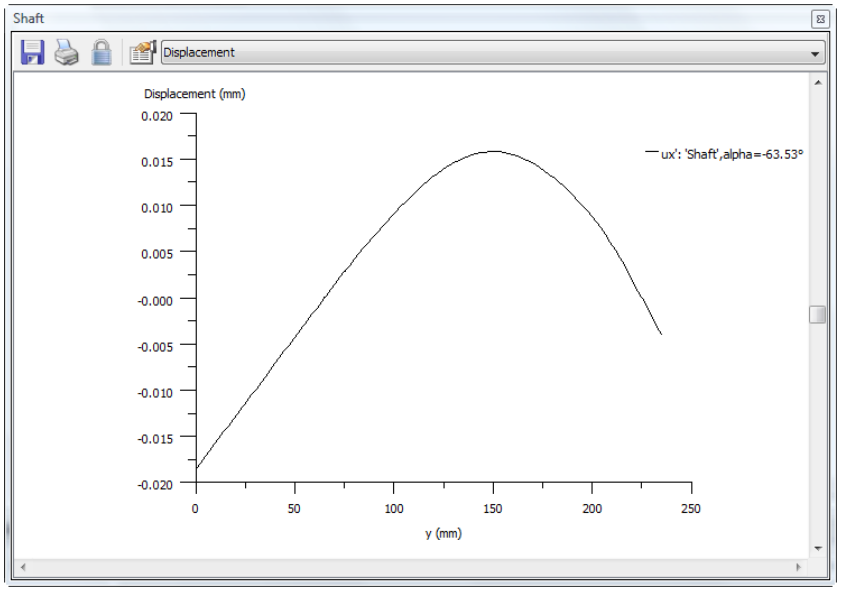


Figure 24.3: `Displacement` graphic with a diagram of the diagram of bending in the plane $\alpha = -63.53^{o}$

NOTE

Although the data about equivalent stress gives an initial indication of the static strength of a shaft, it cannot be used to calculate fatigue resistance. To do this you must perform the actual strength calculation. However, entering the equivalent stress is useful for beams because the load they are subjected to is usually only a static load. If the moment of resistance in torsion has not been defined for beams, torsional stress is not included in the comparative stress calculation. Despite this you can still perform the calculation.

25.1.1 Calculating force on bearings with a contact angle

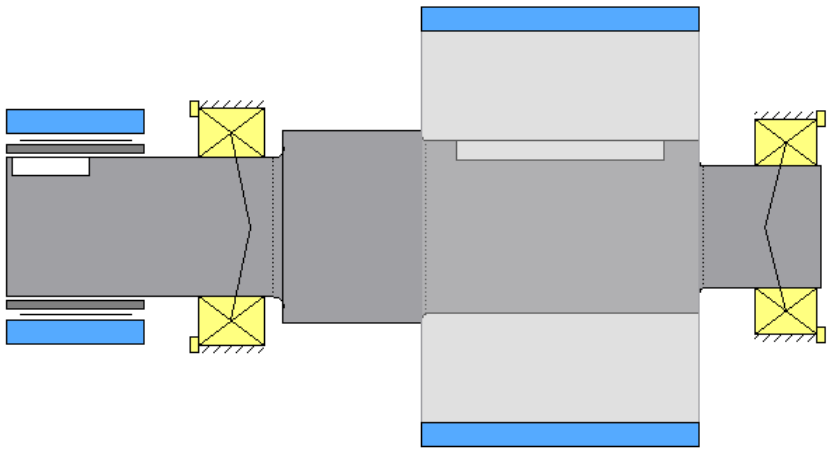


Figure 24.4: Representation of bearings with contact angles

Bearings with contact angle must be handled as a special case when you calculate shafts and bearings. The bearing center used to calculate the bearing reactions is determined at the point at which the compression force line of action intersects with the shaft centerline. In the roller bearing manufacturers' catalogs this is described as the axial forces resulting from the oblique position of the bearing housing. You can use this to define the data (radial and axial loads) required to calculate the roller bearing service life expectancy. It is harder -and also not clearly documented in the technical literature- to calculate the load progression in the shaft. Here, two modeling types are possible:

In bearings that have a contact angle, the effective bearing force line of action passes through the pressure center point. For this reason, you can calculate the bearing forces because, for calculation purposes, the bearing can be considered as being at the pressure center point. This corresponds to the procedures used to define the roller bearing load (Variant I).

However, you cannot introduce the bearing force on the shaft outside the bearing width. This is why KISSsoft places the bearing force in the center of the bearing. At the same time, the eccentric application of force creates an additional bending moment which equals the product of the distance of the bearing- and pressure center point, times the radial force (Variant II).

Both variants supply the same progression of bending moment between the pressure centers. There is, however, a difference in the area of the pressure/bearing centers. The shoulder on the right of the picture would be considered as not subject to a force in Variant I (it could, therefore be ignored), whereas Variant II displays both shearing force and a bending moment.

In real life, the load is not necessarily applied to the center of the bearing but to the entire area of the bearing. Therefore, the bending moment can be placed precisely on the shaft shoulder. However, this then causes a problem in the strength calculation if the load is applied directly on the proof point (i.e. when the proof point lies between the bearing center and the shaft shoulder).

The calculation of the diagram of bending produces a difference in that, in Variant I, the deflection is zero in the pressure center and, in Variant II, it is at the bearing position. Here, Variant II is certainly more precise, especially when large contact angles are involved where the pressure center lies outside the bearing width. Only Variant II allows the calculation to include cases in which the pressure center point lies outside the shaft.

As often happens, in such cases the reality lies somewhere between Variant I and II. More precise calculations can only be performed using time-consuming FEM calculations which take into account the properties of the bearing housing. Variant II is more precise and convenient for shaft calculations, (because it allows for pressure center points being outside the shaft), which is why this variant has been included in KISSsoft shaft calculation functions from release 04-2004 onwards. In special cases, when the modeling in Variant II is queried, you can modify the loads in the strength verification according to more precise observations when the proof point lies between the bearing center and the pressure center points.

One more point about the shaft strength calculation: any strength verification based on the nominal stress concept (DIN 743, . . has limited validity, in the load application zone (e.g. internal roller bearing ring on the shaft shoulder) when the local stress distribution does not correspond to the estimated nominal stress. In practice, the results calculated on these points must be interpreted with caution.

In KISSsoft, the additional internal axial force that is present in the case of bearings with a contact angle is calculated as Fr * 0.5/Y, as described in "Die Wälzlagerpraxis" and different bearing product catalogs. [FAG as here, NSK with a factor 0.6 instead of 0.5, SKF for taper roller bearings, as here, and for angular contact ball bearings with a factor 1.14 (Catalog 2004 as a function of Fa/C)]. If factor Y is not present in the bearing database, no additional axial force is taken into consideration. Therefore the calculation process is the same as the KISSsoft bearing calculation.

25.2 Eigenfrequency

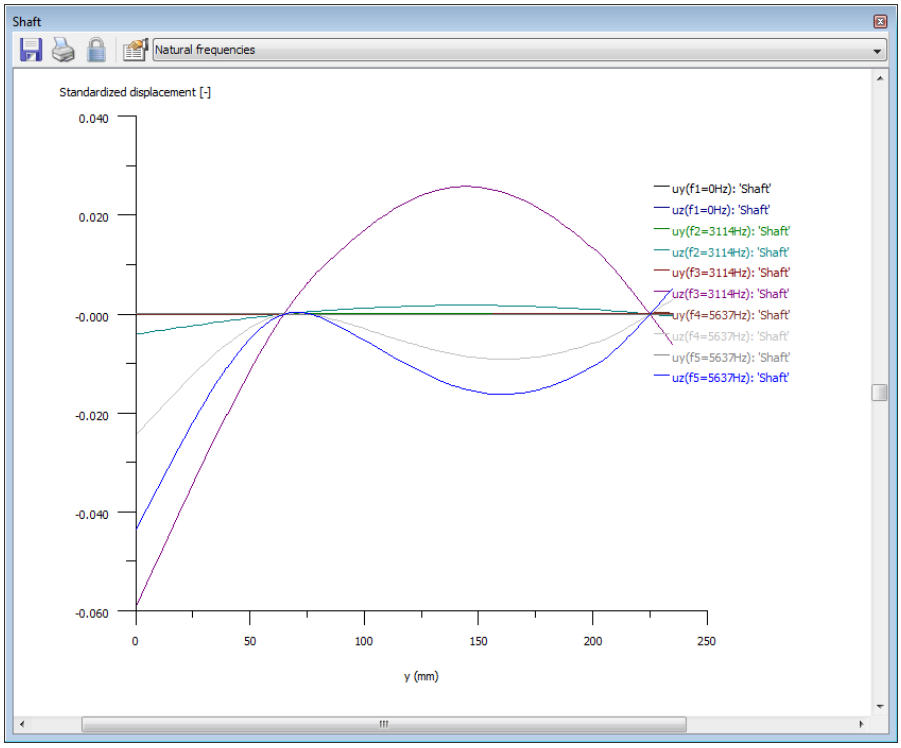


Figure 24.5: Graphics window: `Eigenfrequencies`

Click on `Graphics > Shaft > Eigenfrequency` to access the results of eigenfrequency calculation on the modeled shafts system with or without additional masses. The calculation is based on a one-dimensional Finite Element Method (FEM) which takes into account the type of bearings and their stiffness.

The calculation enables you to:

- Calculate any number of Eigenfrequencies[7]
- Display natural modes
- You can include the gyroscopic effect of large spinning masses if you click on the `Consider gyroscopic effect` checkbox in the `Basic data` input window. The critical speed (bending mode) is calculated for the forward and backward whirl. In a synchronous parallel run, an imbalance increases the bending oscillations because the angular speeds of the rotating shaft and the shaft's peripheral center point are the same. However, the synchronous counter run is, in most cases, not technically important.

[7] *Only limited by computing power.*

- For beam profiles, the critical bending mode (eigenfrequency) is calculated in both main planes.
- Gears can be included automatically and handled like masses. In this situation, KISSsoft takes into account the mass and the moments of inertia of the gear (see section "Gears" on page III-677) seated on the shaft.

25.2.1 Bending critical speed

The calculation of critical speed takes into account any masses located on the shaft. However, applied forces have no effect on the calculation. For this reason, additional masses must be handled as masses and not as loading forces.

25.2.2 Torsion critical speed

- Calculation of the critical rotating eigenfrequencies of shafts.
- Calculation of any number of rotating eigenfrequencies.
- Graphical display of natural oscillation.

25.3 Buckling

You use this function to calculate the buckling load of shafts and beams. All peripheral conditions, bearings and effective axial forces (point or line loads) are taken into account in the calculations. Only the axial forces you specify are used to calculate the buckling load. This function calculates the factor by which all these forces have to be multiplied to create a situation under which buckling occurs. This factor therefore corresponds to the safety against buckling.

25.4 Strength

To access the strength calculation, click the `Strength` tab in the `Shaft calculation` module user interface.

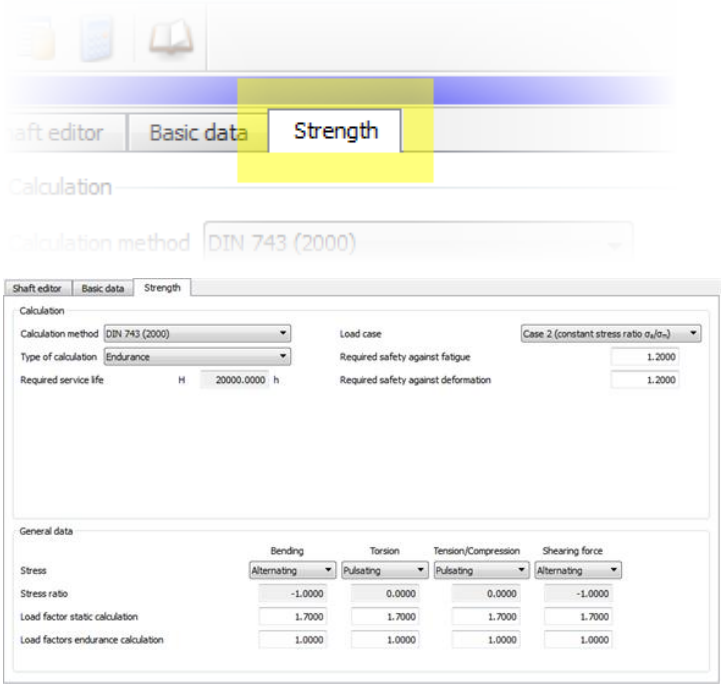


Figure 24.6: `Strength` input window in the `Shaft Analysis` module with the associated tab (above)

In KISSsoft, you can use the following methods to calculate the strength of shafts and axes:

- DIN 743:2012-12
  Load capacity of shafts and axes [9] including FVA proposed update concerning fatigue strength and tensile strength []
- FKM Guideline (2012)
  Analytical strength verification of steel, cast iron and aluminum materials in mechanical engineering, 6th Edition 2012
- Hänchen & Decker
- No strength calculation
  In this case, the strength verification is not performed. However, all the other results (diagram of bending, equilibrium of forces, bearing reactions etc.) will still be calculated.

A static and proof of fatigue strength can be applied in each case. The proof according to FKM and DIN can also be performed using a load spectrum.

Some of the shaft-specific data for the strength calculation can be defined in the Elements editor of a particular shaft.

25.4.1 Calculation method

In this drop-down list, you can select one of the calculation guidelines mentioned above. The sections that follow describe the guidelines in greater detail.

25.4.1.1 Hänchen & Decker

The calculation in accordance with R. Hänchen and H. K. Decker: [42] is an older, but tried and tested, method. If insufficient notch factor data is present, the equations produced by the TÜV in Munich, Germany, are used: they are derived from known test results.

Material values

As shown in Figures 52, 56, 60 in accordance with [42] for construction, heat treatable and case hardening steels. The empirical formula used is in accordance with Hänchen [42], page 37

$$\sigma_{bw} = 0.4 \cdot \sigma_b$$

You can enter the material data in the database (see page I-114).

Calculation of equivalent stress

In the case of bending and torsion, KISSsoft calculates the equivalent stress value σ_V in accordance with the hypothesis of the largest distortion energy (see [42], section 3.2.5.).

Calculation of safety against fatigue failure

- Maximum load in accordance with [42] equation (4a); Operating factor as defined in [42] Table 1 (page 24).
- Design bending fatigue limit in accordance with [42] Equation (42a).
- Safety against fatigue failure in accordance with [42] Equation (46).

- Required safety against fatigue failure in accordance with [42] Figure 156, depending on the frequency of the maximum load.
- Result of the calculation is the ratio of the required safety margin and the calculated safety margin as a percentage.

Important formulae

A)= Comparative stress (fatigue stress)

$$\sigma_v = \sqrt{\sigma_b^{\ 2} + 3 \cdot (\alpha_0 \cdot \tau_t)^2} \tag{24.1}$$

$$\sigma_b = \frac{M_b}{W_b} \tag{24.2}$$

$$\tau_t = \frac{M_t}{W_t} \tag{24.3}$$

A1) Comparative stress (strength against overload failure and deformation ($\tau_t = 0$)

$$\sigma_v = \sqrt{(\sigma_z + \sigma_b)^2 + 3 \cdot (\alpha_0 \cdot \tau_q)^2} \tag{24.4}$$

$$\sigma_z = \frac{F_z}{A} \tag{24.5}$$

$$\tau_q = \frac{F_q}{A} \tag{24.6}$$

B) Calculation of the safety against fatigue failure:

$$\sigma_{bWG} = \sigma_{bW} \cdot b_O \cdot \frac{b_d}{b_{kb}} \tag{24.7}$$

$$S_D = \frac{\sigma_{bWG}}{(\sigma_v \cdot f)} \tag{24.8}$$

α_0	a.0	Stress ratio factor	
A	A	Cross section area	*(cm3)*
b_d	b.d	Thickness number	
b_{kb}	b.kb	Notch factor (bending)	
b_o	b.o	Surface number	
f	f	Total load factor	
F_q	F.q	Shearing force	*(N)*
F_z	F.z	Tension/Compression force	*(N)*
M_b	M.b	Bending moment	*(Nm)*
M_t	M.t	Torque	*(Nm)*
σ_b	s.b	Bending stress	*(N/mm²)*
σ_{bW}	s.bW	Fatigue strength under reversed bending stresses	*(N/mm²)*
σ_{bWG}	s.bWG	Deformation strength under reversed bending stresses	*(N/mm²)*
σ_v	s.v	Equivalent stress	*(N/mm²)*
S_D	S.D	Safety against fatigue failure	
τ_q	t.q	Shear stress (shearing force)	*(N/mm²)*
τ_t	t.t	Torsional stress	*(N/mm²)*
W_b	W.b	Axial moment of resistance	*(cm³)*
W_t	W.t	Polar moment of resistance	*(cm³)*

Stress ratio factor

Table 24.2. contains values for the stress ratio factor.

Bending	alternating	alternating	static	static	static	static
torsion	pulsating	alternating	pulsating	alternating	static	static

Structural steel	0.7	0.88	1.45	1.6	1.0	1.0
Case hardening steel	0.77	0.96	1.14	1.6	1.0	1.0
Heat treatable steel	0.63	0.79	1.00	1.6	1.0	1.0

Table 24.2: Stress ratio factor α_0 in accordance with Hänchen page 28 [42] or Niemann, I, page 76 [64]

25.4.1.2 DIN 743 (2012)

The German DIN 743 standard [9] uses the most up to date information to calculate shafts and includes the following points:

- Consistent distinction between the different load classifications (tension/compression, bending, torsion) and between mean stress and stress amplitude.
- Surface factor: The influence on the strength is documented when using thermal processes (nitriding, case hardening) and mechanical processes (shot peening, rolling).
- Notch factors: Data for construction elements other than the usual notch factors is mentioned in all specialized books. This data, such as relief grooves, interference fit with relief groove or square notches (recesses for a Seeger ring) is widely used nowadays but has, until now only been poorly documented. All notch factors are documented for tension/compression, for bending and for torsion.
- Materials: An extensive list of materials, as well as instructions on how to derive estimated values for undocumented steels.
- Fatigue strength: the calculation of load strength in accordance with the "Miner extended" method is described in part 4 of the standard.

The critical limitations of the DIN 743 standard are:

- Shearing load (shear forces) is not included. This is not a disadvantage except for shafts with a very short distance between bearings.
- It only applies to steels and operating temperatures between -40°C and +150°C.

- As defined in the standard, the minimum safety margins for deformation and fatigue failure are defined as stated in 1.2. However, these safety margins only cover the lack of precision in the calculation method, and do not cover the problems encountered in load assumptions, or the consequences if the material fails. The required safety margins must therefore be checked or agreed by both the customer and contractor.

25.4.1.3 FKM-Richtlinie, Edition 2012

The FKM guideline (FKM: Forschungskuratorium Maschinenbau e.V., Frankfurt [Board of Research in Mechanical Engineering]) is based on the standards of the former German Democratic Republic ("East Germany" as was), and includes the latest knowledge on materials theory. It will probably form the basis of a new VDI guideline. The FKM guideline is extensive (running to approximately 175 pages plus 400 pages of commentaries), and includes not only fatigue strength calculations, but also endurance strength calculations and service life calculations, taking into account load spectra. It also provides calculation approaches for special problems such as operating temperatures above 100°C.

The calculation is performed in accordance with the 6th edition (2012) of the FKM Guideline, using the solutions proposed by Haibach.

Fatigue strength

The service strength coefficient $K_{BK,S}$ is determined in accordance with section 2.4 of the guideline. The number of cycles at knee point N_D is 10^6.

$K_{BK,S}$ is greater than 1.0 if the number of load cycles is less than N_D. Above N_D, $K_{BK,S}$ usually equals 1.0.

Normal calculations with a given load (without load spectrum) are referred to as an "individual load". This is calculated in accordance with Section 2.4 of the guideline. For load spectra, three different processes (see section "Type of calculation" on page III-702) are available.

25.4.2 Type of calculation

You can perform a safety analysis using one of these four different methods:

- **Static**. Proof for yield safety.
- **Endurance limit**. Proof for endurance limit (in the horizontal section of the S-N curve (Woehler line), no load spectra used)
- **Fatigue strength**. Calculates the safety against fatigue for a given number of cycles. Here, a constant load is used (no load spectra).

- **Miner consistent/elementary/extended**. These methods differ in the way they calculate the inclination of the S-N curve (Woehler line) above the knee point.

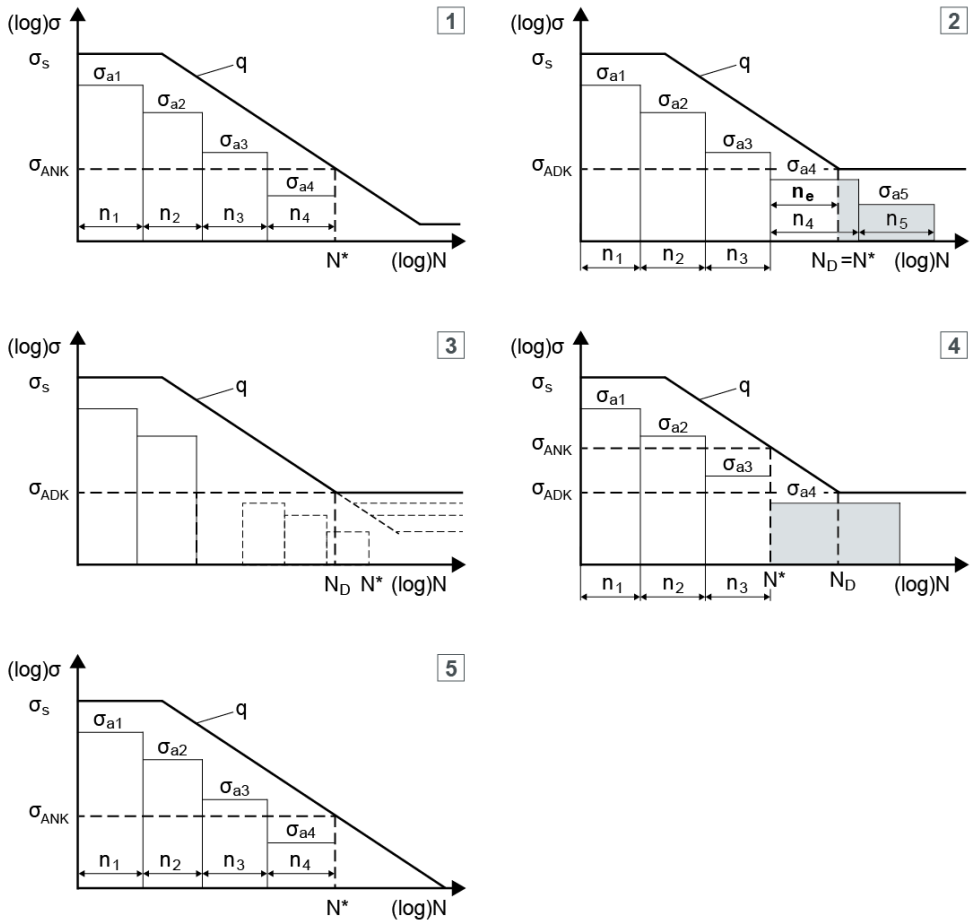


Figure 24.7: Miner hypotheses

Legend:

1) Miner elementary according to FKM guideline

2) Miner extended according to DIN 743-4:2012

3) Miner consequent according to FKM guideline

4) Miner original according to Haibach

5) Miner elementary according to Haibach

The gray fields are the fractions that are ignored.

NOTES

The calculation methods according to Miner are only available if you have selected the `Consider load spectra` option in the `Load spectra` drop-down list in the `Basic data` input window. As you can define load spectra (see section "Define load spectrum" on page II-287) in the KISSsoft database tool, you then only need to select them in the calculation.

25.4.3 Service life

The required service life in number of revolutions is calculated from the required service life in hours.

25.4.4 Strength parameters in accordance with Hänchen and Decker

25.4.4.1 Frequency of load

This value refers to the load value you entered previously (such as torque). If a load applies to the whole service life of the shaft, the frequency is 100%, otherwise it is correspondingly lower.

25.4.4.2 Notch factors

- Thickness number: in accordance with [42], Figure 120.
- Surface number: as stated in [42], Figure 119, Definition of the associated machining process in [42], Table 4.
- The following graphs have been preprogrammed:

Coarsely cut out	Graph with b_o = 0.50 at 150 kp/mm^2
Milled/finely turned	Graph with b_o = 0.70 at 150 kp/mm^2
Ground	Graph with b_o = 0.94 at 150 kp/mm^2
Polished	Graph with b_o = 0.97 at 150 kp/mm^2

- Shoulder notch effect coefficient during bending in accordance with [42], Figure 131.

- Wheel seat with key: proposed values after consulting with TÜV, Munich. Only very few details are given in [42], section 6.4.
- Interference fit: proposed values after consulting with TÜV, Munich. Details given in [42], section 6.4.
- Bearings are handled as weak interference fits. Only very few details given in [42], section 6.4.
- Shaft-hub connections (multi-wedge toothing): Diameter quotients and section modulus in accordance with [42], section 8.5. Conversion of the diameter quotient into the notch effect coefficient in accordance with [42], section 5.6, Formula (36) and (37b) or (37c) with the radius of the substituting notch in accordance with [42], Figure 112.
- Thread: Diameter quotient in accordance with [42], Figure 123. Conversion into notch effect coefficient as shown above.

25.4.4.3 Safety against deformation/fracture

KISSsoft calculates the required safety against fatigue failure, depending on the frequency of the maximum load, using Hänchen's definitions. If the frequency is 100%, the specified margin of safety is 2.0. At 0% it is 1.0. However, in between these two extremes, the margin of safety does not follow a linear progression.

The required safety against overload failure is 3.5 to 5.0, depending on the type of application or guideline involved. The required safety against deformation (yield point) is usually 2.0 to 3.5.

25.4.5 Strength parameters in accordance with FKM

25.4.5.1 Temperature duration

The FKM guideline takes into account thermal creep in various materials. Constant high temperatures will reduce the shaft's strength and therefore also reduce its safety.

Part temperatures in the range from -40°C ÷ +500°C are taken into consideration in accordance with the FKM guideline. For temperatures above 100°C (for fine grain steels above 60 degrees C), temperature factors (for tensile strength, yield point, and resistance to change) are used to take the reduction in strength into account.

NOTE

You can define the shaft temperature in the `Elements editor`. To do this, click on the shaft you want in the `Elements tree` and then enter the corresponding value in the `Temperature` field.

25.4.5.2 Protective layer thickness, Aluminum

If you have selected aluminum as the shaft's material, enter the value for the thickness of the aluminum oxide layer in this field.

25.4.5.3 Enter safeties

Click on this checkbox to set required safety values on the right-hand side of the Calculation group. Alternatively, click the button to open the `Define safeties` dialog window where you can specify required safeties as defined in FKM.

The safety factors for the static strength calculation, j_m (for overload failure) and j_p (for deformation), are determined in accordance with section 1.5 of the guideline, and the safety factor for fatigue resistance, j_D, is determined in accordance with Part 2.5 of the guideline. You will find detailed comments in the guideline.

Steel		$j_m = 2.0$	$j_p = 1.5$	$j_F = 1.5$	$j_F = 1.5$
GS, GJS	-not checked	$j_m = 2.8$	$j_p = 2.1$	$j_G{*}j_F = 2.6$	$j_G{*}j_F = 2.6$
	-non-destruction tested	$j_m = 2.5$	$j_p = 1.9$	$j_G{*}j_F = 2.4$	$j_G{*}j_F = 2.4$
GJL, GJM	-not checked	$j_m = 3.3$	$j_p = 2.6$	$j_G{*}j_F = 3.1$	$j_G{*}j_F = 3.1$
	-non-destruction tested	$j_m = 3.0$	$j_p = 2.4$	$j_G{*}j_F = 2.9$	$j_G{*}j_F = 2.9$

j_m, j_p: The values apply for
- severe damage as the result of failure
- high probability of load occurrence

If only minor damage results from the fracture, the safety factors can be reduced by about 15%. Provided the probability of the same load occurring again is low, the safety factors can be reduced by about 10%.

$j_G{*}j_F$: The values apply for
- severe damage as the result of failure

- irregular inspection

If only minor damage results from the fracture, the safety factors can be reduced by about 15%. Provided inspections are carried out regularly, safety factors can be reduced by about 10%.

25.4.5.4 Load case

The load case identifies four hypothetical scenarios for the development of the stress ratio σ_a/σ_m if load increases, starting at the operating point.

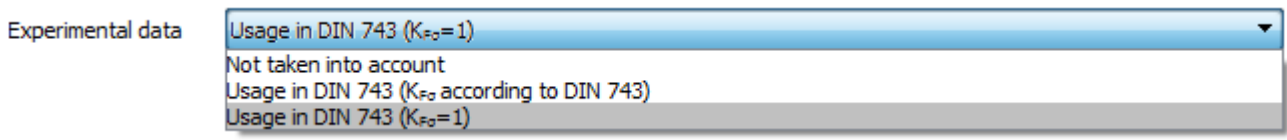
25.4.6 Strength parameters in accordance with DIN

25.4.6.1 Load case

The load case identifies two hypothetical scenarios for the development of the stress ratio σ_a/σ_m if load increases, starting at the operating point.

25.4.6.2 Calculation with experimental data

Use this option to define a Haigh diagram, which has been determined from experimental data. If you input a file name (e.g. WMAT-001.dat) in the **Experimental data** field for module-specific material data as defined in DIN 743, a selection list appears in the **Strength** tab.



- **Not taken into account**: the data is ignored.
- **Usage in DIN 743 (KFσ according to DIN 743)**: the data is imported from the file which was defined for the materials under Experimental data, and the KFσ coefficient is defined according to DIN 743.
- **Usage in DIN 743 (KFσ =1)**: the data is imported from the file which was defined for the materials under Experimental data, and the KFσ coefficient is always set to 1.

The method used to define data from the file requires separate instructions, on request. The measured Haigh diagram is not interpreted exactly as described in

DIN 743. The overall influence coefficient divides the Haigh diagram into x- and y-coordinates so that the results are much smaller.

The influence of mean stress as defined in DIN 743 increases as the notches become sharper, and should not decrease. This modification ensures that this influence always increases.

25.4.6.3 Safety against fatigue/deformation

In these input fields, you specify the required safeties for endurance/yield. A warning message appears if these values drop below the limit you specified for any one cross section.

25.4.7 Stress

This is where, in particular, you define how the loads calculated by KISSsoft (e.g. bending moment) are to be converted into amplitude or means stress. You can select usual loads (alternating, pulsating, static load) from the list. For exceptional situations, select `Own Input` from the `Stress` drop-down list and enter the required value in the `Stress ratio` input field (see next section). Rotating shafts normally have an alternating bending and a pulsating or static torsion.

25.4.8 Stress ratio

You must also enter a stress ratio because KISSsoft requires this value to split the load on the corresponding cross section into mean load and load amplitude.

Maximum stress per load cycle:		σ_o
Minimum stress per load cycle:		σ_u
Stress ratio		$R = \sigma_u/\sigma_o$
Mean stress:	σ_m	$= (\sigma_o + \sigma_u)/2$
		$= (\sigma_o + R \cdot \sigma_o)/2$
		$= \sigma_o \cdot (1 + R)/2$
Stress amplitude:	σ_a	$= (\sigma_o - \sigma_u)/2$
		$= (\sigma_o - R \cdot \sigma_o)/2$

$$= \sigma_o \cdot (1 - R)/2$$

For:

Pure alternating stress	$(\sigma_u = -\sigma_o)$	$R = -1$
Pulsating stress	$(\sigma_u = 0)$	$R = 0$
Static stress	$(\sigma_u = \sigma_o)$	$R = 1$

Normally valid for rotating shafts or axes:

Bending and shearing force:	$R = -1$
Torsion and tension/compression:	$R = 0$ (ev. R = 0...1)

NOTE

In contrast to the calculation in accordance with DIN or FKM, where there is a clear differentiation between the mean stress and amplitude stress, when a strength calculation in accordance with Hänchen (see page III-698) is performed, the loads that are entered are converted into an equivalent stress that is then compared with the fatigue limit for bending. For this reason, if you select this method, the stress ratio only affects the value of the stress ratio factor α_0.

25.4.9 Load factor for static analysis

The static calculation normally uses the greatest possible load. The maximum load factor covers the difference between the load value you specified and the peak value.

Maximum stress: $\sigma_{max} = \sigma_o \cdot f_{max}$

You can specify individual factors for every type of stress (bending, tension/compression, etc.).

The load factor is not used if the forces or power ratings are specified in free cross sections.

EXAMPLE

Electric motor with a permanent torque 100 Nm, starting torque 180 Nm. When you specify the shaft data, enter 100 Nm and set the maximum load factor to 1.8.

25.4.10 Load factor for endurance calculation

If necessary, the mean stresses and the stress amplitudes can be multiplied by a load factor. As the DIN743 standard does not include this factor, you should generally predefine it as 1.0. Using a factor > 1 is a good idea if you specify the nominal torque in a shaft calculation without taking into account the increases in torque due to the vibrations caused when the shaft rotates.

The load factor is not used if the forces or power ratings are specified in free cross sections.

The calculation in accordance with Hänchen includes the following information:

Total load factor as defined in (Hänchen [42], page 24):

$$f = f_{un} \cdot f_{betr} \cdot f_{leb} \tag{24.9}$$

f_{un} Uncertainty in maximum load (1.0 or 1.2 to 1.4)

f_{betr} Operational approach (shocks) (1.0 to 3.0)

f_{leb} Importance of part (1.0 or 1.2 to 1.5)

NOTE:

The Hänchen method uses only one load factor, which is the larger of the two values entered for bending and torsion.

25.4.11 Cross sections

Yield safeties and safeties for fatigue failure are evaluated at specific cross sections along a shaft that are defined by you. To define a cross section:

In the Elements tree you will see the `Cross section` entry at group level (→ see Figure on page III-645). Click the right-hand mouse button on this entry to open a context menu in which you can select either `Free cross section` or `Limited cross section`.

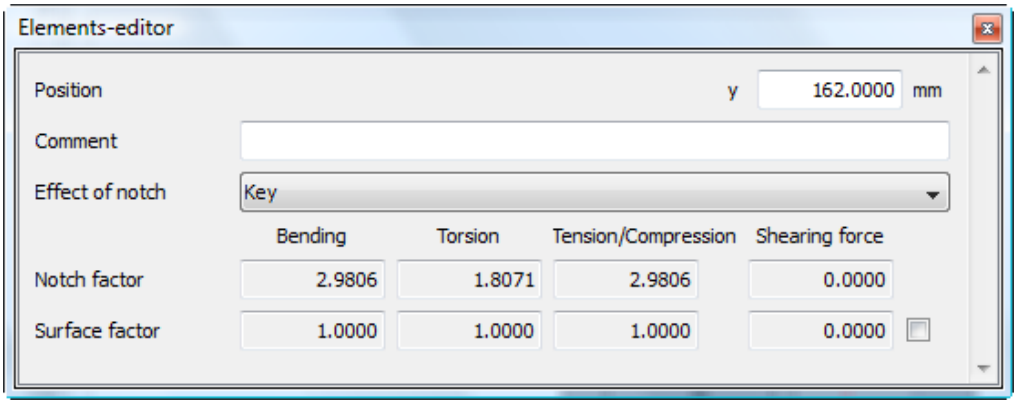


Figure 24.7: `Elements editor` for setting parameters for `Limited cross section`

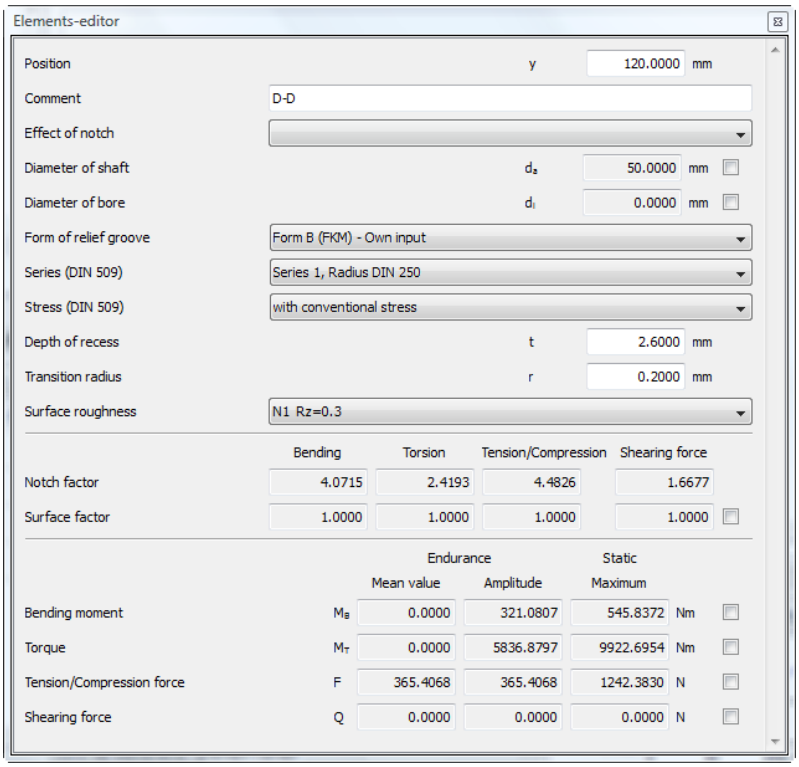


Figure 24.8: `Elements editor` for setting parameters for `Free cross section`

25.4.11.1 Surface roughness

If you enter a value for surface roughness as defined in ISO 1302, the corresponding surface roughness, R_Z, is displayed in the selection list. This value, R_Z, is then used in the calculation. In the calculation in accordance with DIN or FKM, the surface roughness has already been included in the notch factor in some cases. In such situations, the surface factor is always 1.0, no matter what value you input as the roughness.

25.4.12 Sizing

You can select the `Size` option in the context menu for the `Cross section` entry in the elements tree, to make it easier for you to define the cross sections that need to be recalculated.

In this sizing, KISSsoft automatically finds cross sections (shaft shoulders, interference fits in bearings, key-grooves and special notch effects) which have been defined in the `graphical shaft input` and in which a notch effect occurs. It displays the cross sections that have the lowest safety. You must check these cross sections carefully.

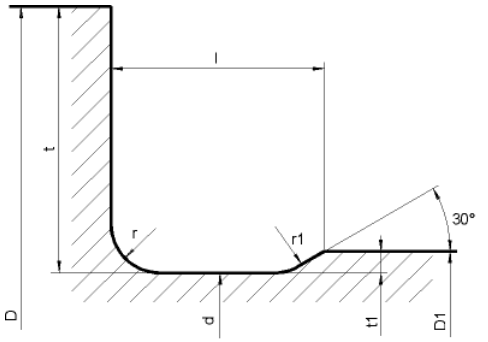
NOTE

Check for further notch effects, which KISSsoft cannot find, such as thread or cross hole.

25.4.13 Cross section types

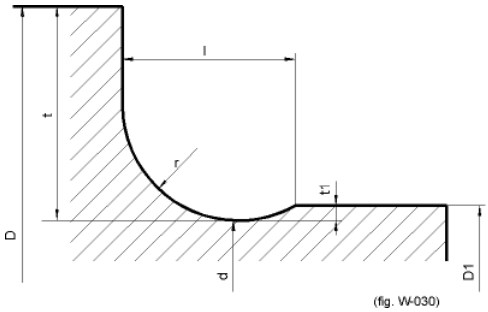
- **Shoulder**

- **Shoulder with relief groove**



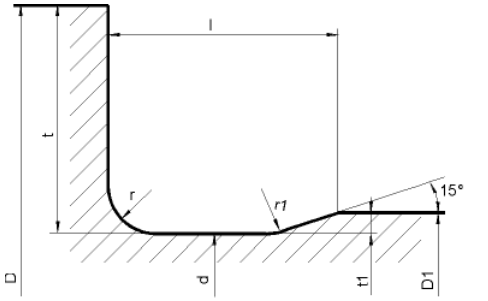
(fig. W-031)

FKM Form B



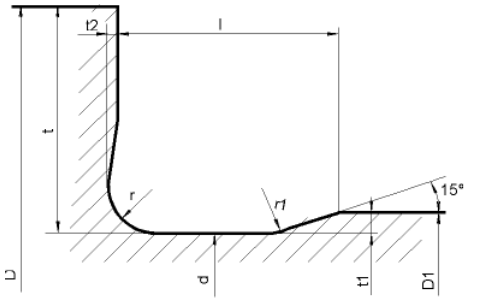
(fig. W-030)

FKM Form D



(fig. W-031-E)

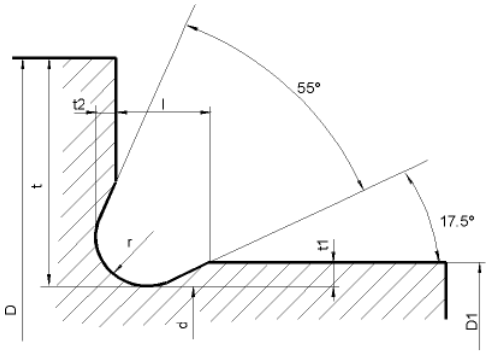
DIN 509 Form E



(fig. W-031-F)

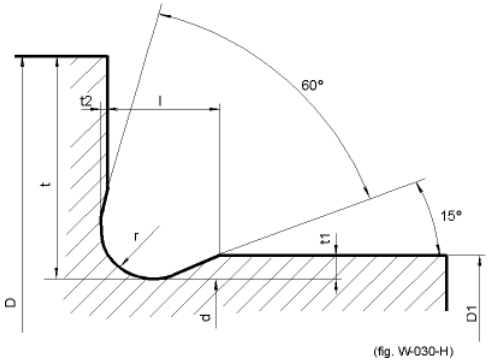
DIN 509 Form F

In accordance with FKM, these shapes are handled like shape B.



(fig. W-030-G)

DIN 509 Form G

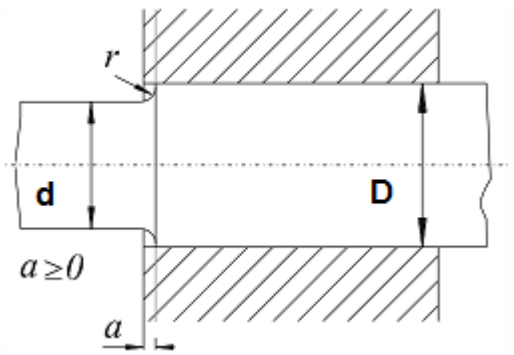


(fig. W-030-H)

DIN 509 Form H

In accordance with FKM, these shapes are handled like shape D.

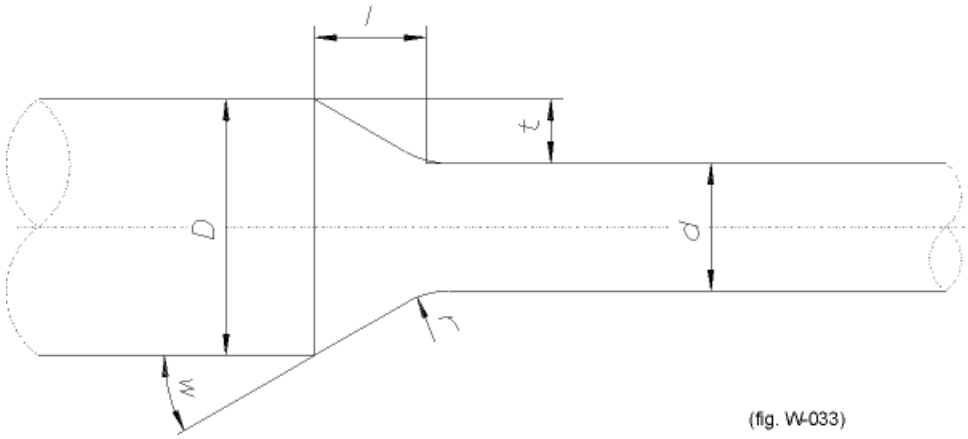
Shoulder with interference fit



In Hänchen+Decker:	not possible
In DIN 743:	The notch factor is calculated like a shoulder, but with the ratio d/(1.1*D). The maximum transmission for D/d ~ 1.1 and for r/(D/d) ~2. This condition is only applied if D/d >= 1.1, otherwise the notch effect of the shoulder is used.
In the FKM guideline:	The notch effect coefficient is determined for the fit H7/n6. The notch effect coefficient is also calculated for a shoulder and then used, in the least favorable case, in subsequent calculations.

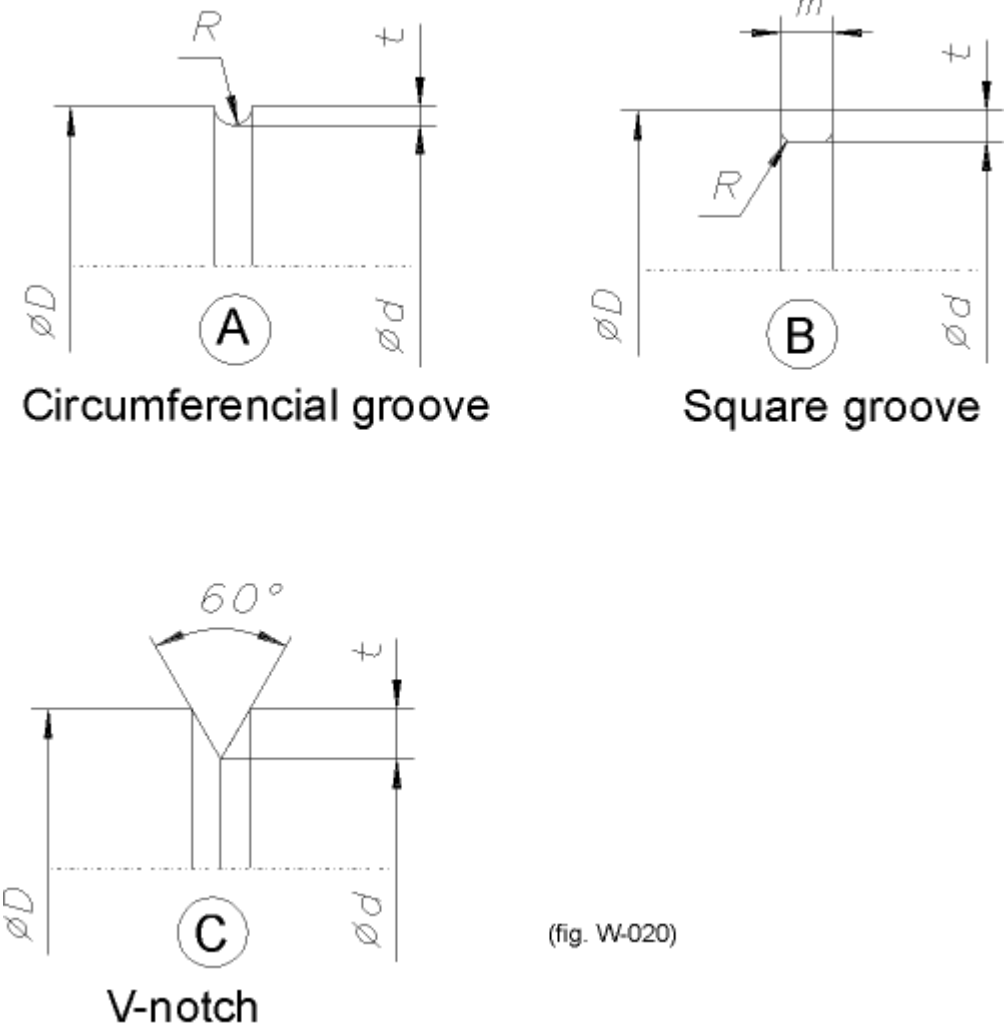
Notch factors are documented in the different methods. The notch factors calculated in FKM are usually significantly larger than in DIN.

Shoulder with conical transition



Shaft grooves

With the following variants:



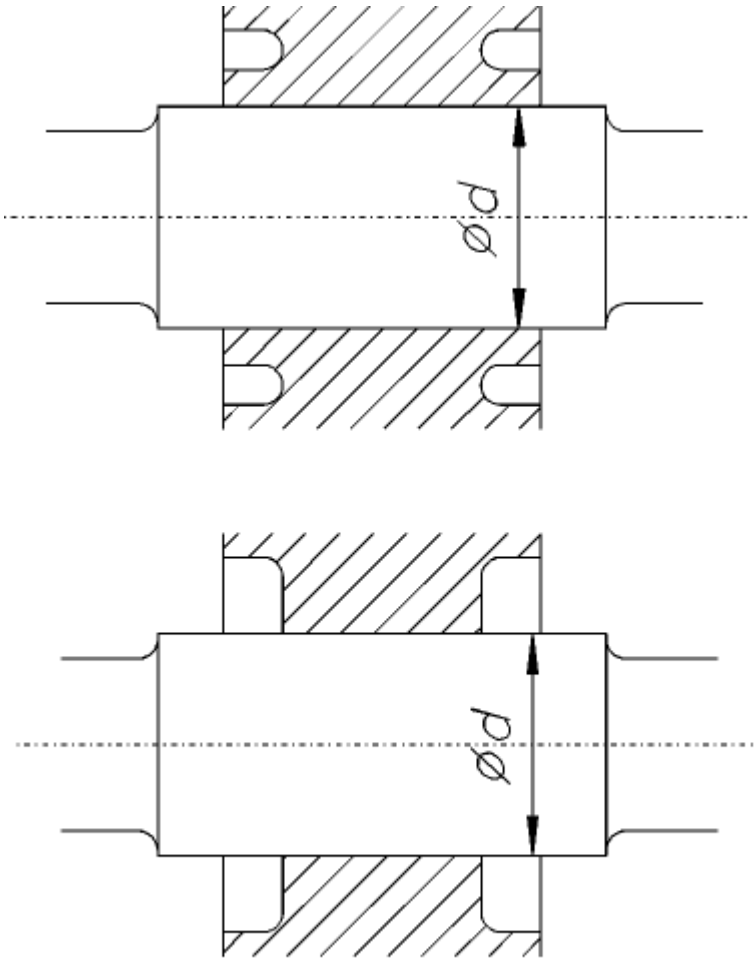
(fig. W-020)

Thread

Notch factors for threads are not described as a separate topic in the specialist literature. For this reason, notch factors for threads are handled like those for V-notches.

Interference fit

Interference fit (Firm interference fit, Slight interference fit, Interference fit with relief grooves).



(fig. W-021)

Top: Interference fit with relief grooves.
Below: Interference fit with end relief.

Key

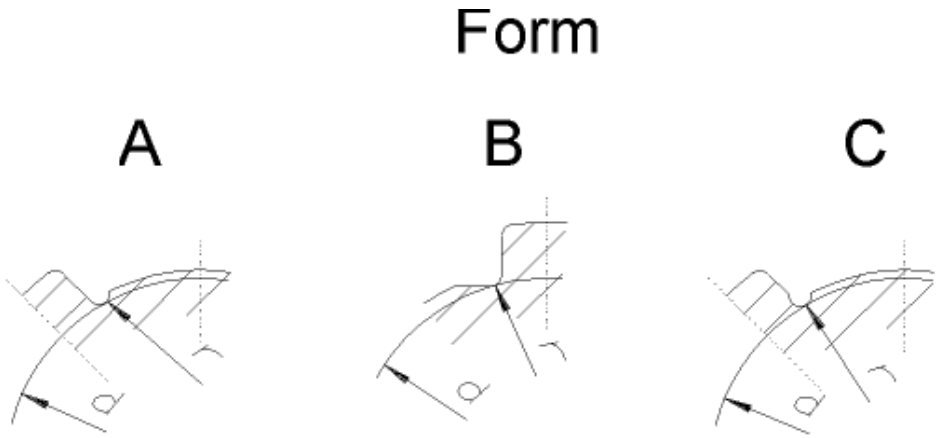
In every method, the moment of resistance for bending is determined from shaft diameter d. As described by Hänchen, the moment of resistance for torsion is computed from the incorporated circle d - t. In accordance with FKM and DIN it is calculated from the outer shaft diameter.

Notch factors are documented in the different methods. However, Hänchen provides very little information about this that can be used to extrapolate values for steel of higher strength (with the appropriate comment about the calculation). In contrast, these values are well documented in the DIN standard

and the FKM guideline (in the tables for Interference fit with key).

The program includes tables for cross sections with keys. The data is imported from a data file which includes the DIN 6885.1 (corresponds to ISO/R 773), DIN 6885.2, DIN 6885.3 standards. You can also specify other standards.

Groove toothing and spline shaft



(fig. W-032)

Shape of the spline shaft

To calculate groove toothings or spline shafts you must first enter tip and root circle data. All other values are used purely for documentation purposes.

To calculate the section modulus:

In Hänchen+Decker:	From the mean value ($d_a/2 + d_f/2$)
In the FKM guideline and DIN 743:	From the root circle

Notch factors are documented in the different methods.

Cross hole

Smooth shaft

If you select `Smooth shaft` the notch factor is set to 1. You should select this for the cross section with the maximum stress.

- **Define your own input notch factors** (see page III-710)

- **Intersecting notch effects** (see page III-787)

25.4.14 General entries

25.4.14.1 Thickness factors from the shaft diameter

You can derive material values that depend on the diameter either from the effective shaft diameter (d or D) or from the thickness of the raw material. The first choice of shaft diameter gives more reliable safety results but can only be used if the shaft is through hardened before it is turned.

However, if you select `Pre-turned to actual diameter (for shoulders K1 from d)` the material data for shoulders is derived from the smaller diameter (d). If you select `Pre-turned to actual diameter` it is derived from the larger diameter (D). Although deriving these values from D gives slightly lower strength values, the results are therefore on the slightly safer side. The standard does not comment on this.

25.5 Tooth trace modification

For various purposes it is important that you know how much a specific point in the shaft cross section moves in a particular direction due to elastic deformation (bending and torsion). An example of this is calculating the gap between the two halves of a coupling that are mounted on each end of the same shaft. In this situation, the displacement of a point on the shaft cross section is calculated in the axial direction.

The most important application of this calculation is to determine shaft deformation in the meshing area. The deformation for the pitch point is calculated along the facewidth. In this situation, the displacement of a point on the shaft cross section due to bending and torsion is calculated only in the direction of the normal to the flank. A displacement parallel to the flank only results in a very minimal change in sliding velocity and can therefore be ignored.

In the "Tooth trace modification" tab, you can directly select the gear currently present on the shaft. The data you have already input is used to define the necessary defaults for the calculation (Facewidth from and to, Coordinates meshing point, Direction of the normal to the tooth flank in the pitch point) which are displayed in the user interface. Therefore, assuming that the counter gear has infinite stiffness, the progress of the meshing point displacement due to deformation can be determined along the facewidth.

To display this deformation, also called "gaping", click `Graphics -> Tooth trace modification -> Deformation`.

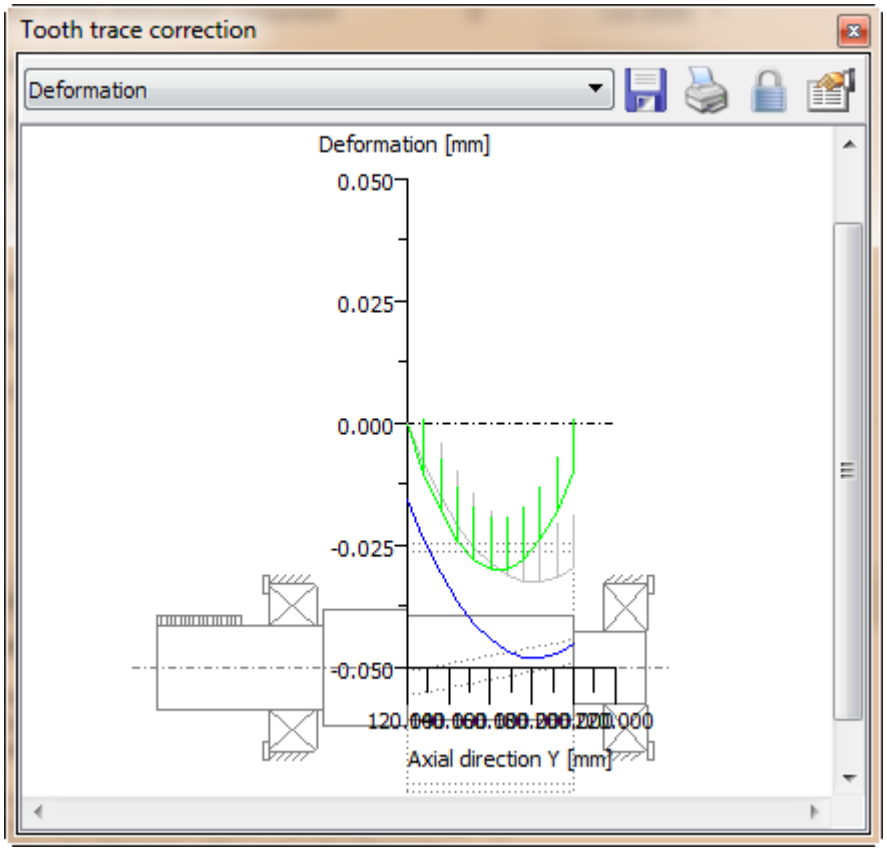


Figure 24.9:`Diagrams for tooth trace modification and> deformation`

This shows the deformation in the pitch point. It also shows a proposed value for an optimum tooth trace modification. This modification would achieve a homogenous load distribution along the facewidth.

You can input the tooth contact stiffness cγ in another input field. For steel gears, the tooth contact stiffness per mm facewidth is approximately 20 N/mm/°. The values of cγ are calculated precisely and documented in the cylindrical gear calculation. This stiffness can then be used to calculate the load distribution along the facewidth. Click `Graphics -> Tooth trace modification -> Load distribution` to see the result.

Calculate the load distribution coefficient KHβ for gear calculations

The results window also shows the load distribution coefficient KHβ calculated according to ISO 6336 with equation KHβ = wmax/wm from the average line load (wm) and the maximum line load (wmax). This calculation enables the face load factor to be estimated with significantly more accuracy, similar to Method B in ISO 6336. The procedure is basically similar to Appendix E of ISO 6336. However, you must be aware that the shaft of the counter gear used here is assumed to have infinite stiffness. This is permitted if the shaft of the counter gear has much greater stiffness. Manufacturing allowances are also only included if, for example, they have been defined by inputting an angular deviation of the shaft (bearing displacement) as part of the shaft data.

The addendum modification of the gear body determined from an FE (Finite Element) calculation can also be taken into account as a displacement matrix. To do this, select the "Take additional displacement matrix into account" option in the cylindrical gear force element. You will find the deviation.dat file, which gives an example of a displacement matrix, in the dat. directory.

NOTE:

If KHβ is to be determined whilst taking into account the deformation of the two shafts: The deformation components of two shafts can be combined in the cylindrical gear calculation in the "Contact analysis" tab.

Sizing the tooth trace modification

le

n

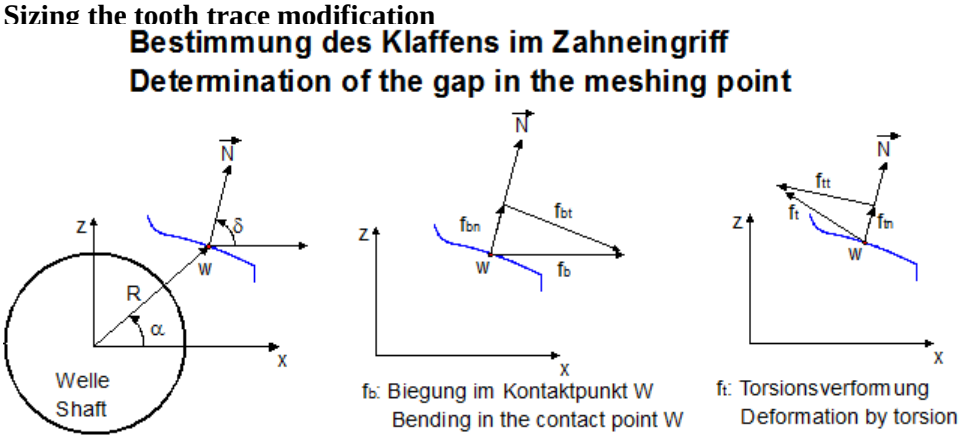
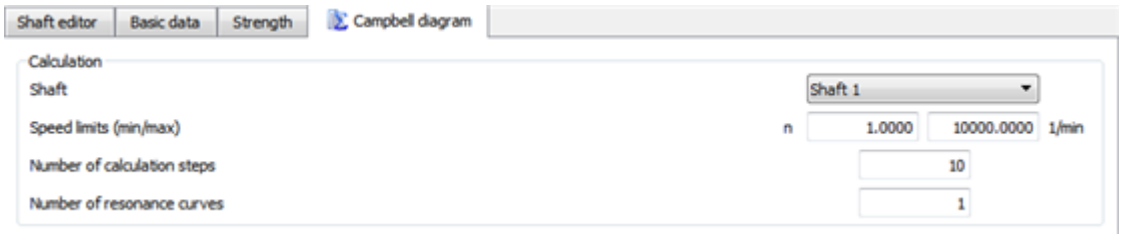


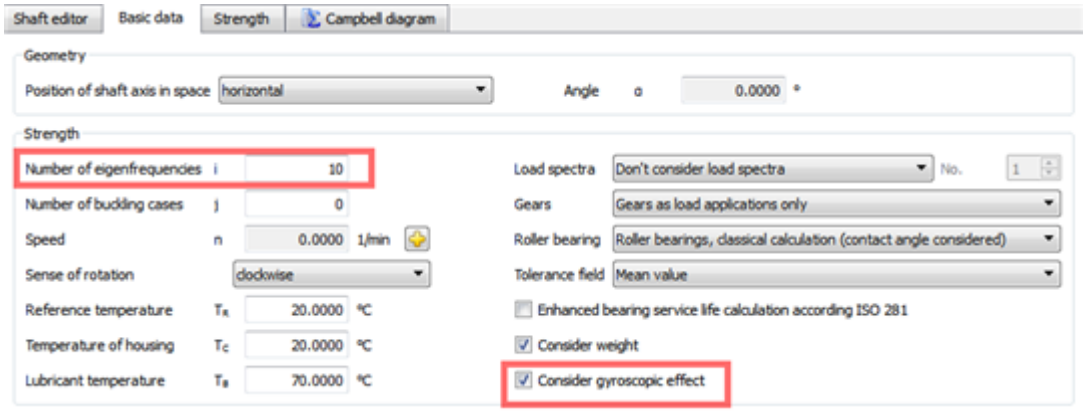
Figure 24.10: Determining the gap in the meshing point

25.6 Campbell diagram

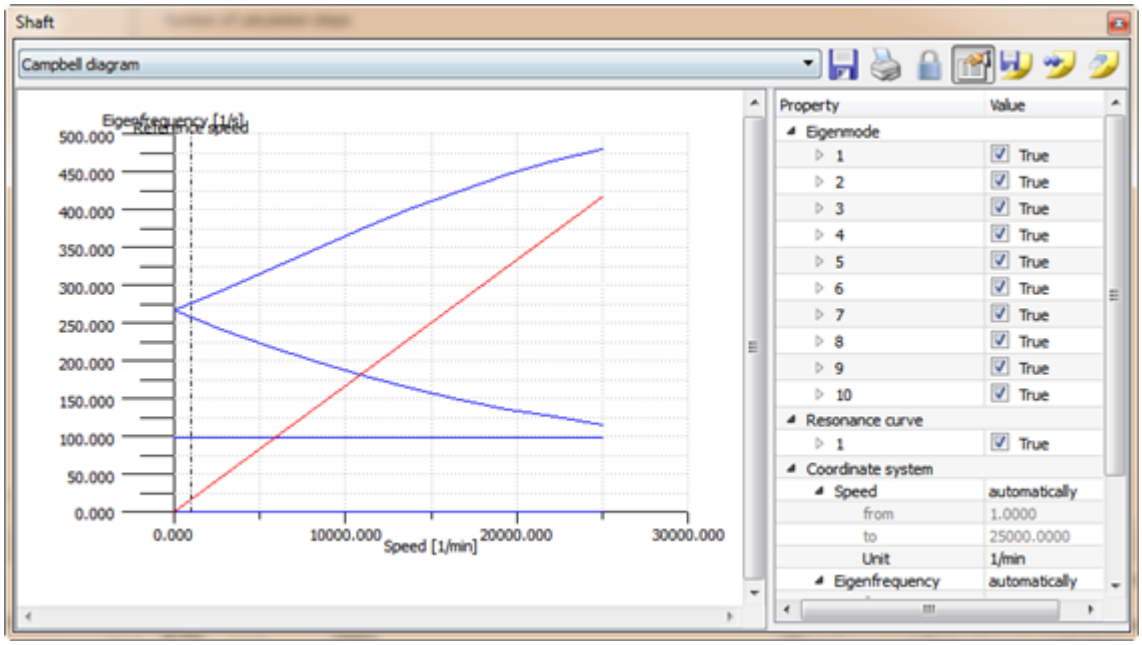
Select Calculation > Campbell diagram to enable the special calculation tab for the Campbell diagram. The user can set the shaft to be analyzed, range of shaft speeds, number of calculations of the speed range, and number of resonance curves (synchronous speed curves) to be displayed.



The Campbell diagram shows the eigenfrequencies in a wider range of shaft speeds, and then we can follow the forward and backward whirls associated with the eigenmodes. In order to calculate the Campbell diagram, the number of eigenfrequencies should be set in the Basic data tab. The gyroscopic effect causes large changes in the eigenfrequencies and can be taken into consideration by setting the "Consider spinning effect" checkbox in the Basic data tab.



In normal situations, the backward mode drops in frequency, while the forward mode increases. For forward whirl, as shaft speed increases, the gyroscopic effects essentially increase the spring stiffness and increase the eigenfrequencies. The effect is reversed for backward whirl, and increasing shaft spin speed reduces the effective stiffness, thus reduces the eigenfrequency. The eigenfrequencies are also affected by the stiffness of the bearings.



Chapter 25
Bearing calculation General

26.1 Classification of bearings

Bearings can be classified according to:

- the type of motion as for plain bearings, where the gliding motion takes place between the bearing and the supported part, and as for roller bearings where the rolling elements describe a rolling motion.
- the direction of the bearing forces for radial and thrust bearings.
- the function in fixed bearings which can take up shearing forces and axial forces in both directions and in non-locating bearings which allows displacement in a longitudinal direction.

26.1.1 Properties

The most important properties for the operational performance and use of plain and roller bearings can often be identified by examining their advantages and disadvantages.

There are hardly any rules to tell you how and when to use roller bearings. The choice of bearing depends partly on the properties which are determined from the advantages and disadvantages and partly from the operational requirements such as size and type of load, maximum speed, required service life and practical experience.

26.1.1.1 Roller bearing

Advantages: If used correctly, hardly any friction occurs when roller bearings are used, therefore the starting torque is required is only slightly higher than its working moment (major benefit when used for driving units!); they use little lubricant; they are easy to maintain; they do not require any running-in time; a large degree of standardization means roller bearings are easy to purchase and are widely exchangeable with each other.

Disadvantages: They are especially sensitive to impacts and shocks, when they are not in use or running at low speed; their service life and maximum speed are limited; their sensitivity to pollution can lead to added expense for sealing the bearing (wear, loss in efficiency!).

26.1.1.2 Plain bearing

Advantages: Due to their large, load-absorbing and lubrication area, plain bearings are insensitive to impacts and shocks, and they can run at unlimited speed; if fluid friction is used, they have an almost unlimited service life; split construction allows

easy mounting and dismounting; adjustable bearings give outstanding operational accuracy.

Disadvantages: Plain bearings require a larger starting torque (major disadvantage!); because of their initial dry friction they consume large quantities of lubricant and require constant supervision; they are generally slightly less efficient than roller bearings.

Chapter 26
Roller Bearings (Traditional Analysis)

Manufacturer catalogs (such as SKF) include fairly comprehensive methods for verifying the service life and the static load capacity of roller bearings. Specialized technical literature is also available to help you resolve more detailed problems [39].
KISSsoft includes bearing data from well-known bearing manufacturers. The user can extend these values.

In the KISSsoft initial window, select `Shaft and Bearings -> Roller bearings` from the Module tree.

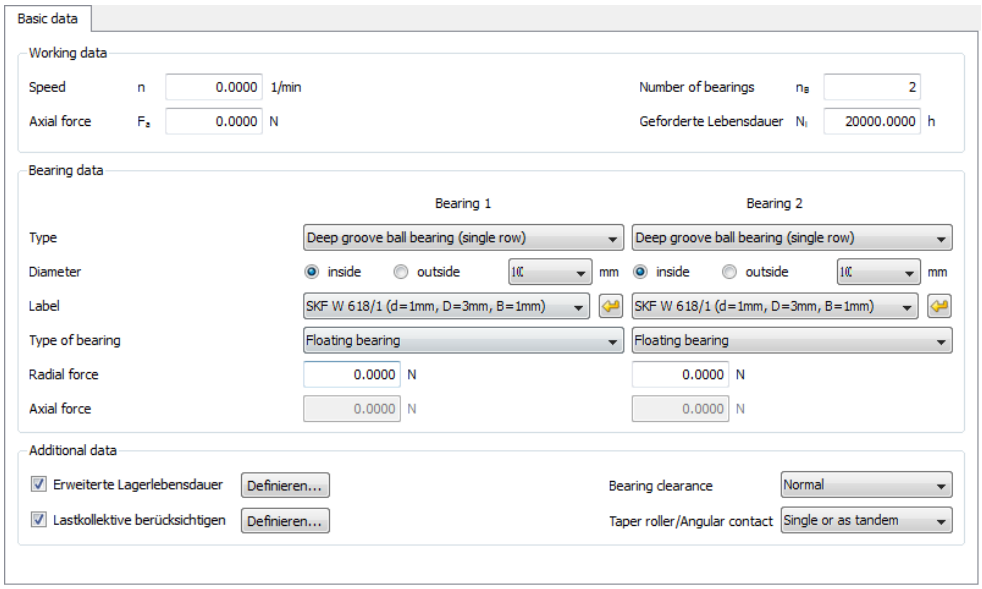


Figure 26.1: Basic data: Roller bearings

There is not much to explain here because the calculation provides numerous options, such as extended service life calculation or load spectra.

In the Basic data tab you will see a button for every bearing, next to its `Label` field. This function shows the service life of every bearing in the database (including the type and diameter). This makes it easy for you to select the best bearing for your purpose.

27.1 Selecting the type of roller bearing

27.1.1 Properties of the most important bearing types

Selecting the most suitable type of roller bearing is sometimes no easy matter. The table below presents an overview of the critical properties of the most important types of roller bearing:

- **Deep groove ball bearing (DIN 625):**
  The single row radial deep groove ball bearing is the most commonly used, because it is both extremely versatile and inexpensive. This bearing can withstand relatively high radial and axial forces in both directions.

- **Single row angular contact ball bearing and four-point contact bearing (DIN 628):**
  Each ring of a self-holding single row angular contact ball bearing has one lower shoulder and one higher shoulder. The grooves on the higher shoulder are positioned so that the contact angle is normally $\alpha = 40^o$. The higher number of rollers in this configuration means it can withstand not only radial forces but also larger axial forces in one direction (towards the higher shoulder) than deep groove ball bearings. Axial reaction forces due to the angle of the groove will be generated when the bearing is subjected to a radial load. You must take this into account when sizing the bearing. Because of its one-sided axial loading capacity, these types of bearings are usually installed in pairs where the second one is mounted in the opposite direction. The axial force that acts on the bearing in the case of a back to back or face-to-face arrangement is calculated and displayed in the screen. See also 26.3.17.

- **Double row angular contact ball bearing (DIN 628):**
  The double row angular contact ball bearing corresponds to a pair of mirror image compounded single row angular contact ball bearings (back-to-back arrangement) with $\alpha = 25^o$ or 35°, and can therefore withstand radial and high axial forces in both directions.

  Areas of use: To support the shortest possible bending-resistant shaft that is subject to strong radial and axial forces: integral worm shafts, shafts with angled spur gears and bevel gears.

- **Double row self-aligning ball bearing (DIN 630)**
  This is a double row bearing with a cylindrical or conical bore (bevel 1:12). It can compensate for shaft displacement and misalignment (up to approximately 4° angular deviation) thanks to its hollow sphere race in the outer ring. It can be subjected to radial loads and axial loads in both directions.

Areas of use: Bearings which are inevitably subject to inaccurate mounting and bending of the shaft, e.g. transmissions, conveyors, agricultural machinery, etc.

- **Cylindrical roller bearing (DIN 5412):**
  Cylindrical roller bearings can support larger radial loads than ball bearings of the same size (point contact area!) because the contact between the rollers and the races is made along a line. Demountable cylindrical roller bearings can only support small axial forces (if at all) and require accurately aligned bearings.
  Depending on the rim arrangement, you can identify (construction) types N and NU that have an unconfined outside and inside ring and which can be used as non-locating bearings, type NJ as a step bearing, and types NUP and NJ which can be used as a fixed bearing or locating bearing for axial shaft support in both directions.

  Areas of use: In gearboxes, electric motors, for axles of rail vehicles, for rollers in a rolling mill. In general for bearings that are subject to large radial loads.

- **Needle roller bearings (DIN 617):**
  are a special type of cylindrical roller bearing in which a cage separates the needle rollers to keep them at a specific distance from, and parallel to, each other. The bearing is supplied with or without an inner ring and is only suitable for radial forces. It can be characterized by its small overall diameter, its high degree of rigidity in the radial direction, and by its relative insensitivity to an uneven load.

  Areas of use: Predominantly used at low to medium speed and in oscillatory motion, e.g. as connecting rod bearings, rocker-shaft bearings, swivel arm bearings, jointed cross-shaft axle bearings (vehicles), spindle bearings, etc.

- **Taper roller bearing (DIN 720):**
  The ring races in taper roller bearings are cone-shaped shells which must converge into one point due to the action of kinematic forces. The bearings with $\alpha = 15^o(30^o)$ can support high loads both in radial and axial directions. The detachable outer ring makes them easy to assemble and dismantle. Taper roller bearings are installed in mirror image pairs. The bearing clearance can be set and adjusted as required. Due to the angle of the race, a radial force produces an axial reaction force.

  Areas of use: Hub bearings of vehicles, cable pulley bearings, spindle bearings in machine tools, shaft bearings in worm gears and bevel gears.

  Calculation: The axial force which you must specify when calculating a dynamic equivalent load is defined in several theories (for example page 296 of the FAG Wälzlager catalog WL 41520DE (1995)). The axial force acting on

the bearing is displayed in the screen. See also section 26.3.17. The bearing forces that include the contact angle can be calculated directly.

- **Barrel-shaped bearings (DIN 635), toroidal roller bearings (CARB), and double row self-aligning ball bearings (DIN 635):**
  Spherical races in the outer ring and barrel-shaped rollers (toroidal-shaped for CARB bearings), as in double row self-aligning ball bearings, enable barrel-shaped, toroidal roller (CARB) and double row self-aligning ball bearings with a cylindrical and conical bore (1:12) to compensate for misalignment and for the angular dislocation of the shaft (oscillating angle 0, 5° to 2°). Barrel roller bearings are suitable for high radial loads but can only withstand low axial forces. In contrast, double row self-aligning roller bearings ($\alpha = 10^o$) can be used for the highest radial and axial forces. Toroidal roller bearings (CARB) have an extensive range of uses in many load applications. Toroidal roller bearings combine the angular flexibility of double row self-aligning roller bearings with the axial displacement options of cylindrical roller bearings.

  Areas of use: For heavy wheels and cable pulleys, propelling shafts, rudder posts, crank shafts, and other heavily loaded bearings. Toroidal bearing: Paper making machinery, blowers and, generally, in planetary gear units.

27.1.2 Comparing types

Selecting the most suitable type of roller bearing is sometimes no easy matter. The table below presents an overview of the most important properties. The bearing you select for specific operating conditions has often already been determined by its properties and characteristics. You can use this information to select the bearing you require for frequently occurring working cases and for specialized requirements. However, results may overlap, and therefore the cost factor may be decisive.

Radial bearing:

Features	**a**	**b**	**c**	**d**	**e**	**f**	**g**	**h**	**i**	**j**	**k**	**l**	**m**	**n**
Radial load capability	⊗	⊗	⊗	∅	⊗	+	+	+	+	+	+	+	+	+
Axial load capability	⊕	⊗	⊗	⊗	∅	-	⊕	⊕	-	⊕	⊕	+	⊕	⊗
Inside position adjustment	-	-	-	-	-	+	∅	-	+	∅	-	-	-	-
Mounting position adjustment	⊕	⊕	⊕	-	⊕	-	-	⊕	-	⊕	⊕	⊕	⊕	⊕
Dismountable bearings	-	-	⊕	⊕	-	+	+	+	+	⊕	-	+	-	-
Alignment error adjustment	∅	-	-	-	+	∅	∅	∅	-	∅	-	∅	+	+

Increased precision	⊕	⊕	⊕	∅	-	⊗	⊕	⊕	+	-	-	⊗	-	-
High speed running	+	+	⊕	∅	⊗	+	⊗	⊗	+	-	-	⊕	⊕	⊕
Quiet running	+	⊗	∅	∅	∅	⊕	∅	∅	⊕	-	-	∅	∅	∅
Conical bore	-	-	-	-	+	⊗	-	-	+	-	-	-	+	+
Gasket on one/both sides	⊕	-	⊕	-	⊕	-	-	-	-	-	⊕	-	-	⊗
High stiffness	⊗	⊕	⊕	⊗	∅	⊕	⊕	⊕	+	+	+	+	⊕	⊕
Low friction	+	⊕	⊗	⊕	+	⊕	⊕	⊕	+	-	-	⊕	⊗	⊕
Fixed bearing	⊕	+	⊕	⊕	⊗	-	⊗	⊕	-	⊗	⊗	+	⊕	⊕
Non-locating bearing	⊗	⊗	⊗	-	⊗	+	⊗	∅	+	⊗	⊗	∅	⊗	⊗

+ very good ⊕ good ⊗ normal/possible ∅ with restrictions - not suitable/no longer relevant

a Deep groove ball bearing

b Angular contact ball bearing (single row)

c Angular contact ball bearing (double row)

d Four-point contact bearing

e Double row self-aligning ball bearings

f Cylindrical roller bearings NU, N

g Cylindrical roller bearings NJ

h Cylindrical roller bearings NUP, NJ+HJ

i Cylindrical roller bearings NN

j Cylindrical roller bearings NCF, NJ23VH

k Cylindrical roller bearings NNC, NNF

l Taper roller bearing

m Barrel roller bearing

n Double row self-aligning roller bearings

Thrust bearing:

features	**o**	**p**	**q**	**r**	**s**	**t**
Radial load capability	-	-	∅	-	-	∅
Axial load capability	⊗	⊗	⊗	⊗	⊗	+

Inside position adjustment	-	-	-	-	-	-
Mounting position adjustment	-	-	-	-	-	-
Dismountable bearings	+	+	-	+	+	+
Alignment error adjustment	⊕	⊕	∅	-	-	+
Increased precision	⊗	-	+	+	⊕	-
High speed running	⊕	∅	⊗	+	∅	∅
Quiet running	∅	-	∅	∅	-	-
Conical bore	-	-	-	-	-	-
Gasket on one/both sides	-	-	-	-	-	-
High stiffness	⊕	⊕	⊗	+	⊕	⊗
Low friction	⊗	∅	⊕	⊕	-	-
Fixed bearing	⊕	⊕	+	+	⊗	⊗
Non-locating bearing	-	-	-	-	-	-

+ very good ⊕ good ⊗ normal/possible ∅ with restrictions - not suitable/no longer relevant

o Axial deep groove ball bearing (one-sided)

p Axial deep groove ball bearing (two-sided)

q Axial angular contact ball bearing (one-sided)

r Axial angular contact ball bearing (two-sided)

s Cylindrical roller thrust bearing

t Spherical thrust roller bearings

27.2 Load capacity of roller bearings

The dynamic load capacity of the rotating bearing, and the static load capacity at standstill, vary at very slow speed or very small oscillations, depending on the working state, but not on the effect of the load.

27.2.1 Dynamic load capacity

The dynamic load capacity is a property of the entire bearing. In accordance with ISO 281, a number of various properties of a roller bearing are included, that the bearing has when it experiences specific mechanical loading under specific conditions at specific speeds. This data is then used to calculate the number of operating hours (this is usually based on a failure probability of 10%).

27.2.2 Permissible static stress

The static load capacity includes properties that a roller bearing must display in order to withstand certain mechanical loading situations at standstill, at very low speed (n < 20 rpm), or during oscillatory motion.

Plastic deformation (indentation) occurs between the rolling elements and the races when the bearing is subjected to a moderate static stress due to the weight of the shaft and the other elements. Its size gradually increases as the stress increases. However, the plastic deformation must not be so great as to influence the operational properties of the bearing in its rotational movement. As defined in ISO76, the static characteristic value $S_0 = C_0/P_0$ is a safety factor against detrimental plastic deformation which is a measure of the sufficient static load capacity.

The static load number, which is used to determine the bearing size, can be determined by taking into account the safety margin which depends on the operating conditions:

$S_0 > 2$	for shocks and impacts as well as exacting requirements for smooth operation and for axial double row self-aligning roller bearings
$S_0 = 1$	for normal operating conditions and low noise requirements
$S_0 = 0.5...0.8$	for smooth and non-impact operating conditions with few requirements (non-loaded bearings with adjusting or swivel motion)

27.2.3 Bearing calculation with inner geometry

The calculation of the bearing reference rating life is based on ISO/TS 16281.

The results of this calculation are the reaction forces, torques, the displacements and rotations, the maximum Hertzian pressure on the inner and outer race (right and left ring for a thrust bearing), the static safety, the reference and modified reference rating life in hours, the stiffness matrix at the operating point, and the load distribution and pressure curve on each rolling element.

For more detailed information, see 25 - Roller bearing inner geometry (see section "Roller Bearings (Inner Geometry)" on page III-751)

If the bearing inner geometry is given by the manufacturer then it is used in the calculation. If it is unknown then KISSsoft runs an approximation method that tries to determine the inner geometry using the bearing load ratings (both static C0 and dynamic C) given by the manufacturer. This procedure is based on ISO 76 and ISO 281-4 and normally gives quite useful results.

In the special case that the user only knows the number of the rollers or balls, and wants to use this data when performing a calculation in accordance with the standard, we suggest the following:

- Run a calculation based on bearing inner geometry. Since you don't know the geometry, it will be approximated for you.
- Create a bearing report, and note down the bearing inner geometry data.
- Open the KISSsoft database with write access authorization. Navigate to the required bearing type, and add a new bearing. In the "Internal geometry" tab, copy in all the internal geometry data you noted down in step 2. In the number of rolling elements field (Z), input the number of bearings you know are being used.
- Save and close the database.
- Go to the Elements editor, and update your bearings to the one you added to the database above. Rerun the calculation and get the results.

If the internal geometry the user added to the database is not sufficient or incorrect, then the input is ignored and the internal geometry is approximated. The log then contains a note to say that an approximate value has been used for the inner geometry.

The internal geometry cannot be taken into account in the calculations for every bearing type. The calculations where this is currently possible are listed in the Roller bearing inner geometry (see page I-147) database chapter.

27.3 Thermally permissible service speed

The definition of the thermally permissible operating speed is described in DIN 732 [8]. The calculation of the thermally permissible operating speed is based on a heat balance at the bearing. The thermally permissible operating speed is derived from the thermal reference speed and by using the thermal nominal speed. The result of this calculation is the speed that will be reached by the bearing running at the permitted temperature in an actual situation. This thermally admissible speed may differ greatly from other operating speed limits, depending on lubrication type, because the reference conditions only apply to quite specific cases. In order to define the thermally permissible operating limit, you must first define the thermal nominal speed for each case.

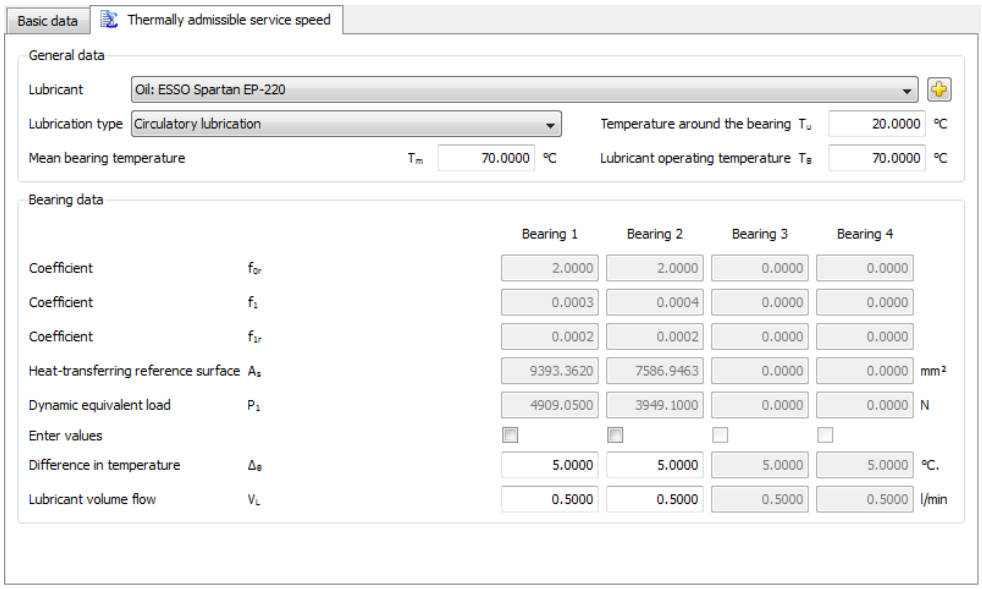


Figure 26.2: Thermally admissible operating speed

27.3.1 Thermal reference speed

The definition of the thermal nominal operating speed is defined in DIN ISO 15312 [7]. The thermal nominal speed is the bearing-specific speed reached under a given set of nominal operating conditions such that equilibrium is achieved between heat development (friction) and heat dissipation (through bearing contact and lubricant). Mechanical or kinematic criteria are not taken into account for this speed. The reference values (temperatures, load, viscosity of the lubricant, reference face of the bearing, . . .) are fixed so that the reference speed using oil or grease lubricated bearings will result in identical values.

27.3.1.1 Dissipated Heat Flows

The heat flow Qr is calculated from the reference heat flow density specific to a roller-bearing arrangement qr (for heat flow dissipated through bearing contact and lubricant) as well as heat dissipation through the reference surface Asr.

$Qr = 10^{-6} * qr * Asr$

qr, Asr are defined under reference conditions in accordance with DIN ISO 15312.

27.3.1.2 f0r and f1r coefficients

The coefficients f0r and f1r used to define the reference thermal operating speed are different, depending on which bearing type/series (also lubrication type for f0r) is used. They are shown in Table A.1 of the standard. Not all bearing variants are listed in the table.

The following values have been assumed for bearings and bearing types for which no data has been defined in the standard:

	f0r (tabular value)	f1r
Ball bearing	1.7	0.00015
Roller bearing	3	0.0003
Thrust ball bearing	1.7	0.00015
Thrust roller bearing	3.5	0.0015

27.3.1.3 Calculating the thermal nominal speed

The dissipating heat flows and the friction power are set as equal values so that the energy balance of the bearing is correct. The equation for the energy balance is:

$NFr = 10^{3} * Qr$

NFr: Friction power [W]
Qr : dissipated heat flow: [kW]

The subsequent equation becomes:

$(\pi * n\theta r)/30 * (10^{-7} * f0r * (\nu r * n\theta r)^{2/3} * dm^{3} + f1r * P1r * dm) = qr * ASr$

nθr: thermal reference speed [1/min]
f0r: Coefficient from Table A.1, DIN ISO 15312 [-]
r: Reference viscosity[mm2/s]

dm: average roller bearing diameter [mm]
f1r: Coefficient from Table A.1, DIN ISO 15312 [-]
P1r: Reference load [N]
qr: Roller bearing specific reference heat flow density (bearing contact, lubricant) [kW/m^2]
ASr: Reference surface area dissipating heat [mm2]

The value nθr can be determined from this equation.

27.3.2 Process for calculating thermally permitted operating speed (DIN 732-2)

As, when calculating the reference thermal operating speed, this calculation is based on equilibrium in the bearing. Dissipating heat flow:
$Q = Q_S + Q_L + Q_E$

Q_S: heat flow
$Q_L$dissipated across the bearing contacts by lubrication (only when there is circulatory lubrication) (the lubricant density ρ = 0.91 kg/dm3 and specific heat capacity cL = 1.88 KJ/(kg *K) are predefined.)
Q_E: additional heat flow (it is assumed that Q_E= 0 for the calculation)

27.3.2.1 Friction coefficients f0 and f1

The coefficient values f0 and f1 and the dynamic equivalent Load P1, are only needed to define the load and lubrication parameters. These values differ depending on the specific bearing type/model, lubrication type, or load direction. They are listed in Table 1 in the standard. Not all bearing variants are listed in the table. The following values for various types of lubrication have been defined (and incorporated in KISSsoft). They are based on the notes about f0 in Table A.1 in the standard.

- Oil, dip lubrication, bearing in oil mist: f0 = 0.5 * f0 (tabular value)
- Oil, dip lubrication, oil level up to middle bearing: f0 = 2.0 * f0 (tabular value)
- Oil, dip lubrication, oil level up to middle of the lowest rolling element: f0 = 1.0 * f0 (tabular value)
- Oil, circulatory lubrication: f0 = 2.0 * f0 (tabular value)
- Grease, run-in bearing: f0 = 1.0 * f0 (tabular value)

- Grease, newly greased: f0 = 2.0 * f0 (tabular value)

The following values have been assumed for bearings and bearing types for which no data has been defined in the standard:

	P1	f0 (tabular value)	f1
Ball bearing	3.3*Fa - 0.1*Fr (P1 <= Fr) where P1 = Fr	1.7	0.0007*(P0/C0)^0.5
Roller bearing	Fr	3	0.0003
Thrust ball bearing	Fa	1.7	0.0007*(P0/C0)^0.5
Thrust roller bearing	Fa	3.5	0.0015

27.3.2.2 Calculating the thermally permissible operating speed limit

The thermal permissible operating speed is calculated from the reference thermal operating speed with the help of the speed ratio, nθ = fn * nθr

The load and lubrication parameters have to be calculated before the speed ratio can be iterated from the following equation.

$KL * fn^{5/3} + KP * fn = 1$

Load parameter *KL*:

$KL = 10^{-6} * (\pi/30) * n\theta r * 10^{-7} * (f0r * n^{2/3} * n\theta r^{2/3} * dm^3)/Q$

Lubrication film parameter *KP*:

$KP = 10^{-6} * (\pi/30) * n\theta r * (f1 * P1 * dm)/Q$

nθr: thermal reference speed [1/min]
f0: Coefficient of friction from Table A.1, DIN 732 [-]
f1: Coefficient of friction from Table A.1, DIN 732 [-]
n: Lubricant viscosity [mm^2/s]
dm: average roller bearing diameter [mm]
P1: Reference load [N]
Q: Total dissipated heat flows [kW]

27.4 Moment of friction

In a roller bearing, the heat development, and therefore the operating temperature, is caused by the friction between individual components. The moment of friction in the roller bearing is indicated by several distinct losses due to resistance during operation. Select this calculation method under `Calculation > Settings`.

You can only perform this calculation by clicking the "Modified rating life according to ISO 281" option (`Basic data tab/Strength`).

27.4.1 Calculation in accordance with SKF Catalog 2004

As this calculation has to take into consideration a myriad of factors and influences, it is only performed if selected as an option in the extended service life calculation. However, this calculation can also be performed without these default values. The calculation of the total moment of friction in accordance with the 2004 SKF catalog is determined by a combination of rolling and sliding friction in the roller contacts (between rolling elements and cage, the bearing surface, the lubricant, and the sliding friction from grinding seals caused in sealed bearings). The calculation of the moment of friction depends on various coefficients:

- Load
- Type of bearing
- Bearing size
- Operating speed
- Lubricant properties
- Lubricant quantities
- Seals

The following working conditions must be present for the calculation to be performed:

- Grease or oil lubrication (oil bath, oil mist, or oil injection process)
- Load equal or greater than minimum load
- Load constant in size and direction
- Nominal operating clearance

If the rating is less than the minimum load, the calculation continues using the minimum load. If a minimum load value has been entered in the database, this value is used. If not, the software will determine this value. In the case of radial bearings, the minimum load is converted into a minimum radial force. In thrust bearings, the minimum axial force is defined by the software. The value for the minimum load is not used here.

The formula for the total moment of friction is:

$M = \phi_{ish} * \phi_{rs} * M_{rr} + M_{sl} + M_{seal} + M_{drag}$

ϕ_{ish}: **Lubricant film thickness factor**
In a lubricant flow, the lubricant is exposed to shear forces caused by the movement of the rolling elements. This produces heat and therefore reduces rolling moment of friction.

ϕ_{rs}: **Lubricant displacement factor**
The constant rolling action squeezes excess lubricant away from the contact zone of the rolling elements. This reduces the thickness of lubrication film and therefore reduces the rolling moment of friction.

M_{rr}: **Rolling moment of friction**
The rolling moment of friction depends on the type of bearing, the average diameter, the radial and axial loading, the rotation speed, and the viscosity of the lubrication. The design coefficients required to calculate the rolling moment of friction are defined using the series of the roller bearing. The design coefficients and coefficients used in the calculation are taken from the SKF Catalog 2004.

Assumptions have been made for those bearing types and bearing series for which no design coefficients were defined in the catalog, so that the moment of friction can still be calculated despite this. Definitions of design coefficients and the corresponding formulae:

	Factors	**Formula(e)**

Ball bearing	R1 = 0.00000042; R2 = 1.7;
	S1 = 0.0037; S2 = 55;
		If (Fa <= 0)
		Grr = R1*dm^1.96*Fr^0.54;
		Gsl = S1*dm^(-0.26)*Fr^(5/3);
		otherwise
		alfaF = 24.6*(Fa/C0/1000)^0.24;
		Grr = R1*dm^1.96*(Fr+R2*Fa/sin(alfaF))^0.54;
		Gsl = S1*dm^(-0.145)*(Fr^5+S2*dm^1.5/sin(alfaF)*Fa^4)^1/3;

Roller bearing	R1 = 0.00000122; S1 = 0.16; S2 = 0.0015;	Grr = R1*dm^2.41*Fr^0.31; Gsl = S1*dm^0.9+S2*dm*Fr;

Thrust ball bearing	R1 = 0.00000225; S1 = 0.16;	Grr = R1*dm^1.83*Fa^0.54; Gsl = S1*dm^0.05*Fa^(4/3);
Thrust roller bearing	R1 = 0.00000225; S1 = 0,154;	Grr = R1*dm^2.38*Fa^0.31; Gsl = S1*dm^0.62*Fa;

M_{sl}: **Sliding moment of friction**
The sliding moment of friction depends on the type of bearing, the average diameter, the radial and axial loading, and the viscosity of the lubrication. The design coefficients required to calculate the sliding moment of friction are defined using the series of the roller bearing. You will find the coefficients used for this calculation in the SKF 2004 catalog.

M_{seal}: **Moment of friction for grinding seals**
The moment of friction for grinding seals depends on the bearing type, the bearing size, the diameter of the seal-lip mating surface, and the layout of the seal. As the type of seal, the diameter of the seal-lip mating surface, and the seal layout, differ from one manufacturer to another, it is difficult to define a generally applicable moment of friction. You will find values for the seal types used in your bearings in the SKF catalog, which is integrated in the KISSsoft software. If you find a familiar gasket label in the bearing label, the moment of friction for grinding gaskets is calculated using the coefficients listed in the catalog. Otherwise it is set to zero.

Example of a seal label in the name of a roller bearing:

`SKF: 623-2RS`: this means that the bearing has a RS1 type seal on both sides. The KISSsoft system then searches for names with "-2RS1". If this label is present, the coefficients from the SKF catalog are applied and the moment of friction for grinding seals is calculated.

In KISSsoft, the diameter of the mating surface is calculated with:

ds = d + (D - d) * 0.2

M_{drag}: **Moment of friction caused by lubrication losses**
This moment of friction is caused by flow, splash, or injection losses during oil bath lubrication. To calculate this torque, you must also input the oil level (h_{Oil}), which you specify in `Calculation > Settings`. You will find a more detailed description of this entry in the Oil level and Lubrication type (see page III-750) section. The design coefficients KZ and KL for roller bearings with a cage are also applied to toroidal roller bearings (CARB).

27.4.2 Calculation in accordance with SKF Catalog 1994

The prerequisite for calculating the moment of friction is that the bearing rotating surfaces must be separated by a film of lubrication. The total bearing moment of friction results from the sum:

$$M = M_0 + M_1 \tag{26.1}$$

M_0: load-independent moment of friction

M_0 is determined by the hydrodynamic losses in the lubricant. It is especially high in quickly rotating, lightly loaded bearings. The value M_0 depends upon the quantity and viscosity of the lubricant, as well as the rolling speed.

M_1: load-dependent moment of friction

M_1 is limited by the elastic deformation and partial sliding in the surfaces in contact, especially due to slowly rotating, heavily loaded bearings. The value M_1 depends on the bearing type (bearing-dependent exponents for the calculation), the decisive load for the moment of friction and the mean bearing diameter.

For axially loaded cylindrical roller bearings, an additional axial load-dependent moment of friction, M_2, is added to the formula:

$$M = M_0 + M_1 + M_2 \tag{26.2}$$

M_2: axial load-dependent moment of friction

M_2 depends on a coefficient for cylindrical roller bearings, the axial loading, and the bearing's average diameter.

Coefficients f0, f1 (see page III-737) and P1 (values that depend on the bearing type and bearing load) used for the calculation have been taken from DIN 732. The formulae, exponents and coefficients are taken from the SKF Catalog, 1994 Edition.

27.5 Maximum Speeds

Roller bearings are reliable and can be expected to reach their calculated service life as long as the maximum speed (speed limit) is not exceeded. This depends on the type, size and lubrication.

A warning message appears if the maximum permissible speed is exceeded.

Depending on the lubrication type, the actually permitted maximum speed can be much lower. For more details, see the "Thermal admissible operating speed" section 26.3.

27.6 Service life

The nominal service life is calculated using the formulae given in ISO 281 and corresponds to the formulae that can also be found in the manufacturers' catalogs. Usually the service life is calculated at 90% (10% probability of failure, see also section 26.7) in hours. The label used here is L_{10h} (h: hours; 10: probability of failure).

27.6.1 Extended service life calculation in accordance with Supplement to DIN ISO 281 (2007)

ISO 281 contains the regulations for "modified service life" which take into account the influence of loads, lubricant conditions, materials specifications, type, material internal stresses and environmental factors.

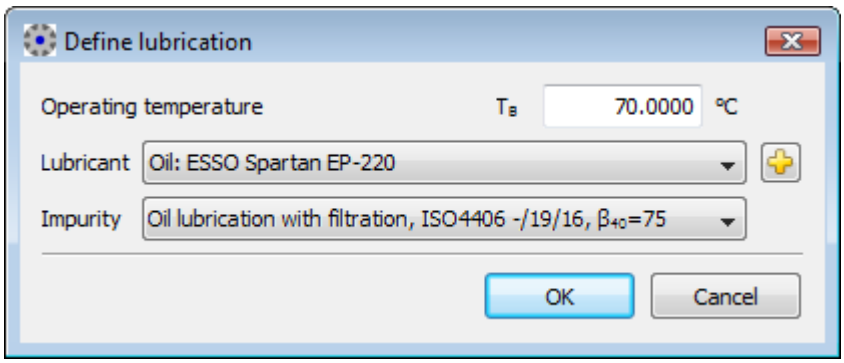


Figure 26.3: Dialog for extended service life calculation

The service life coefficient a_{ISO} can be defined as follows:

$$a_{ISO} = f\left(\frac{ec \cdot Cu}{P}, \kappa\right) \tag{26.3}$$

a_{ISO}:	service life coefficient from diagram [-]
ec :	impurity characteristic value [-]
Cu :	fatigue load limit [N]
P :	dynamic equivalent load [N]
κ	viscosity ratio = n_u/n_{u1}
n_{u1}:	reference viscosity diagram [$mm^2/2$]

n_u: VT diagram for the lubricant [mm^2/2]

The fatigue load limit Cu is specified by the bearing manufacturer. If none of these values are known, you can calculate them with the approximate formula as defined in ISO 281.
The impurity characteristic value ec (between 0 and 1) is taken directly from the degree of cleanliness.

27.6.2 Service life calculation with load spectra

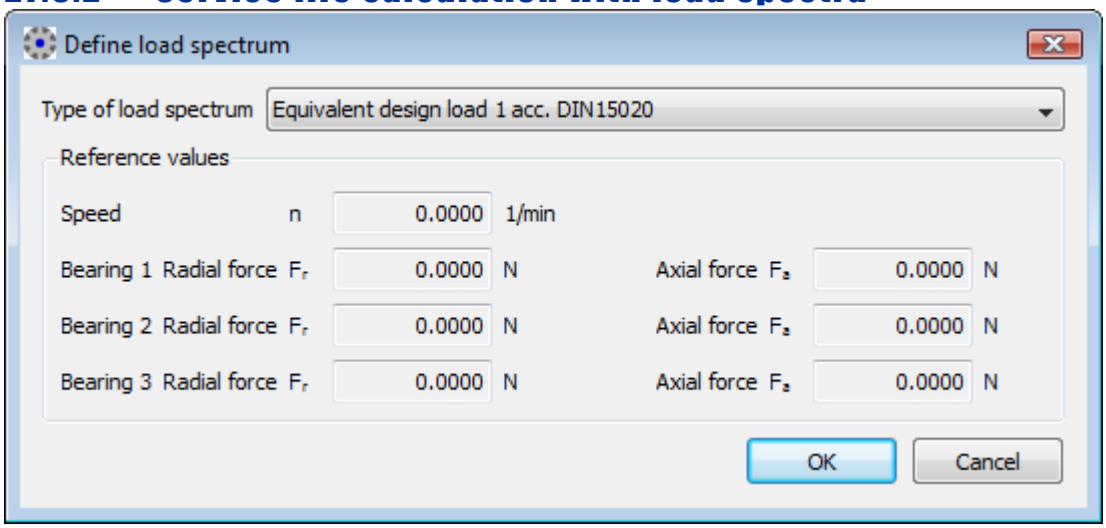


Figure 26.4: Dialog for selecting the load spectrum

The load spectrum on the bearing has these values:

k number of elements in the load spectrum

q_i: frequency (load spectrum element i) (%)

n_i: speed (load spectrum element i) (rpm)

F_{ri}: radial force (load spectrum element i) (N)

F_{ai}: axial force (load spectrum element i) (N)

You can take this load spectrum data from the shaft calculation, in which case you may obtain different load spectra for radial and axial forces. Or alternatively you can select a load spectrum from the database. For bearing forces, the important factor here is the torque factor (not the load factor) and a negative prefix operator will only affect the axial force.

Achievable service life with simple calculation approach

You calculate the service life by defining an equivalent design load and the average speed. You can then use the usual formulae to calculate the service life.

$$n_m = \frac{n_1 \cdot q_1}{100} + \frac{n_2 \cdot q_2}{100} + \ldots + \frac{n_k \cdot q_k}{100} \tag{26.4}$$

$$P_m = \sqrt[p]{\frac{P_1^p \cdot q_1}{100} + \frac{P_2^p \cdot q_2}{100} + \ldots + \frac{P_k^p \cdot q_k}{100}} \tag{26.5}$$

n_m: average speed

p exponent in the service life formula (3.0 or 10/3)

P_i: dynamic equivalent load (load spectrum element i)

P_m: average dynamic equivalent load

Achievable service life with extended service life calculation:

When the Extended service life calculation is used, the service life is calculated separately for every equivalent load spectrum element. The result is then used to determine the total service life:

$$L_{hna} = \frac{100}{\frac{q_1}{L_{hna1}} + \frac{q_2}{L_{hna2}} + \ldots + \frac{q_k}{L_{hnak}}} \tag{26.6}$$

L_{hnai}: service life (load spectrum element i) in the case of speed n_i and load F_{ri}, F_{ai}

L_{hna}: Total service life

27.7 Failure probability

Normally, the failure probability is assumed to be 10%. This means there is a 90% probability that the nominal service life will be achieved. In this case the coefficient a_1 is equal to 1.0. If the failure probability value has to be lower, this coefficient must also be lower (at 1%, a_1 = 0.21).

You define the failure probability in `Calculation > Settings`.

27.8 Bearings with radial and/or axial force

For every bearing, you can specify whether it is subject to a radial or axial force. If the bearing is subject to axial force, you must also specify whether the force is applied in both directions (<>), in the direction of the Y-axis (- >) or in the opposite direction (< -).

27.9 Calculating axial forces on bearings in face-to-face or back-to-back arrangements

Because of the inclination of the races in the bearing a radial load generates axial reaction forces in taper roller bearings, high precision angular contact ball bearings and angular contact ball bearings, this data must be taken into account when the equivalent design load is analyzed.

Axial reaction forces are calculated in accordance with SKF (roller bearing catalog) which exactly match the values defined in FAG.

For bearings in an back-to-back arrangement, left bearing A, right bearing B, outer axial force in A-B direction, the following data applies:

Condition	**Formula**
$\frac{F_{rA}}{Y_A} \leq \frac{F_{rB}}{Y_B}$	$F_{aA} = F_a + \frac{F_{rB}}{2 \cdot Y_B}$ $F_{aB} = F_a - F_{aA}$
$\frac{F_{rA}}{Y_A} > \frac{F_{rB}}{Y_B}$ $F_a > \frac{1}{2} \cdot \left(\frac{F_{rA}}{Y_A} - \frac{F_{rB}}{Y_B} \right)$	$F_{aA} = F_a + \frac{F_{rB}}{2 \cdot Y_B}$ $F_{aB} = F_a - F_{aA}$
$\frac{F_{rA}}{Y_A} > \frac{F_{rB}}{Y_B}$ $F_a \leq \frac{1}{2} \cdot \left(\frac{F_{rA}}{Y_A} - \frac{F_{rB}}{Y_B} \right)$	$F_{aA} = F_a + F_{aB}$ $F_{aB} = \frac{F_{rA}}{2 \cdot Y_A} - F_a$
F_{rA}, F_{rB} Y_A, Y_B F_a F_{aA}, F_{aB}	Radial force on bearing A, B Y coefficient of bearing A, B External axial force Axial force on bearing A, B

For all other cases, (face-to-face arrangement or axial force in the other direction) simply reverse the formula.

These calculated pretension force values are displayed in the main window. If the actual internal forces are higher, for example, due to the use of spring packages, you can change the value manually.

27.10 Oil level and Lubrication type

Input the oil level and the lubrication type under `Calculation > Settings`. These entries are needed to define the moment of friction due to lubrication losses. The value h is given in the shaft calculation and results in the following formula for every bearing:

$$H = \frac{D}{2} - h \geq 0$$

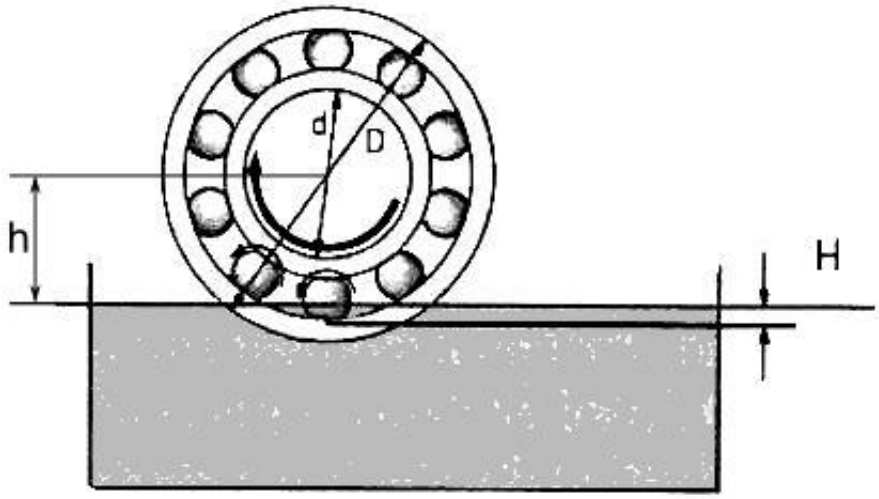


Figure 26.4: Oil level in the bearing

Two different types of lubrication can be defined:

- Oil bath lubrication
- Oil injection lubrication

If you select the `Oil injection lubrication` option, the value determined for oil bath lubrication is multiplied by 2 to give the lubricant loss.

Chapter 28
Roller Bearings (Inner Geometry)

In addition to the classic bearing calculation method (see 24 Roller Bearings (see section "Roller bearing" on page III-727)) KISSsoft also provides a calculation method that complies with ISO 16281. This method calculates the bearing loads and the service life of the bearing based on its inner geometry.

This method is available either as an Embedded Feature in shaft calculation (see 20.1.11 Roller bearings (see section "Roller bearing" on page III-677)) or as an stand alone KISSsoft module. Unless otherwise specified, this chapter describes the stand alone module because the majority of the functionalities and features are used by both these two variants.

This module is designed to be used by bearing experts, or users who know the inner geometry of their bearings. It makes it possible to calculate the life of a bearing, if you know the loading conditions. To start this module, go to the KISSsoft modules tree and double-click on "Shafts and Bearings" > "ISO 16281".

28.1 Bearing data tab

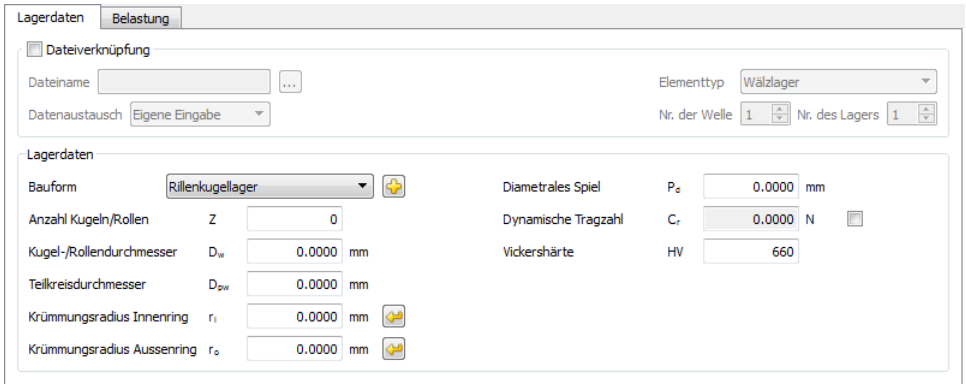


Figure 28.1 Bearing data tab

28.1.1 File interface

The user can use this module to link to a shaft calculation file. This allows bearing information to be transferred automatically from the shaft calculation file, without having to reenter the data. The user must input the:

- File name: name of the shaft calculation file (extension .W10), from which the selected bearing data will be extracted
- Element type: here the user selects whether the bearing is a roller bearing that belongs to a shaft, or a connecting roller bearing
- Shaft number: if the bearing belongs to a shaft, the user must input the shaft number here. The program then runs through the shafts Elements tree from top to bottom (see Figure 28.2)
- Bearing number: number of the selected bearing, either on the corresponding shaft or from the list of connecting elements. The program runs through the shafts Elements tree from top to bottom (see Figure 28.2)
- Data exchange: determines how data is exchanged between the shaft file and this module. In each case, the geometry of the selected bearing is transferred from the shaft file.
  - Bearing loads: the information transferred from the shaft file is the applied force and torque of the bearing as well as the lubricating conditions
  - Bearing displacements: the information transferred from the shaft file is the displacement and rotation of the inner ring of the bearing as well as the lubricating conditions

– Own input: only the geometry of the bearing is transferred. The user can determine their own load and lubrication conditions

Example:

a shaft file whose Elements tree has the structure shown in Figure 28.2. To extract the information for the bearing "Roller bearing 2" which belongs to "Shaft 1", the correct selection would be:

Element type = Roller Bearing, Shaft number = 1, Bearing number = 2

In contrast, to extract the information for the bearing "Roller bearing 1" from the list of connecting elements, the correct selection would be

Element type = Connecting Roller Bearing, Bearing number = 1

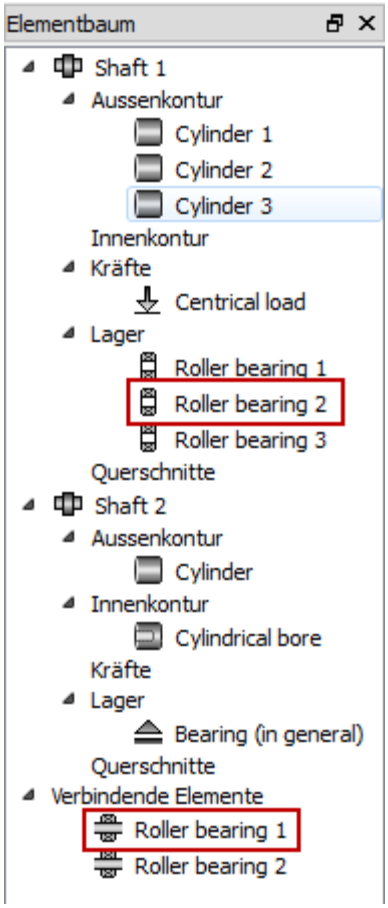


Figure28.2 Example of selecting the shaft and bearing number when linked to a shaft file.

28.1.2 Bearing data

This is where the geometry of the bearing is defined. You fill find more detailed information about this in 24.2.3 Roller bearing inner geometry (see section "Bearing calculation with inner geometry" on page III-734). In addition to the geometry data, you can also specify the dynamic load number, if you know it. If not, this number is calculated using the current geometry data as specified in ISO 281. If you require an extended service life (see 24.6.1 Enhanced bearing service life (see section "Extended service life calculation in accordance with Supplement to DIN ISO 281 (2007)" on page III-744)), input the fatigue load limit Cu. If Cu is not known, it will also be calculated on the basis of ISO 281.

Note for the shaft calculation: In this module, the effect of surface hardness on the static capacity can be taken into account by entering the Vickers hardness. You will find the formulae for this in [92]. The hardness value of every bearing calculated with their inner geometry is predefined as HV 660 for the shaft calculation.

28.1.2.1 Custom roller profile

The default roller profile used for roller bearings is the logarithmic profile as defined in ISO 16281. However, a custom roller profile can be used instead if required. Click on the plus sign next to the roller length input field and enter the name of the file with the required roller profile function (Figure 28.3a). The definition and coordinate frame of this file are shown in Figure 28.3b). The expected structure of this file is as follows:

```
-- this line is a comment
DATA
1     -0.45   0.000581256
2     -0.41   0.000390587
3     -0.37   0.000277616
4     -0.33   0.000200197
...
...
21    0.33    0.000200197
22    0.37    0.000277616
23    0.41    0.000390587
24    0.45    0.000581256
```

END

Notes:

- Lines that start with "--" are comments and are ignored.
- The profile function definition starts with the keyword DATA and ends with the keyword END
- Each line must contain three columns. The first column is the index and is included only as a reference for the user (its values have no effect). The second column is the non-dimensional position x/Lwe, for which the profile is defined in mm/mm. The values in this column should range between -0.5 and +0.5. The third column is the non-dimensional profile f/Dw, in mm/mm. The values in this column cannot exceed 0.5.
- To save space, the data represented by "..." has been omitted

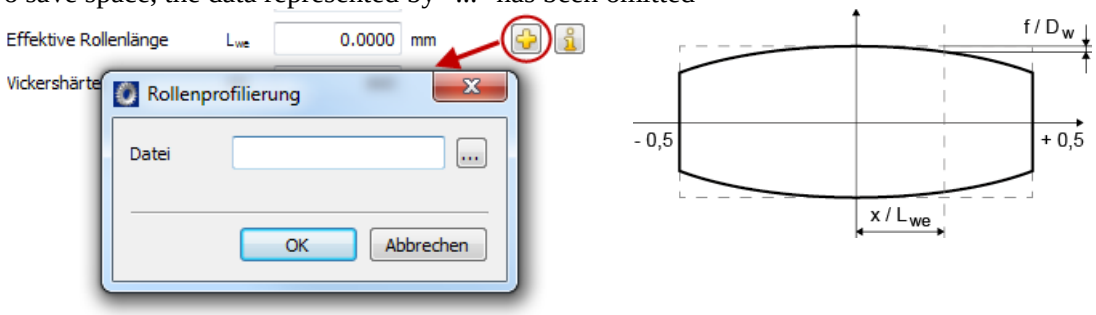


Figure28.3 (a) Definition of a custom roller profile file, (b) Coordinate frame for the definition of the custom roller profile

28.1.2.2 Bearing ring deformations

The inside/outer rings are usually assumed to be rigid (non-deformable). To take ring deformations into account, click on the plus button next to the bearing type definition (figure 28.4a). The expected structure for both files is as follows:

-- this line is a comment

DATA

0	0	0.00E+00	0.00E+00	5.00E-03
1	8	0.00E+00	6.96E-04	4.95E-03
2	16	0.00E+00	1.38E-03	4.81E-03
3	24	0.00E+00	2.03E-03	4.57E-03

...

```
...
41        328      0.00E+00        -2.65E-03        4.24E-03
42.336  0.00E+00        -2.03E-03        4.57E-03
43.344  0.00E+00        -1.38E-03        4.81E-03
44.352  0.00E+00        -6.96E-04        4.95E-03
45.360  0.00E+00        -1.23E-18        5.00E-03
END
```

Notes:

- Lines that start with "--" are comments and are ignored.
- The ring deformation definition starts with the keyword DATA and ends with the keyword END
- Each row must contain 5 columns. The first column is the index and is included only as a reference for the user (its values have no effect). The second column is the angle φ for which the deformation is specified. The next three columns are the x, y, and z components of the ring deformation, all defined in mm.
- To save space, the data represented by "..." has been omitted

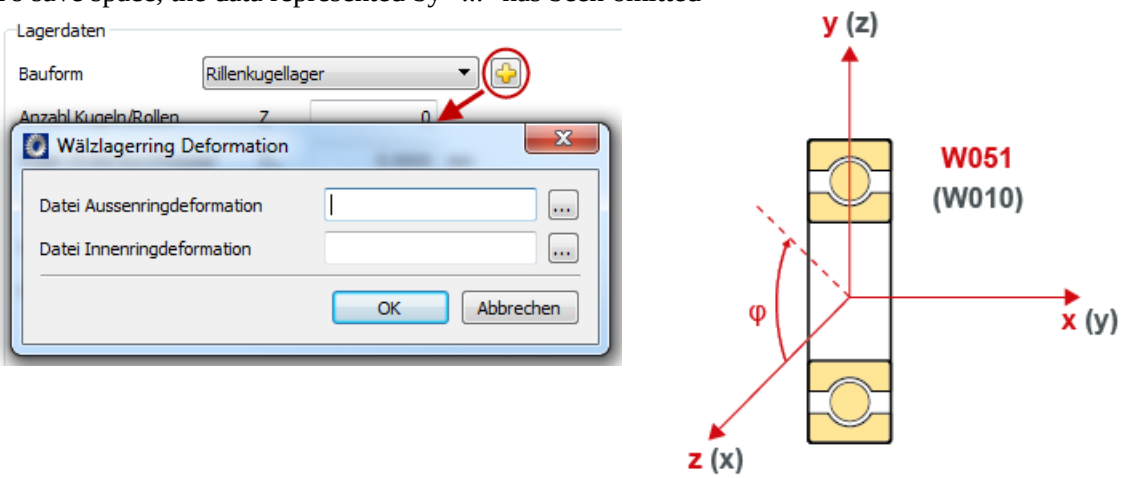


Figure 28.4 (a) Definition of ring deformations, (b) Coordinate frame of this module (W051), which defines the axial (x) and the radial directions (y, z). For the sake of clarity, the coordinate frame of the shaft module (W010) is also displayed.

Note for the shaft calculation: Ring deformations can only be processed in bearing calculation module W051, not in shaft calculation module W010.

28.2 Load tab

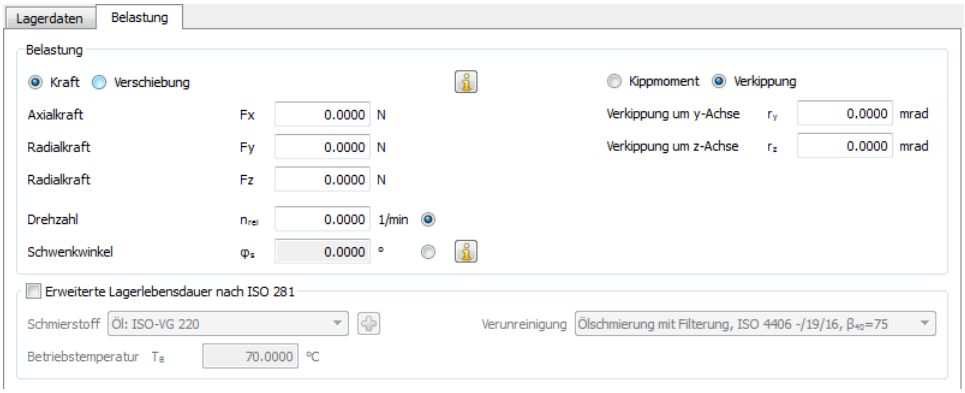


Figure28.5 Load tab

This tab is where the operating conditions of the bearing are defined.

28.2.1 Load

Four combinations of data can be entered here:

- (A) Force-Tilting Moment
- (B) Force-Tilting
- (C) Displacement-Tilting Moment
- (D) Displacement-Tilting

Speed: the speed of the inner ring relative to the outer ring. The outer ring is always assumed to be fixed (non-rotating).

Oscillation angle: the oscillation angle for partially rotating bearings. The lifetime in million oscillations is determined in accordance with [39].

Note for the shaft calculation: The default setting for the shaft calculation process is combination D.

Note: A complete oscillation measures $2 \cdot \varphi_s$

28.2.2 Enhanced service life calculation in accordance with ISO 281

The influence of lubrication, filtration and impurities on the service life can be taken into account here.

Lubricant: the lubricant used

Operating temperature: the temperature of the lubricant

Impurity: the class of the impurity

28.3 Graphics

28.3.1 Load distribution

This shows the load distribution over the rolling elements (balls/rollers).

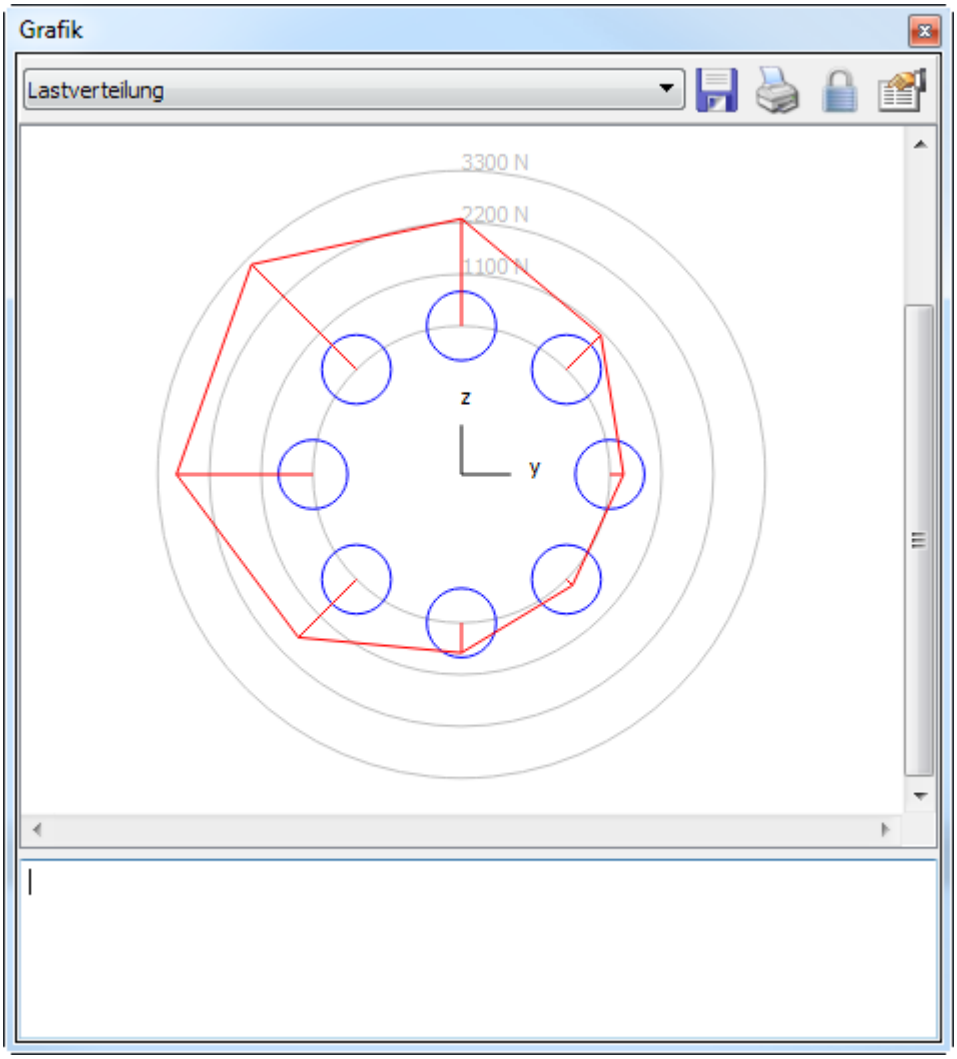


Figure 28.6 Load distribution

28.3.2 Pressure curve

This shows how the pressure develops along the length of each roller, or at every contact point in a ball bearing.

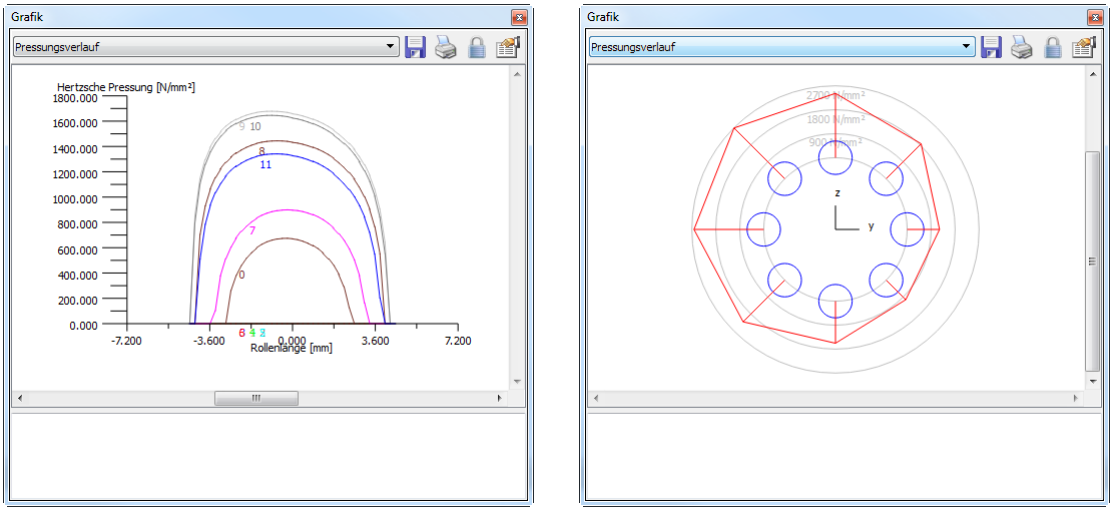


Figure28.7 Pressure curve in a (a) roller bearing, (b) ball bearing

28.3.3 Stiffness curve

This shows the force-displacement curve of the bearing. Both radial and axial stiffness are shown.

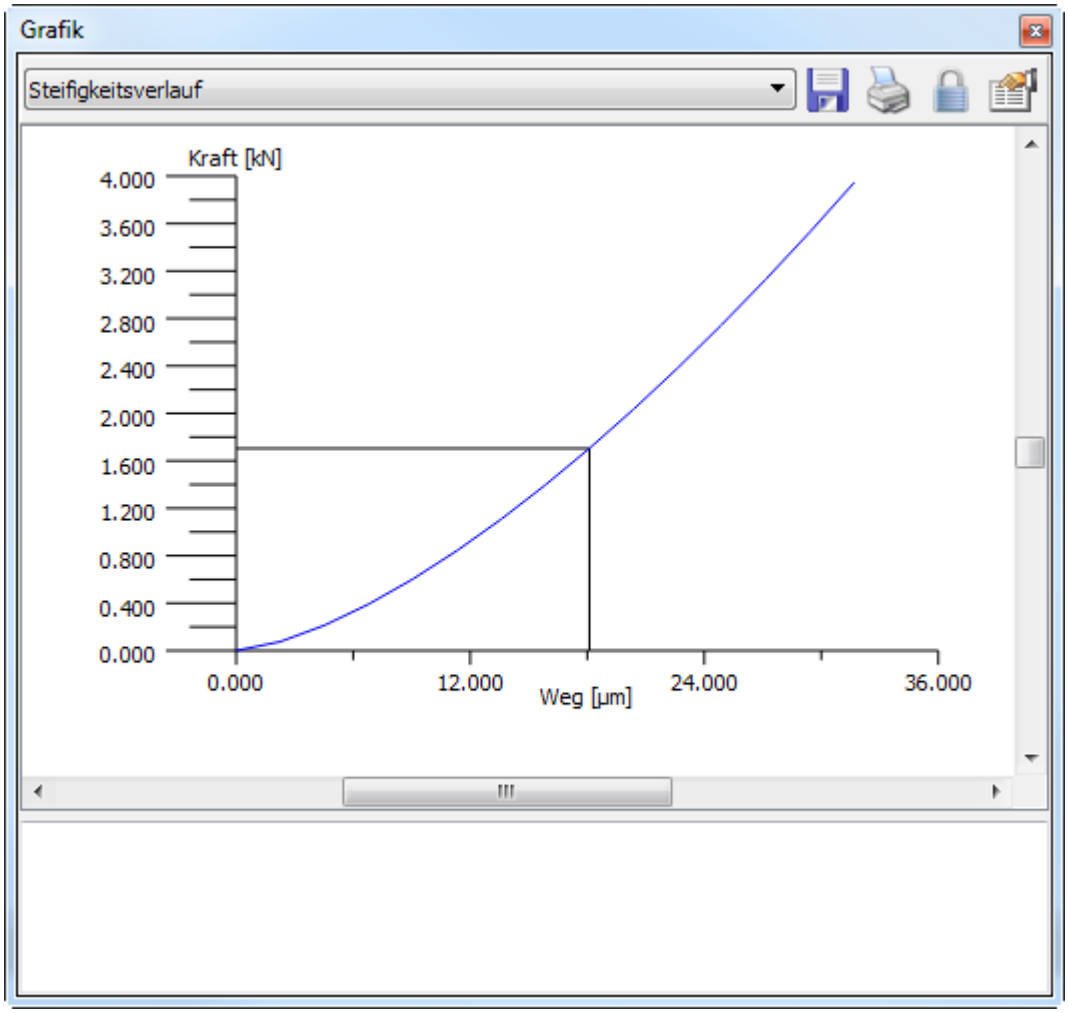


Figure28.8 Stiffness curve

28.3.4 Pressure curve for each rolling body

This graphic shows the pressure curve on each roller element along the roller profile.

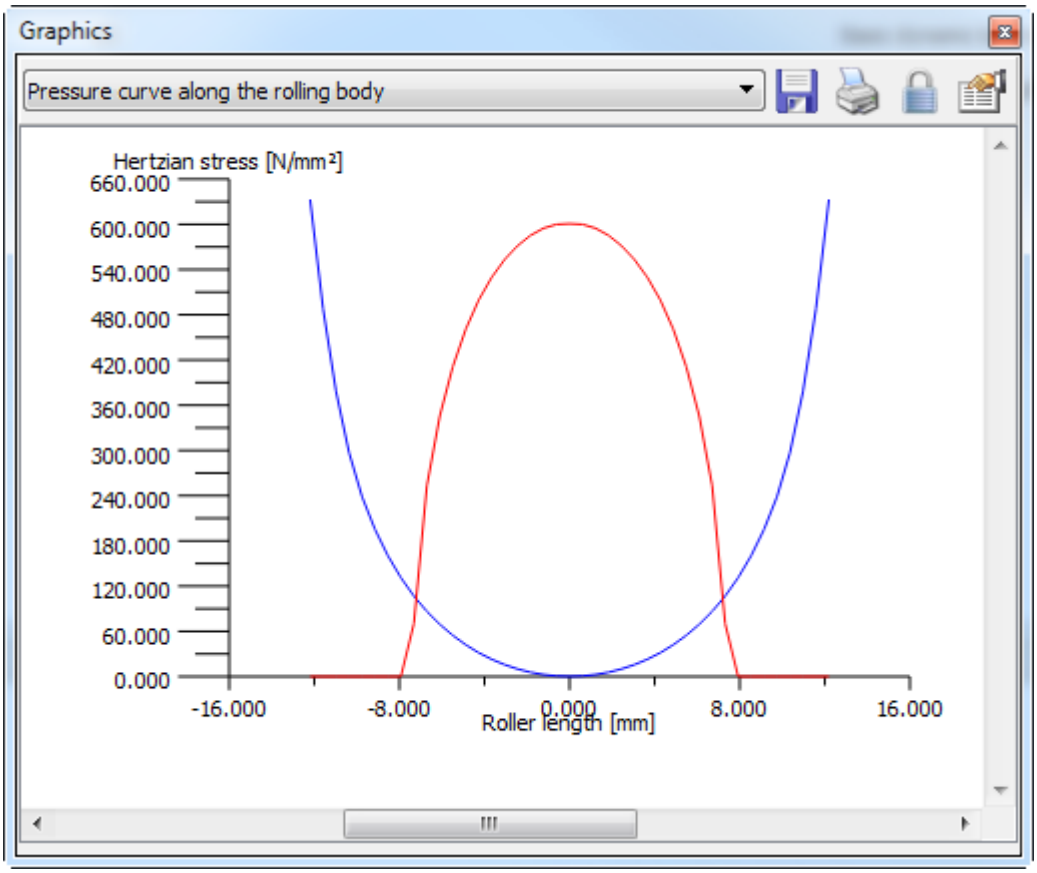


Figure28.9 Pressure curve along the rolling body

Chapter 28
Hydrodynamic plain radial bearing

Niemann [64] provides a very accurate method for calculating plain radial bearings that can run at high speeds. This also produces excellent results for oval-clearance or pivoted-pad plain bearings.

DIN 31652 [33] details a good method for calculating stationary, hydrodynamic plain radial bearings that are to run at low and medium speed.

29.1 Calculation methods

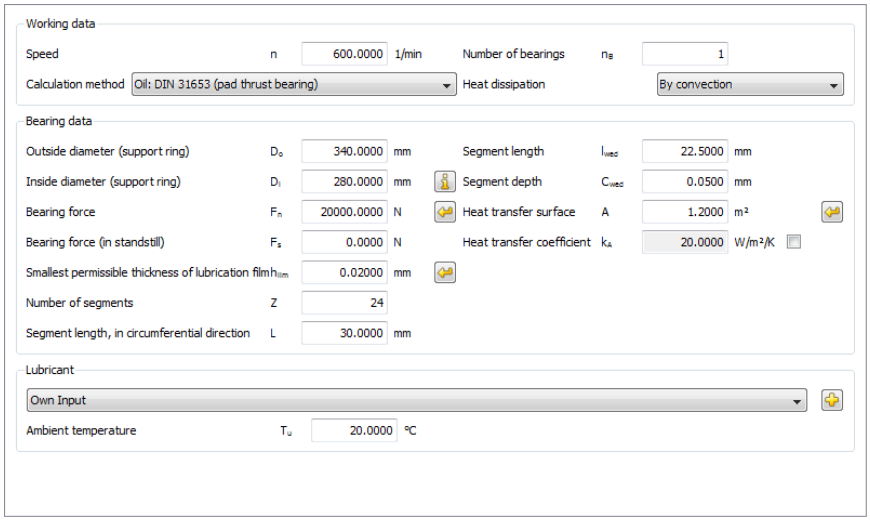


Figure 28.1: Basic data: Plain radial bearings

You can use one of these two methods to calculate oil-lubricated, hydrodynamic plain radial bearings:

a) In accordance with G. Niemann, Maschinenelemente I, 1981, [64].
This method is very suitable for quickly rotating bearings. This also produces excellent results for special construction types such as pivoted-pad bearings or oval-clearance plain bearings.
This method calculates the power loss, oil flow, oil temperature and minimum lubricant gap thickness in accordance with [64] and [57]. This calculation can only be used for pressure lubricated bearings (circulatory lubrication) when the service reliability is also tested.

b) In accordance with DIN 31652, Parts 1 to 3, 1983, [33].
This method is very suitable for slowly rotating bearings. It also determines the oil consumption, oil flow, and the entire heat balance. Calculation in accordance with DIN 31652, Parts 1 to 3 (1983 edition) for pressure-less and pressure lubricated bearings. This takes into account the way in which lubricant is applied (lubrication holes, lubrication groove, lubrication glands). It calculates all the operating data in accordance with DIN 31652, including the operating temperature, minimum lubrication gap width, power loss, oil flow etc. It also checks service reliability.

29.2 Module specific inputs

Calculating the volume-specific heat of the lubricant

The volume-specific heat of lubricants can be calculated in two ways:

- Take into account dependence on temperature
- By a simplified assumption (as in DIN 31652): $1.8 \cdot 10^6 J/(m^3K)$

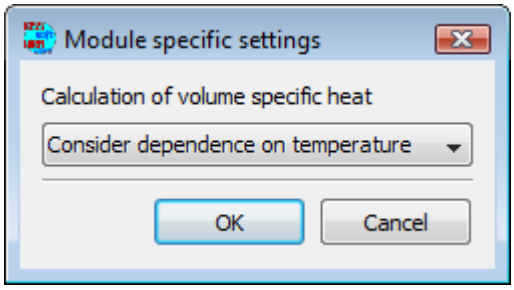


Figure 28.2: Module specific settings

29.3 Thermal expansion coefficients

To calculate the clearance, you require the thermal expansion coefficients of the shaft and (wheel or pinion) center.

These are the coefficients for the most important materials:

Steel	$11.5 \cdot 10^{-6}$
Cast iron	$11 \cdot 10^{-6}$
White metal	$18 \cdot 10^{-6}$
Composite bronze	$18 \cdot 10^{-6}$

29.4 Average surface pressure

You will find the permitted values in:

- Niemann, Volume I, Table 15/1, [64]
- DIN 31652, Part 3, Table 2, [33]

Permitted maximum values for surface pressure:

- White metal bearing: 1...3 N/mm^2
- Bronze: 1...8 N/mm^2

29.5 Lubrication arrangement

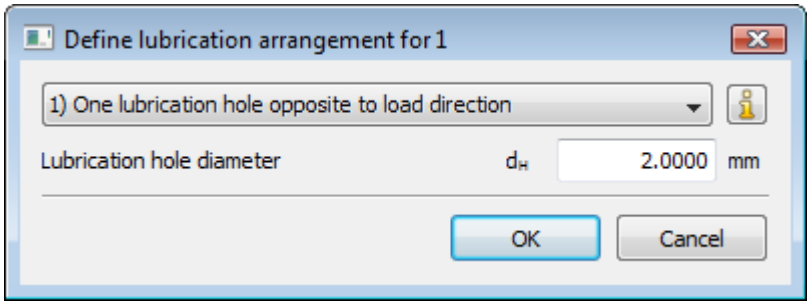


Figure 28.3: Selecting the lubrication arrangement

The different lubrication arrangements are shown in the next three Figures 28.4, 28.5 and 28.6.

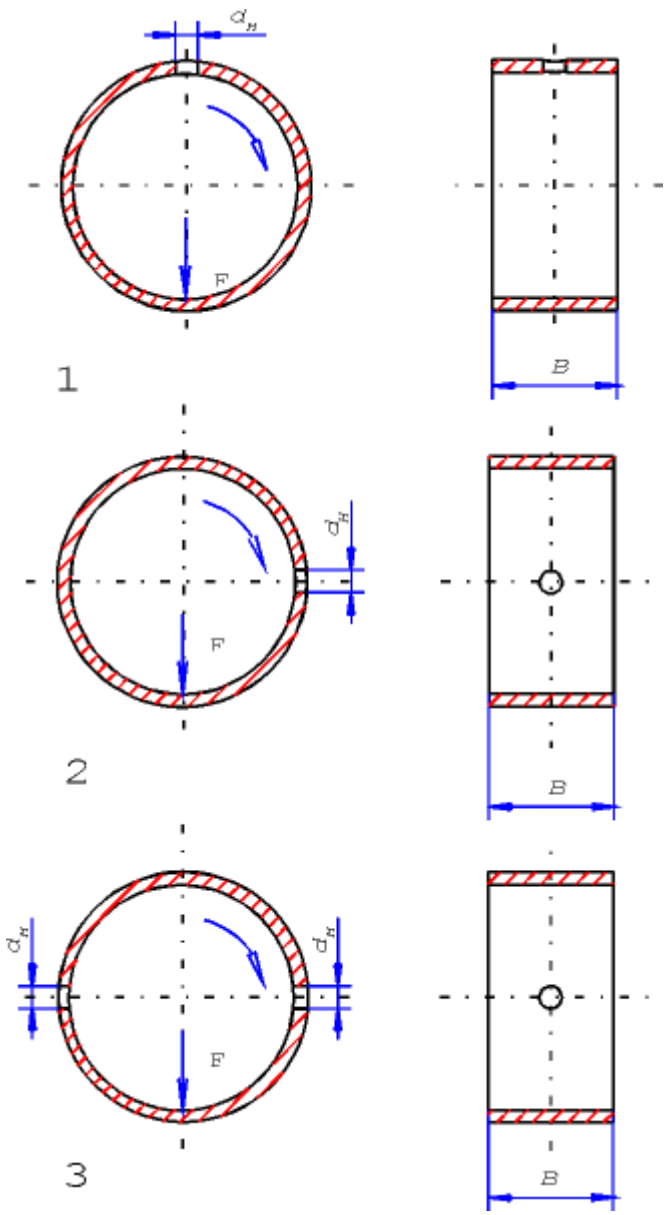


Figure 28.4:
1: One lubrication hole opposite to load direction.
2: One lubrication hole positioned at 90° to the load direction.
3: Two lubrication holes positioned at 90° to the load direction.

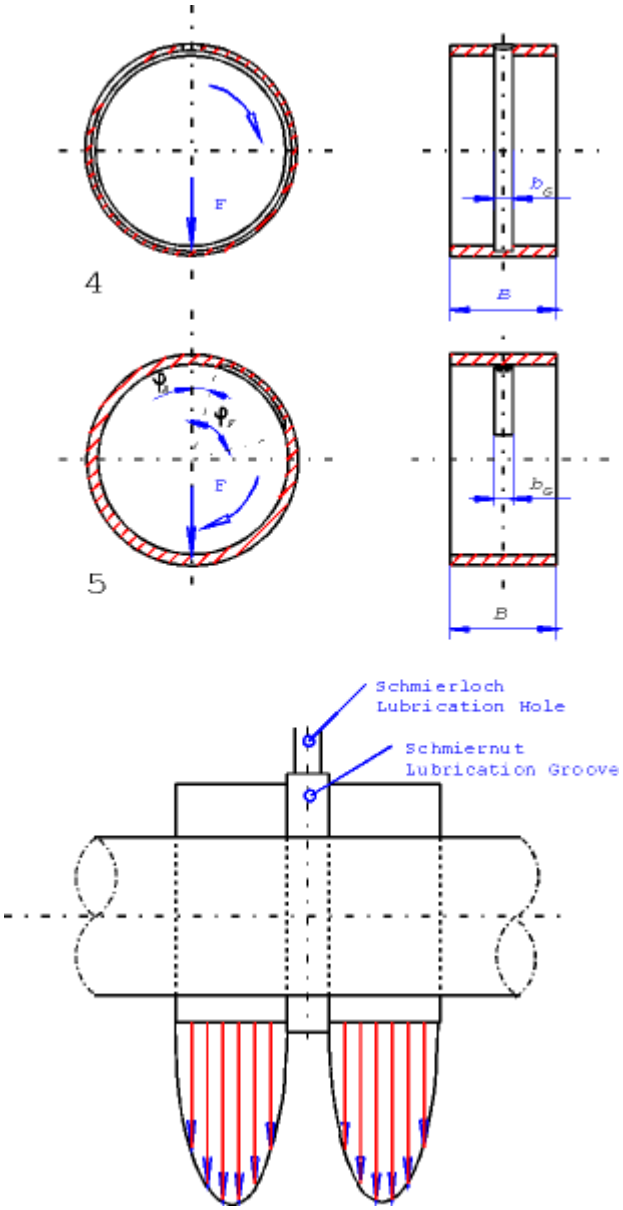


Figure 28.5:
4: Lubrication groove (ring groove).
5: Lubrication groove (circumferential groove).
Note: For lubrication with a circular groove, the calculation is performed for each bearing half with half the load! (see DIN 31652, Part 1, paragraph 3.4 [33]).

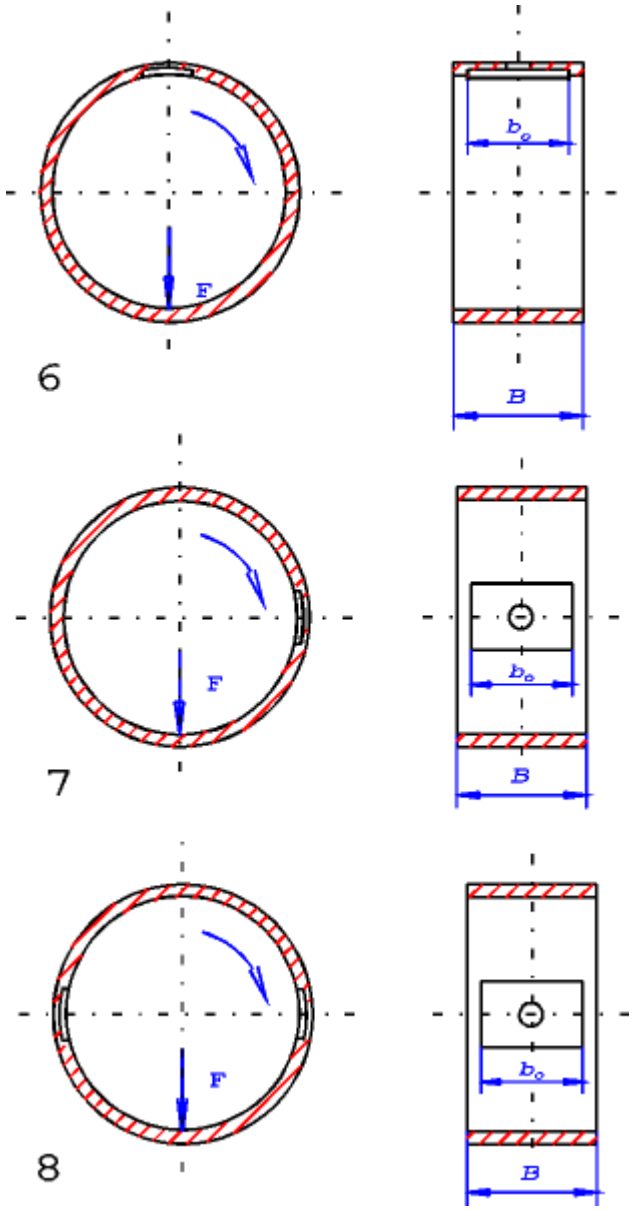


Figure 28.6:
6: One lubrication pocket opposite to load direction.
7: One lubrication pocket positioned at 90° to the load direction.
8: Two lubrication pockets positioned at 90° to the load direction.

29.6 Heat transfer surface

If the values of the heat transfer surface are not known, you can take 10 * d * b to 20 * d * b as a guide value.

d : Bearing diameter
b : Bearing width

29.7 Heat transfer coefficient

If the value of the heat transfer coefficient is not known, you can take 15 to 20 (W/m^2K) as a guide value.

29.8 Oil temperatures

Oil exit temperature:

- normally approximately 60°
- Upper limit for usual mineral oils: 70° to 90°

Oil entry temperature:

- With the usual cooler: 10°C lower than the output temperature
- With a very efficient cooler: 20°C lower than the output temperature

29.9 Sizing the bearing clearance

Bearing clearance = d_bore - d_shaft

In general, a greater bearing clearance makes the bearing more stable and allows it to cool more effectively, however it also results in a reduction in load capacity.

- **Suggestion in accordance with Niemann**
  Suggestion for metal bearings in mechanical engineering in accordance with Niemann, volume I, table. 15/2, [64].

  The following applies for other materials:

Cast iron bearing	:	0.001 * d
Light metal bearing	:	0.0013 * d
Sintered bearing	:	0.0015 * d
Plastic bearing	:	0.003 * d

  d : Bearing diameter

- **Suggestion in accordance with DIN 31652**
  Suggestion for metal bearings in mechanical engineering in accordance with DIN 31652, Part 3, Table 4, [33].
  In this sizing method you can either use the proposal in accordance with DIN 31652 can be adopted, or calculate the clearance from the given output temperature (only where the lubricant is used to dissipate the heat).

- **Suggestion in accordance with K. Spiegel**
  Suggestion for the clearance in accordance with K. Spiegel: Goettner equation clearance: (2.5+50.0/d)/1000.0*d

29.10 Sommerfeld number

You must calculate the Sommerfeld number because it is an important characteristic value for sliding bearings.

Sommerfeld number > 1 occurs in heavily loaded bearings at the limit for b/d: $0 < b/d \leq 2$

Sommerfeld number < 1 occurs in quickly rotating bearings at the limit for b/d: $0.5 < d/b \leq 2$

d : Bearing diameter
b : Bearing width

29.11 Bearing width

Reference value for bearing width as defined in Niemann, volume I, table. 15/1, [64]
Normal range: b/d = 1 to 2

Reference value for bearing width in accordance with DIN 31652, [33]
Normal range: b/d = 0.125 to 1

d : Bearing diameter
b : Bearing width

29.12 Permissible thickness of lubrication film

The suggested value is taken from DIN 31652, Part 3, Table 1, [33].

The values in this table are all empirical values. They therefore assume an average roughness height of < 4μ, low levels of form error, and that the lubricant is to be filtered appropriately.

Chapter 29
Hydrodynamic plain thrust bearing

The DIN standard provides two methods for calculating hydrodynamic plain thrust bearings, according to type.

- Calculation of pad thrust bearings in accordance with DIN 31653 [34]: This standard applies to bearings that have fixed sunken surfaces for lubrication (see Figure 29.2) which are separated from the rotating discs by a film of lubricant.
- Calculation of tilting-pad thrust bearings in accordance with DIN 31654 [35]: This standard applies to bearings that have movable tilting pads (see Figure 29.3) which are also separated from the rotating discs by a film of lubricant.

If you do not consider the influence of the center of pressure on the tilting-pad thrust bearings, the same calculation procedure is described in both standards, which is why it is described here only once. However, any significant variations to these two standards will get a special mention here.

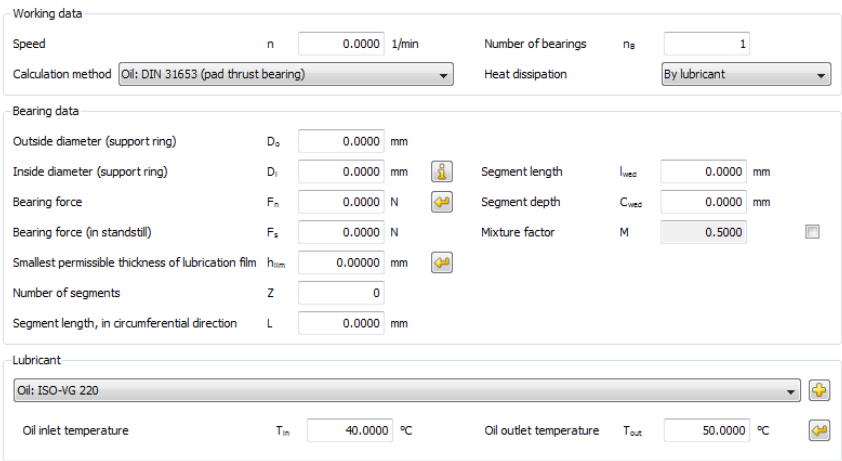


Figure 29.1: Basic data: Plain thrust bearing

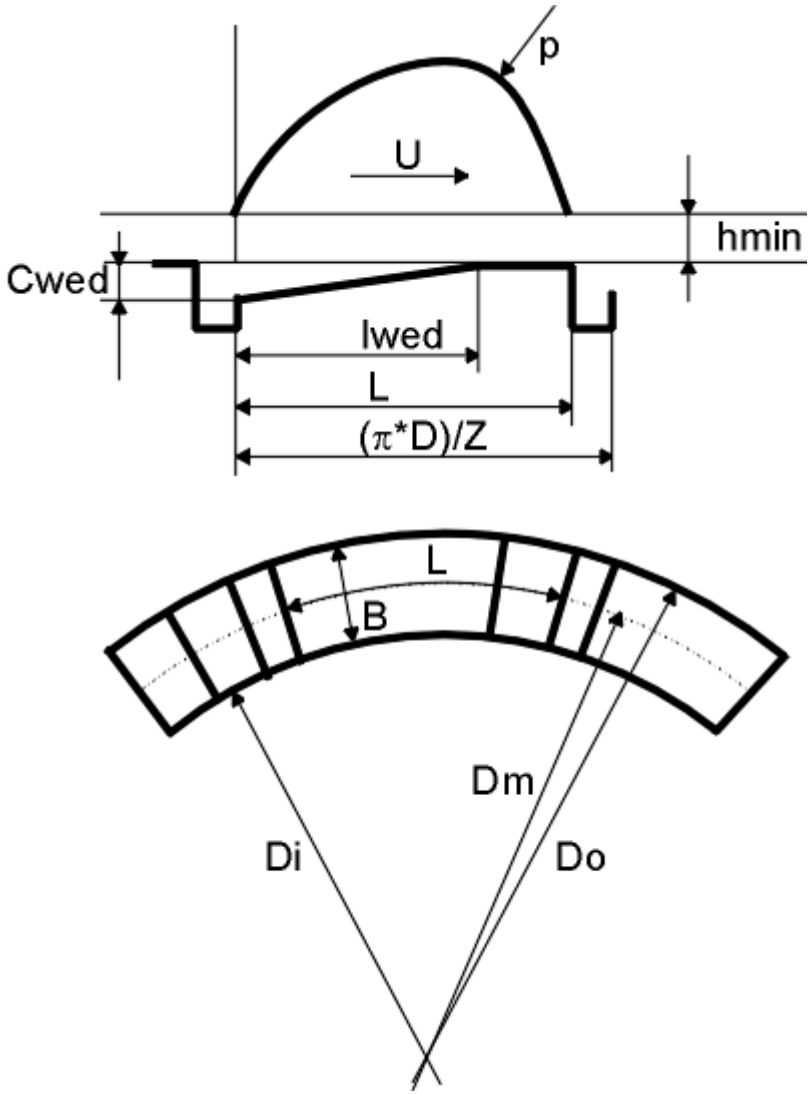


Figure 29.2: Pad thrust bearings as described in DIN 31653

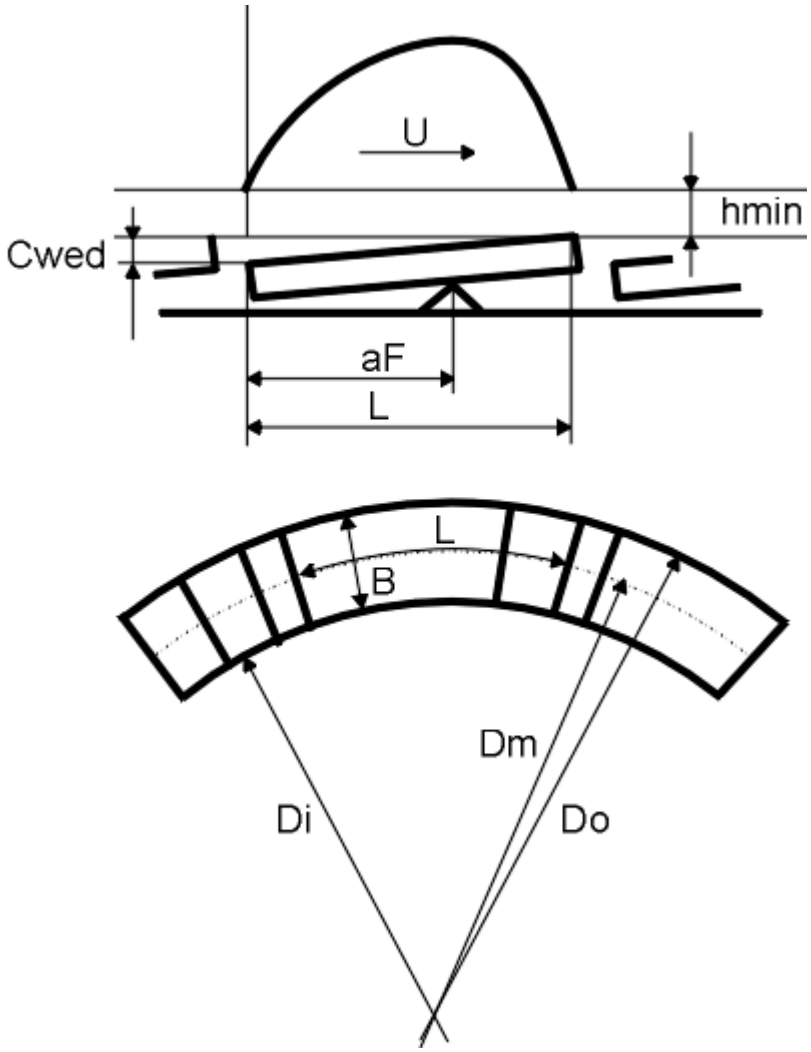


Figure 29.3: Tilting-pad thrust bearings as described in DIN 31654

30.1 Calculation

Essentially, both calculation procedures are based on the equation used to ascertain the thermal balance in the bearing. You can use either convection or circulatory lubrication in this calculation.

- Non pressure lubricated bearings (self-lubricating) dissipate heat out to the surrounding environment by convection. The coefficient of thermal expansion factor kA, in accordance with the standard, lies between 15 . . In the program the default value is 20 W/(m2*K), but you can change this as required.
- Pressure lubricated bearings mainly dissipate heat through the lubricant. Here you need to define a mixture factor M which must lie between . . 1. Experience shows that this usually lies between 0.4 and 0.6. The default setting is 0.5, but this can be changed.

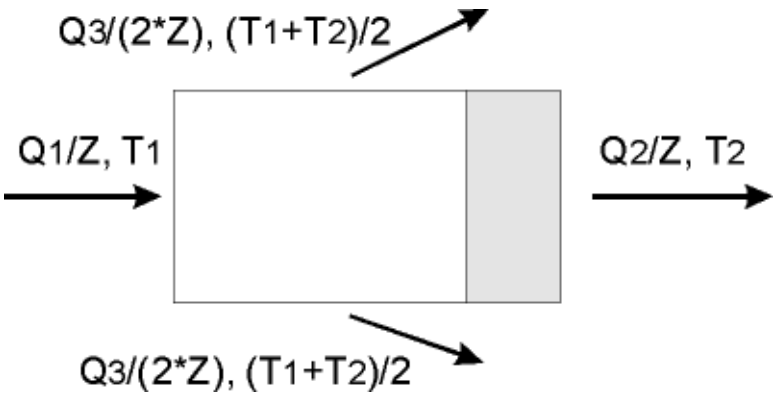


Figure 29.4: Segment lubricant and heat levels

These calculations provide values for the friction power, lowest thickness of lubrication film, and operating temperature. For circulatory lubrication they also calculate the lubricant flow rate.

The bearing force (in standstill) is only used to determine the lowest admissible thickness of lubrication film, and is otherwise irrelevant. The value of the load coefficient, the friction coefficient, and the lubricant flow rates, are calculated in accordance with the formulas stated in DIN 31653/31654 Part 2 (and not in accordance with the diagram or table). For tilting-pad thrust bearings, the ratio hmin/Cwed is calculated from the support position of the tilting-pad aF*. For a tilting-pad thrust bearing, the hmin/Cwed ratio is calculated from the supporting area of the tilting pad aF*. The formula for this is given in DIN 31654 Part 2.

30.2 Sizings

You can also calculate the bearing force (nominal load), but before you can do this, you must enter all the other input values. The bearing force is then calculated using the value you specified for the lowest thickness of lubrication film, hlim.

The lowest thickness of lubrication film hlim can be calculated in accordance with DIN 31653 or 31654. This lowest possible thickness of lubrication film is calculated depending on sliding velocity, average diameter and loading, according to the formula.

For convection:
If you do not know the value for the heat transfer surface, you can use a formula for approximating it, as defined in the standard:
A = (15 to 20) * B * L * Z
Click the button next to the input of the surface value to calculate this value using the formula A = 15 *B *L*Z.

For circulatory lubrication:
Experience shows that the exit temperature is between 10 and 30 K higher than the entry temperature. Click the button next to the Exit temperature to calculate a default value with a 10 K temperature difference.

30.3 Calculation of volume specific heat

In `Calculations/Settings` there are two methods you can use to calculate volume-specific heat:

- `Take into account dependence on temperature`
- `By a simplified assumption (as in DIN 31652):` 1.8 * 10^6J/(m3 * K)

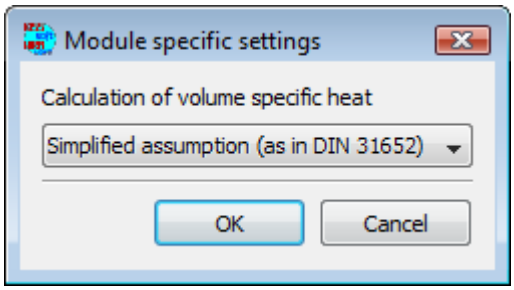


Figure 29.5: Module specific settings

30.4 Limiting values in the calculation

The standards only apply to laminar flow in the lubrication gap. For this to happen, the Reynold number must lie below the critical value of 600.
These results are also checked for highest permissible bearing temperature, Tlim, the lowest possible thickness of lubrication film, hlim, and the specific bearing load. These limiting values are defined in Part 3 of the 31653/ 31654 standard.

Chapter 30
Answers to Frequently Asked Questions

31.1 Intersecting notch effects

If at all possible, notch effects - for example in a shoulder with an interference fit - should not be overlapped when the shaft is designed. However, if this does happen, in the worst case scenario, the FKM Guideline should be applied to calculate the overall notch effect coefficient K_f:

$$K_f = 1 + (K_{f1} - 1) + (K_{f2} - 1)$$

from part notch effect coefficients K_{f1} and K_{f2}. In KISSsoft, this situation can be resolved by selecting `Own Input` for the Notch effect (see page III-712) of a free cross section (see page III-672).

The overall notch effect coefficient can then be calculated as follows:

1. Two cross sections (for example, A-A and B-B) are defined with the same Y-coordinate.
2. Cross section A-A is calculated by selecting notch type (for example, shoulder) Kf1. The notch factors are displayed directly in the Elements editor (see page III-649).
3. This procedure described in 2. is then repeated for cross section B-B.
4. The resulting notch factors for both these notches are noted down and the notch factors Kf are calculated in accordance with the formula given above.
5. Now both cross sections (A-A and B-B) are deleted and a new free cross section C-C with the same Y-coordinate is added. In the Elements editor, now select `Own Input` for the notch effect and the overall notch effect coefficients calculated in 4. are displayed.

31.2 Notch effects on hollow shafts

All the notch factors described in the standards have been determined for solid shafts. No data is available for hollow shafts. KISSsoft calculates the nominal stresses for hollow shafts using the section modulus and taking into account the inside diameter.

31.2.1 Notches on the outer contour

For "small" inside diameters, the error due to calculating notch effect values for solid shafts is relatively small and you can use the results as approximations. However, when "large" inside diameters are involved, you must correct the notch effect values.

According to the FKM Guideline of 1998 you cannot accurately calculate the notch effect values of a round shaft that has a longitudinal bore for bending and tension/compression using the notch effect values of a round solid shaft. You should use the notch effect value of a round solid shaft for torsion and round shafts that have a circumferential notch, shoulder. or conus, but use this value with nominal stress for a round shaft that has a longitudinal bore.

31.2.2 Notches on the inner contour

You cannot use these calculation methods to determine the notch factors of notches on the inner contour.

31.3 Fatigue Limits for New Materials

If you want to add a new material to the database, you must enter its endurance limits as well as the yield point and tensile strength.

Hänchen gives

$$\sigma_{bW} \approx 0.495 \cdot \sigma_B$$

an approximation of the fatigue limit for bending as well as other approximations from different sources. For the tension/compression fatigue limit, this states

$$\sigma_{zdW} \approx 0.45 \cdot \sigma_B \quad oder \quad \sigma_{zdW} = 0.5 \ldots 0.6 \cdot \sigma_B$$

, and for the torsion fatigue limit it states

$$\tau_{tW} \approx 0.55 \cdot \sigma_{bW} \quad oder \quad \tau_{tW} = 0.4 \ldots 0.5 \cdot \sigma_{bW}$$

.

According to DIN 743 the following approximations can be made:

$$\sigma_{bW} \approx 0.5 \cdot \sigma_B \quad \sigma_{zdW} \approx 0.4 \cdot \sigma_B \quad \tau_{tW} \approx 0.3 \cdot \sigma_B$$

For heat treatable steels the FKM guideline proposes the following (but for other material types different values apply):

$$\sigma_{W,zd} \approx 0.45 \cdot R_m \quad \tau_{W,s} \approx 0.26 \cdot R_m$$

31.4 Taking double helical gearing into account in the shaft calculation

In the shaft analysis process, when you input cylindrical gear data in "`Hand of gear`" you can select continuous double helical teeth from the `drop-down list`. A gear with this characteristic always has an axial force 0 N. When double helical gears are transferred from the gear calculation (checkbox `Read data from file` active) the total width (= left side + gap + right side) is also transferred as is the total power. The shaft analysis then takes both the gap and the effective toothing into account. This generally results in a very useful model.

If you require a more precise model, input the two halves of the gear individually, one inclined to the right and the other inclined to the left. Unfortunately, you cannot do this by transferring the data directly from the gear calculation.

Part IV

Connections

Chapter 31
Cylindrical interference fit

- The calculation includes the entirety of the DIN 7190 standard (elastics) with longitudinal, radial and oil interference fits.
- Load in circumferential and axial directions.
- Load with bending moment and radial force.
- Calculating the maximum torque for a non-slipping fit. If slip occurs in the fit, micro gliding will cause corrosion due to friction.
- Influence of centrifugal force.
- Verification of an elastic-plastic loaded interference fit as specified in DIN 7190 with predefined interference (stresses and elongations are calculated only for the elastic case)
- Analysis of hubs with multiple interference fits
- Display stress curves (equivalent, tangential and radial stresses)
- Display tolerance fields:
  - only take into account allowances
  - take into account temperature and centrifugal force (without pressure)
  - take into account temperature, centrifugal force and pressure

You can calculate the safety of the interference fit against sliding and the safety of the shaft material and the hub against fracture and yielding. The calculation also takes into account the effect of centrifugal force on the expansion of the interference fit and on the stress in the shaft and hub. The tolerance system specified in DIN 7151 (e.g. with diameter input 60 H7/f6) has been implemented to make it easier to input data. You can either enter the tolerance manually, or use an automatic option to calculate the tolerance pairing based on the required safety against sliding and the permitted material stress. Input values for surface roughness with qualities as defined in ISO 1302.

Calculating the pressure: For elastic materials according to the theory of mechanics for thick cylinders under internal pressure and thick cylinders under external pressure (e.g. [60], page 399, or [64]).

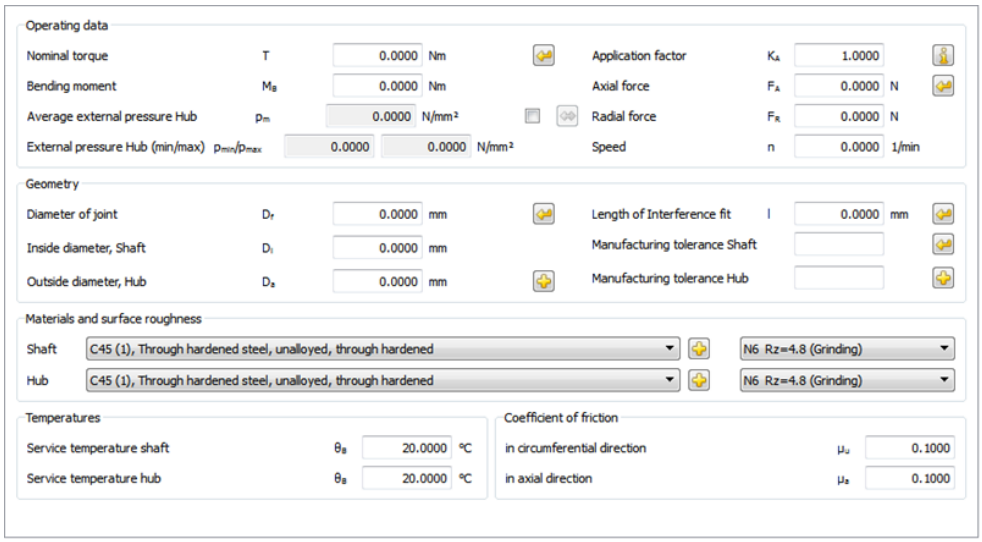


Figure 31.1: Basic data: Cylindrical interference fit

Influence of speed: according to the theory of a cylinder in rotation ([38], page 219)

Amount of embedding: as defined in DIN 7190.

Equivalent stress:
you can change the hypothesis of equivalent stress under Calculations/Settings. The Settings chapter provides more information about this function.

Bending moment and radial force: this takes into account the effect of a bending moment and a radial force on the pressure. The additional amount of pressure is calculated as follows:

$$p_b = \frac{9}{2}\frac{M_b}{d_F l_f^{\,2}} \qquad p_r = \frac{F_r}{d_F l_F} \tag{31.1}$$

To ensure no gaping occurs between the hub and the shaft, the additional pressure must be less than the minimum interference pressure on the connection ($(p_b + p_r) < p_{min}$).

Other values:

Dismounting force	Specified in [64], page 363
Transmission without slip	Specified in [55], equations 2.93 to 2.110

Micro sliding: If the torque of an interference fit is increased continuously until it exceeds the micro sliding limit, a local slip will occur at the position to which the torque is applied. As torque decreases continuously in the interference fit, the slip occurs only in one part of the fit, even if the torque then increases again. This effect is called micro gliding (shaft moving back and forth in the hub) and can cause friction rust. Further explanations and details about this calculation are given in "Welle-Nabe-Verbindungen (Shaft-Hub Connections)", by Kollmann [55].
Comment about the calculation as defined by Kollmann:
To limit torque for micro sliding, use equation 2.110, with k use equation 2.107 and with ır use equation 2.93.

Mounting: You will find details about mounting in the report. The temperature difference for mounting is calculated in such a way that, even if the maximum interference is reached (worst case scenario) there will still be enough play in the joint. Define the mounting clearance under `Settings`. Here you calculate the parameters for mounting the shaft at ambient temperature and for a cooled shaft (shaft at approximately -150°C).

Verification of an elastic-plastic loaded interference fit according to DIN 7190:

Prerequisites: EI = EA, nyI = nyA, n = 0, diI = 0

If all the prerequisites are fulfilled, as defined in DIN 7190, the plasticity diameter DPA of the outer part that is to be mounted can be calculated (diameter at which the plastic range ends). The corresponding compacting pressure and the relationship between ring surface qpA and the overall cross-section qA are also calculated. (Experiential limit according to DIN 7190 for heavily loaded interference fits in mechanical engineering qpA/qA <= 0.3)

32.1 Inputting Tolerances

Tolerances in accordance with ISO/DIN:

To enter tolerances in the same way as, for example 60 H7/f6, you must:

- Enter 60 (mm) as the joint diameter.
- Enter H7 and f6 in the `Tolerances` fields. Here, the first field is where you enter the value for the wheel or hub and the second is where you enter the value for the shaft.
- The program checks automatically to see whether the tolerances you specified actually exist and if you entered the data in the correct format!

Define own tolerances

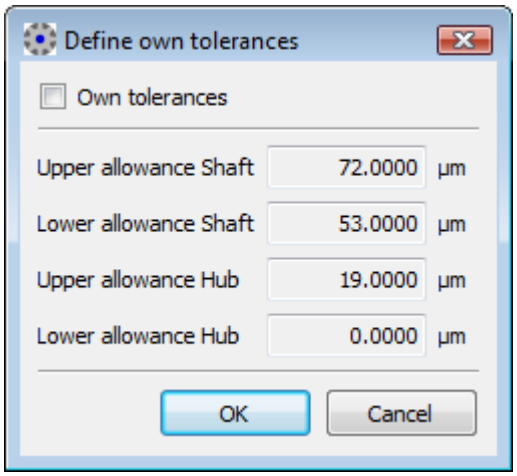


Figure 31.2: Tolerance values

Click the button next to the `Tolerances` field to display the current allowances. You can then change them as required.

32.2 Coefficients of friction

Tables 31.2 and 31.4 contain coefficient of friction values as defined in DIN 7190.

Materials	**Coefficients of friction**			
	dry		lubricated	
	ν_{ll}	ν_{rl}	ν_{ll}	ν_{rl}
E 335	0.11	0.08	0.08	0.07
GE 300	0.11	0.08	0.08	0.07
S 235JRG2	0.10	0.09	0.07	0.06
EN-GJL-250	0.12	0.11	0.06	0.05
EN-GJS-600-3	0.10	0.09	0.06	0.05
EN-AB-44000 and following	0.07	0.06	0.05	0.04
CB495K	0.07	0.06	-	-
TiAl6V4	-	-	0.05	-
ν_{ll}: in axial direction - loosen				
ν_{rl}: lengthwise - sliding				

Table 31.2: Coefficients of friction for longitudinal fits under continuous stress as defined in DIN 7190

Material pairs, Lubrication, Joining	**Coefficients of friction** ν_r, ν_{rl}, ν_u
Steel-steel pairing:	
Oil pressure connection normally joined with mineral oil	0.12
Oil pressure connection with degreased contact surfaces joined with glycerin	0.18
Shrink fit normally after warming the External component up to 300°C in electrical oven	0.14
Shrink fit with degreased contact surfaces after heating in electrical ovens up to 300°C	0.20
Steel-cast iron pairing:	
Oil pressure connection normally joined with mineral oil	0.10
Oil pressure connection with degreased contact surfaces	0.16
Steel-MgAl pairing, dry	0.10 to 0.15
Steel-CuZn pairing, dry	0.17 to 0.25

ν_r: Slipping ν_{rl}: lengthwise - sliding ν_u: in peripheral direction	

Table 31.4: Coefficients of friction for radial interference fits in longitudinal and peripheral direction subjected to sliding as defined in DIN 7190

32.3 Variable hub external diameter

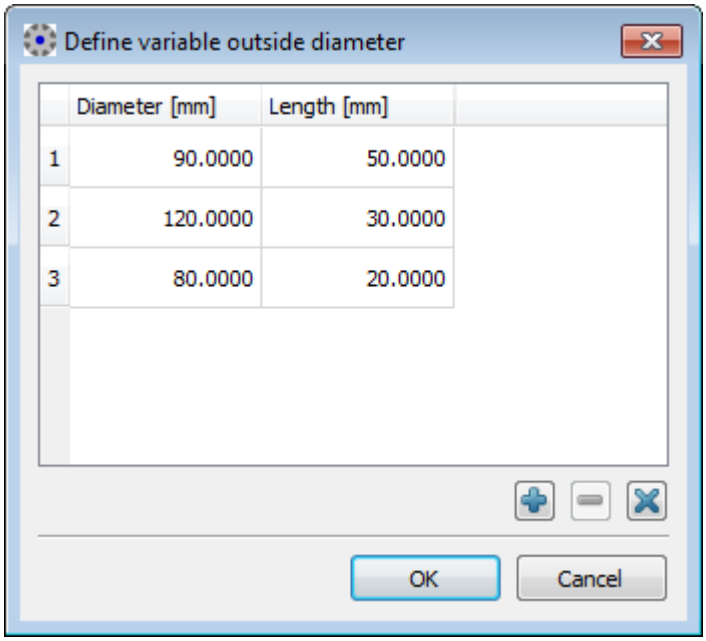
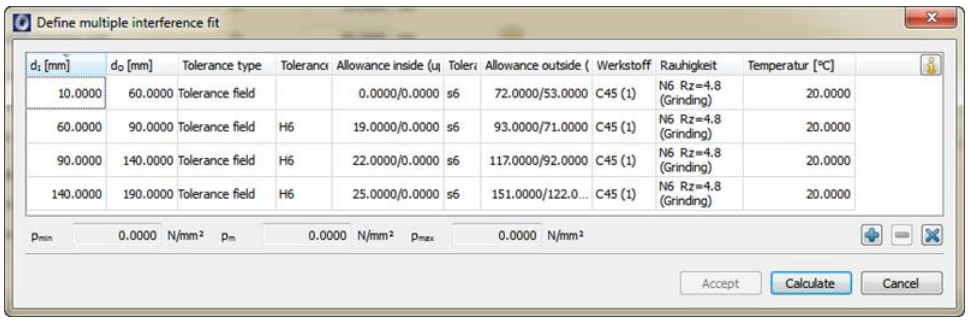


Figure 31.3: Variable external diameter

For a stepped hub out diameter, a single equivalent diameter and length are determined. These values are then used to calculate the stiffness of the external component.

32.4 Convert external pressure with multiple interference fit

The effect of outside pressure on the first hub can be taken into account for a multiple interference fit. This pressure is defined through a series of sequential interference fits, as shown in the following figure, and is calculated by the software (no direct user input).



d_i [mm]	d_o [mm]	Tolerance type	Tolerance	Allowance inside (µ	Toler	Allowance outside (	Werkstoff	Rauhigkeit	Temperatur [°C]
10.0000	60.0000	Tolerance field		0.0000/0.0000	s6	72.0000/53.0000	C45 (1)	N6 Rz=4.8 (Grinding)	20.0000
60.0000	90.0000	Tolerance field	H6	19.0000/0.0000	s6	93.0000/71.0000	C45 (1)	N6 Rz=4.8 (Grinding)	20.0000
90.0000	140.0000	Tolerance field	H6	22.0000/0.0000	s6	117.0000/92.0000	C45 (1)	N6 Rz=4.8 (Grinding)	20.0000
140.0000	190.0000	Tolerance field	H6	25.0000/0.0000	s6	151.0000/122.0...	C45 (1)	N6 Rz=4.8 (Grinding)	20.0000

p_{min} 0.0000 N/mm² p_m 0.0000 N/mm² p_{max} 0.0000 N/mm²

Figure31.3 Multiple interference fit

The following notes apply to this calculation:

- The first two elements are the original shaft and hub (the first two lines in the table correspond to the first interference fit on the shaft), and are just shown in the table for the sake of completeness. The only user-editable values are the external diameter tolerances of the hub.
- The external diameter of each ring also serves as the inside diameter of the next ring.
- When a new ring is added the following default values are used:
  - External diameter = inside diameter + 50 mm
  - The Material and Roughness are those of the original hub
  - The operating temperature is the same as that for the hub
  - The tolerance class of the inner diameter is the same as the tolerance class of the shaft (for example, s6)
  - The tolerance class of the external diameter is the same as the tolerance class of the hub (for example, H7)

The pressure on the hub is calculated for the following three cases of tolerance values of all elements:

- The average tolerance at each contact surface
- The worst case, i.e. the maximum oversize at each contact surface
- The best case, i.e. the minimum oversize at each contact surface

32.5 Materials

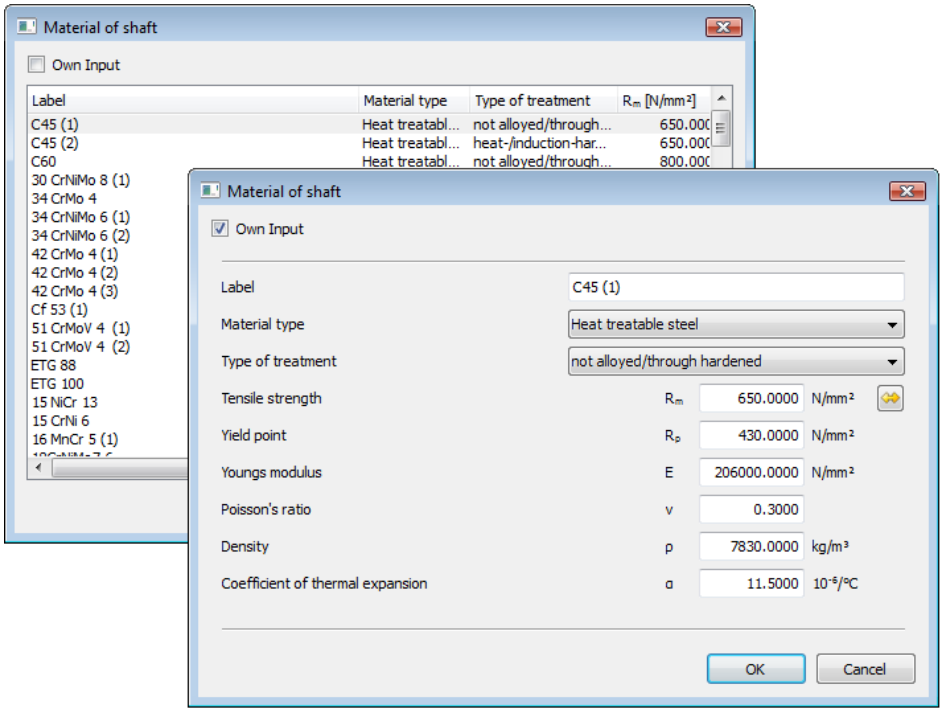


Figure 31.4: Materials screen: Cylindrical interference fit

In the selection list, you can select materials in accordance with the standard.
If you have set the `Own Input` flag, a new dialog appears here. This displays the material data used in the calculation which you can specify to suit your own purposes. You can also define your own materials directly in the database (see page I-114) so that these can also be used in future calculations.

32.6 Settings

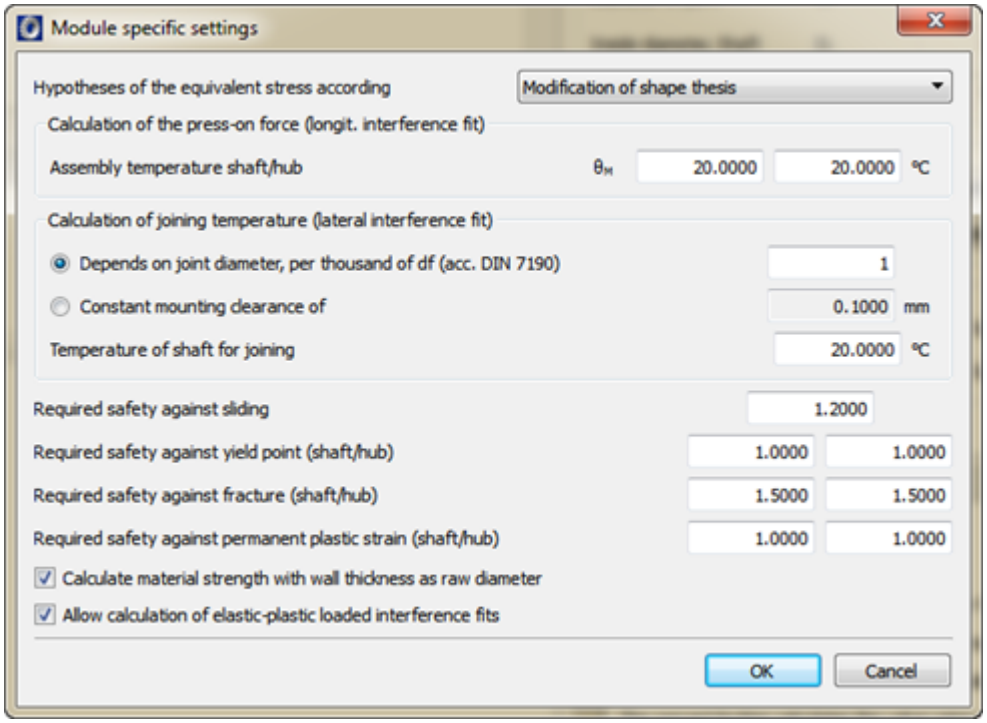


Figure 31.5: Settings: Cylindrical interference fit

- Select equivalent stress
  In accordance with [64], with the modification of shape hypothesis:
  $\sigma_v = \max(|\sigma_{\phi\phi} - \sigma_{rr}|, |\sigma_{\phi\phi}|, |\sigma_{rr}|)$
- In accordance with [55], page 13, with the shearing stress hypothesis:
  $$\sigma_v = \sqrt{\sigma_1^2 + \sigma_2^2 - \sigma_1 \cdot \sigma_2}$$
- `Calculation of joining temperature`
  The mounting clearance can be input either dependent on the diameter of joint D_F (modified for warming) or as a constant mounting clearance. The joining temperature of the external component is then calculated from this value.
  You can also define the **shaft temperature for joining**. This temperature and the mounting clearance are then used to calculate the joining temperature of the hub. The joining temperature of the hub is only output in the report if the shaft temperature during joining lies between -273 °C and 20 °C.
- `Calculate material strength with wall thickness as raw diameter`

If you set this flag, the strength of the hub material is calculated using the wall thickness instead of the raw diameter.

- `Allow calculation of elastic-plastic loaded interference fits`
  If you set this flag, the calculation is also performed for elastic-plastic stress (in accordance with DIN 7190), otherwise, only elastic stress is included.

- `Required safeties`
  Under Settings you can input the required safeties against sliding, yield point and fracture. These required safety factors are then used to define the values you require during sizing. The required safeties against plastic deformation are used to define the plasticity diameter that must be set when an interference fit is placed under plastic-elastic stress.

32.7 Sizings

- Tolerance in accordance with ISO/DIN

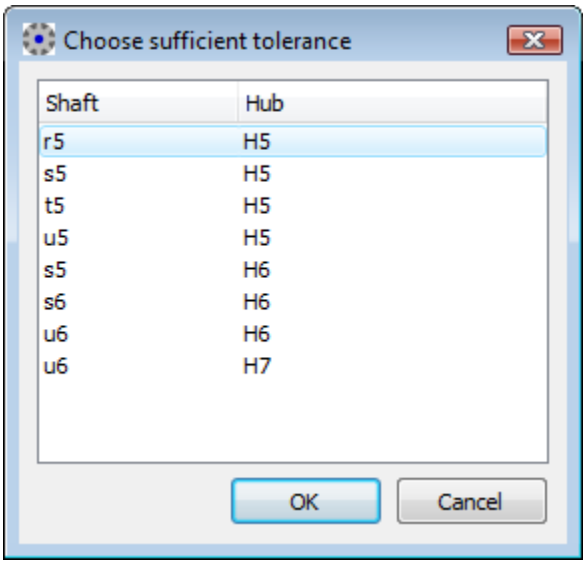


Figure 31.6: Display of possible tolerance pairings

KISSsoft has a very convenient sizing function that you can use for suitable tolerance pairs. Standardized tolerance pairs are stored in the M01-001.DAT file. Click the button next to the Tolerances field in the main screen to start the sizing process.

Based on the nominal required safety (which you can change in `Settings`), you can determine all the tolerance pairs which fulfill the requirements (sufficient safety against sliding, safety against fracture and yield point) and display these pairs in a list.

- Torque, axial force, diameter of joint, and length of interference fit
  KISSsoft can size the maximum transmissible torque, the transmissible axial force, the required length, and the diameter (according the required safety values you entered in Settings).

Chapter 32
Conical interference fit

Calculating the service reliability of a conical interference fit. Defining the mounting conditions.

The calculation methods are:

- Method as defined by Kollmann [55], Verification and sizing.

- Method as defined in E DIN 7190-2 (2013)

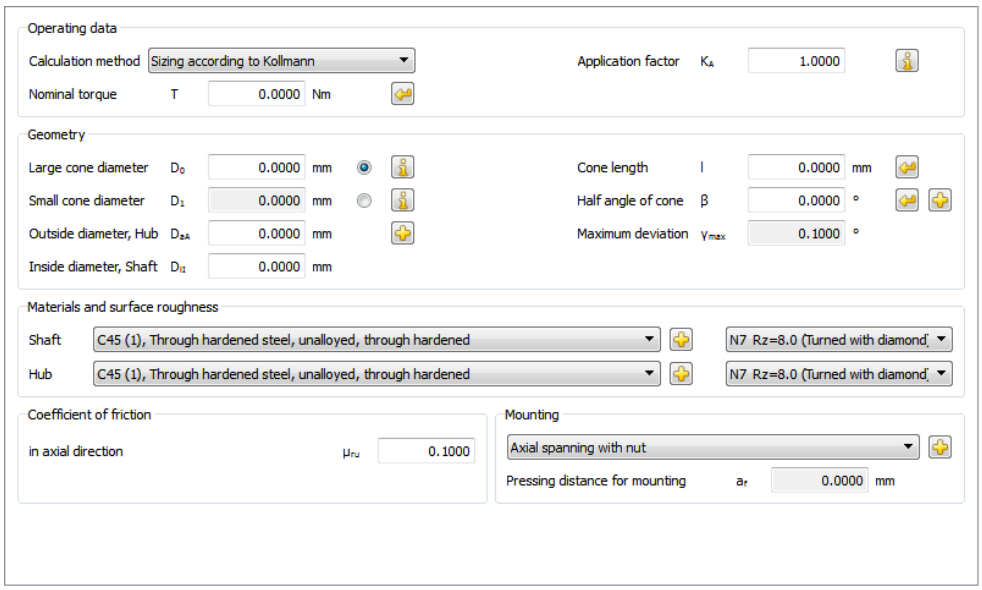


Figure 32.1: Basic data: Conical interference fit

Cone angle: The cone angle is the angle β between the flank of the cone and its axis. The opening angle α of the cone is twice the size of the angle of taper.

Calculating the conical interference fit:

- All known investigations focus on external and internal components made of materials that have the same Young's modulus and internal components that do not have any holes in the area of the cone. (Kollmann)
- If you select the method described in E DIN 7190-2, and the shaft and hub are made of different materials, the material with the higher Young's modulus should be used for the shaft material.

- Conical interference fits must always have a stop at the upper end. For this reason, the program only deals with this situation.
- Conical interference fits are normally spanned (tightened) axially with a bolt. You must check the joint carefully by measuring the displacement of the cone. Just tightening with a torque wrench is not accurate enough. Conical interference fits are only joined by pressing them on in exceptional circumstances.

Adhesive coefficient in the case of slipping in the axial direction: Coefficients of friction (for slipping in axial direction) after investigations by Galle (see Kollmann [55], Table 2.20):

Material pairing	**Previous Rating**	**Coefficient for sticktion**
Ck60/16MnCr5	-	0.299
42CrMo4/16MnCr5	-	0.269
31CrMoV9/31CrMoV9	-	0.247
Ck60/16MnCr5	U	0.407
42CrMo4/16MnCr5	U	0.297
31CrMoV9/16MnCr5	U	0.375
31CrMoV9/31CrMoV9	U	0.468
Ck60/16MnCr5	W	0.357
42CrMo4/16MnCr5	W	0.472
31CrMoV9/31CrMoV9	W	0.387

Load classifications:

- none

U Circumferential bending load

W Fatigue torsion load

No adhesive coefficients for other combinations of materials are available, so you will have to estimate them.

Adhesive coefficients for radial interference fits in longitudinal and peripheral direction subjected to sliding: Coefficients of friction as described in DIN 7190 [29], Table 4, see Cylindrical interference fit (see page IV-796) section.

33.1 Calculation

Sizing in accordance with Kollmann:

If you select this method, the maximum angular deviation γ_{max}, the pressing distance for mounting a_f, and the joining force, can be sized using the required safeties and the specified load and geometry.

Verification in accordance with Kollmann:

The transmittable torque is also calculated if no angular deviation, or maximum angular deviation is present.

The safeties against the yield point and against sliding are also calculated in the verification process.

The safeties defined by Kollmann include the required safeties, which can be defined under `Calculation/Settings`.

As specified in E DIN 7190-2:

In this calculation method, the sliding moment, joining force, and axial force are calculated for the state at which all the elements are already joined, from the input torque, mounting without force and all the geometry data. The safety against sliding is calculated in this way. If different external hub diameters or different inner shaft diameters are present these can also be defined as slices (max. 7). The values are calculated for each individual slice and then added together. The maximum value is used for the pressure in the joint.

$$S_R = T_R / T_{\max}$$

In the case of a central load application, the safety against sliding S_R should have a value of $S_{Rmin} > 1.3$.

Furthermore, if you have defined your own joining force at mounting p_{FA} to calculate the effective equivalent stresses, the maximum equivalent stress is then compared with the yield point of that particular material in order to define the safety. The maximum equivalent stresses appear every time for the inner diameter of the hub or shaft when a hollow shaft is being used.

$$S_P = R_{p0,2} / \sigma_{v\max}$$

Safety against the yield point S_P (plastic flow) should have a value of $S_{Pmin} > 1.3$ to ensure that the elements are not affected by plastic deformation when they are assembled and disassembled.

33.2 Application factor

You define the application factor here in the same way as in the cylindrical gear calculation:

Operational behavior of the driving Machine	**Operational behavior of the driven machine**			
	equal-moderate	moderate shocks	medium shocks	strong shocks
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 32.2 Application factor in calculations in accordance with DIN 6892. You will find more detailed comments in DIN 3990, DIN 3991, ISO 6336.

33.3 Axial spanning with nut

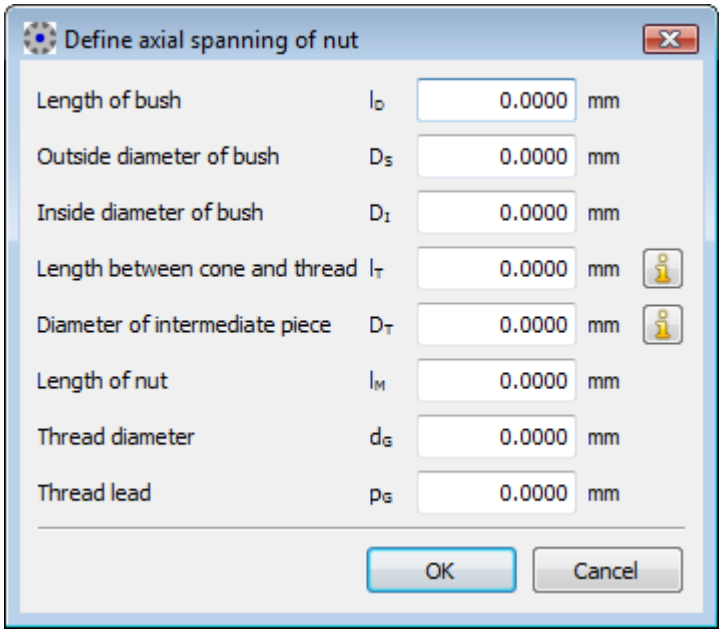
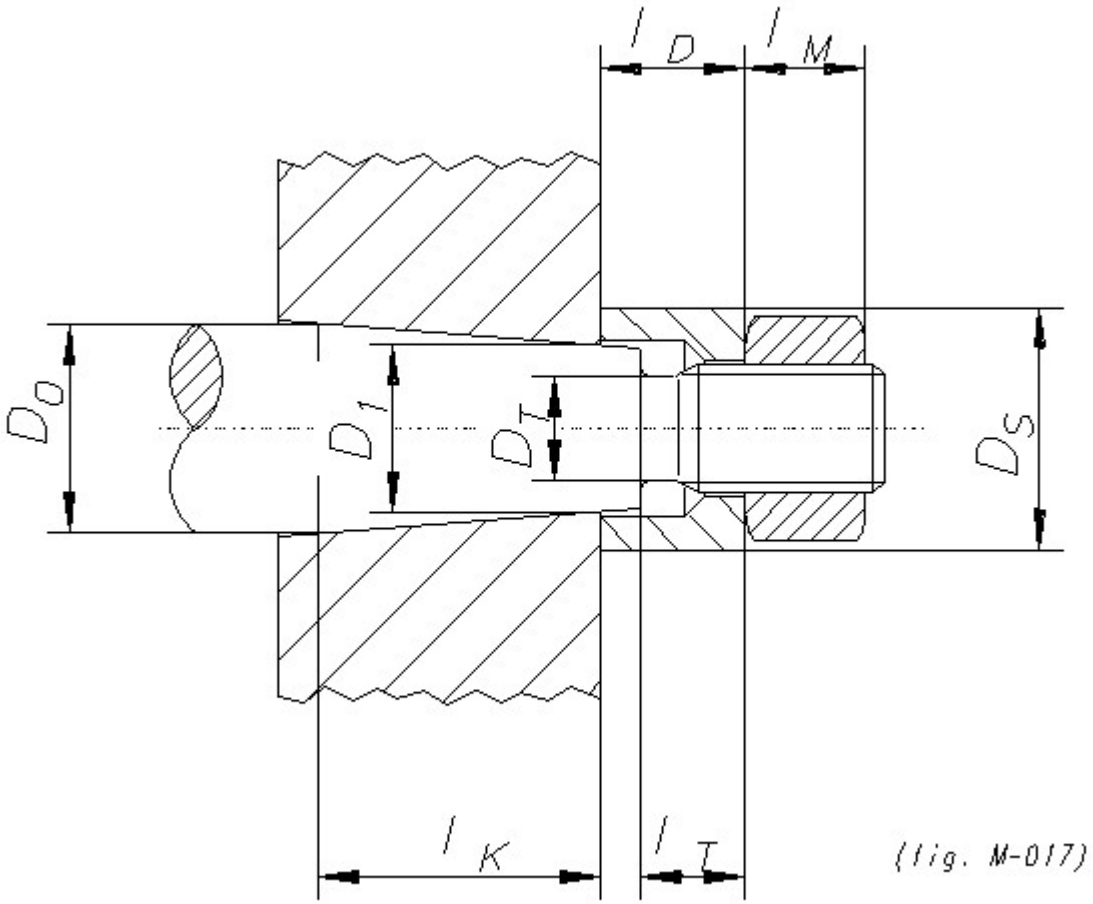


Figure 32.2: Axial spanning of nut

Axial spanning (tightening the nut) produces relative axial displacements which are applied to the individual parts. This causes lateral elongation and therefore increases the compacting pressure on the active surface. The values required for this calculation are shown in the diagram below.



33.4 Variable external diameter of the hub

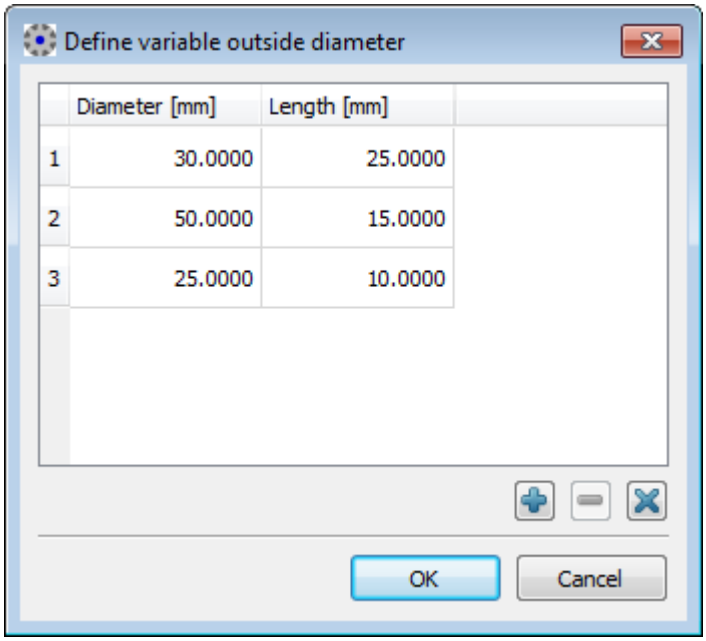


Figure 32.3: Variable external diameter

In the case of a stepped outer diameter, a single equivalent diameter is determined from the diameters and lengths. This value is then used to calculate the stiffness of the outer part.

33.5 Conicity

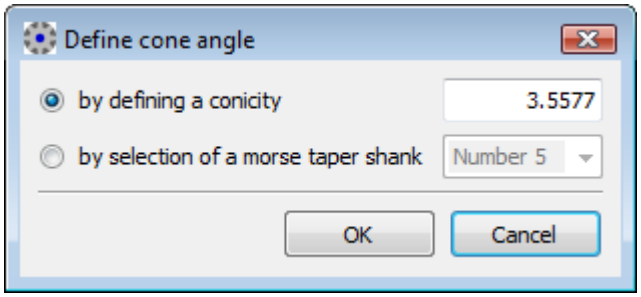


Figure 32.5: Conicity

This additional input dialog gives two methods for defining the conus:

- Conicity:
  conicity is defined as follows:
  $x = l/(D_0-D_1)$. Here, x is the value that must be input.
- Morse tapers:
  Morse tapers are defined in DIN 228 and have a conicity of between 1:19.212 and 1:20.02.

33.6 Materials

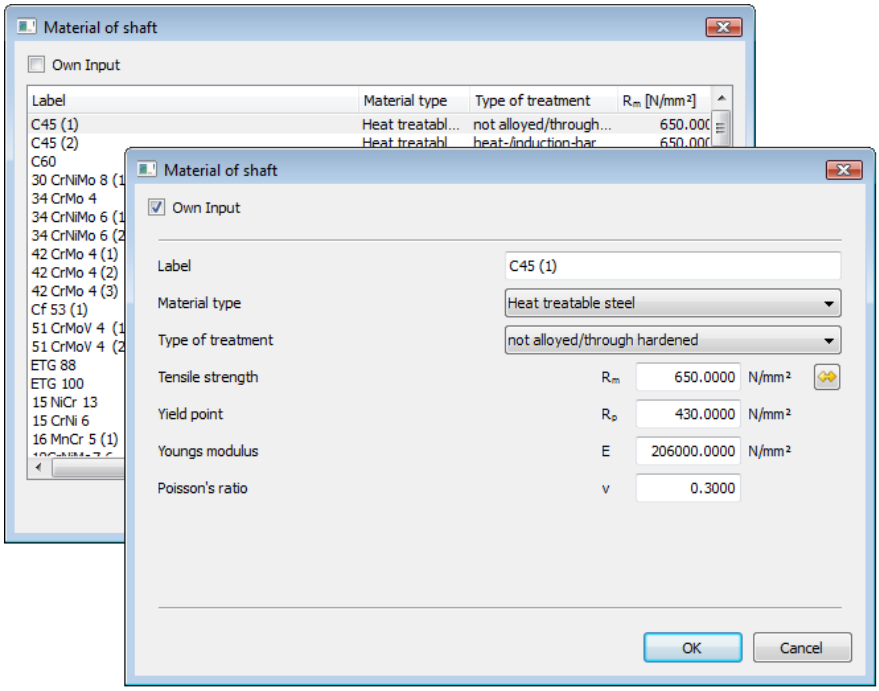


Figure 32.6: Materials screen: Conical interference fit

In the selection list, you can select materials in accordance with the standard.
If you have set the `Own Input` flag, a new dialog appears here. This displays the material data used in the calculation which you can specify to suit your own purposes. You can also define your own materials directly in the database (see page I-114) so that these can also be used in future calculations.

33.7 Settings

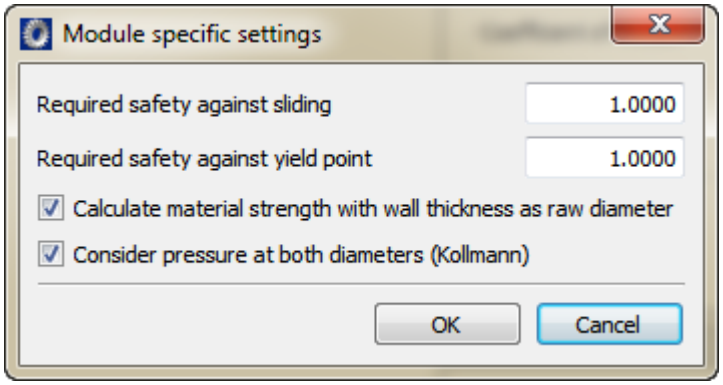


Figure 32.7: Settings: Conical interference fit

If you selected `Calculate material strength with wall thickness as raw diameter`, the strength of the hub material is calculated using the wall thickness instead of the raw diameter.

If you select the `Consider pressure at both diameters (Kollmann)` flag, the pressure at both the large and small cone diameters is taken into account, otherwise only the pressure at the largest diameter is used. However, this only applies to the method described by Kollmann. In E DIN 7190-2 the average diameter is used for the calculation.

Enter the required safety factor for slipping and yield point under Settings. These safety factors are then used to define the values you require during sizing.

In the method defined by Kollmann, the default value set for required safety against sliding is 1.0 and the default value for required safety against the yield point is set to 2.0.

DIN 7190-2 recommends that safety against the yield point should be set to 1.2 to ensure that the elements are not affected by plastic flow when they are disassembled hydraulically.

In the case of a central load application, E DIN 7190-2 recommends a safety against sliding of at least 1.2.

33.8 Sizings

KISSsoft can calculate the maximum transmissible torque, the permitted cone angle (for self-locking), and the length of interference fit, for transmitting the maximum torque.

The torque and the length of interference fit are sized using the defined required safeties.

As specified in E DIN 7190-2 the sizings are calculated using the required safety against sliding, apart from the joining pressure, which is sized using the required safety against yield point.

Kapitel 33
Clamped connections

Clamped connections are only used to transfer low or medium torque (little fluctuation).

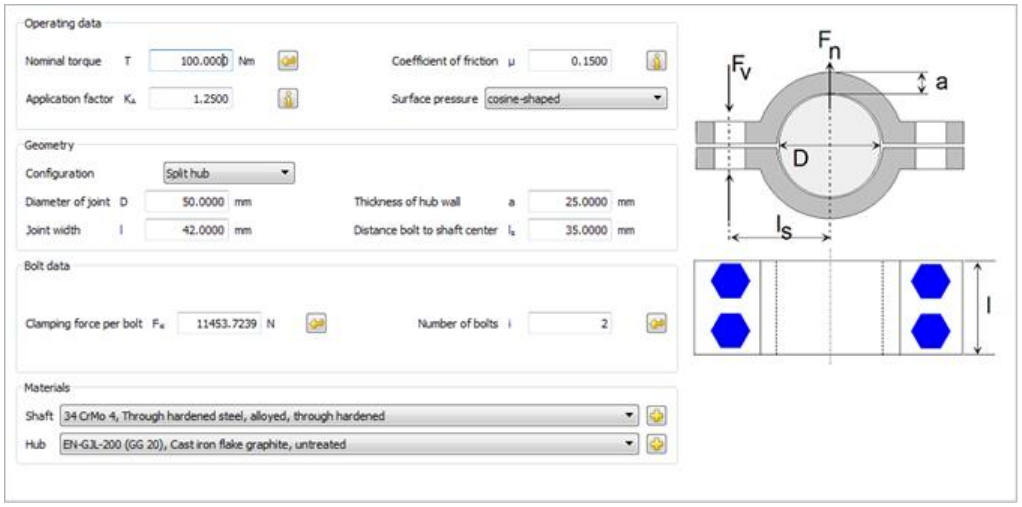


Figure 33.1: Basic data: Clamped connections

There are two different configurations of clamped connections that can be calculated:

- `Split hub`
  In the case of a split hub, it is assumed that pressure is distributed uniformly across the whole joint. The pressure can be equal or cosine-shaped surface pressure or linear contact.
- `Slotted hub`
  We recommend you use as narrow a fit as possible (hubs are also subject to bending) to ensure that the pressure is mostly of a linear nature. The calculation is performed for the least practical case of linear pressure.

Calculations of safety against sliding and surface pressure are described in literature by Roloff Matek [62]. The calculation of bending is performed as specified by Decker [86].

34.1 Calculations

Split hub:

Depending on the type of surface pressure, an additional factor for surface pressure and safety against sliding is used to calculate a shared hub:

- K = 1; uniform surface pressure
- K= π^2/8; cosine-shaped surface pressure
- K = π/2 linear contact

In KISSsoft you can select the type you require from a selection list.

Formula for surface pressure:

$$pF = \frac{2 \cdot KA \cdot T \cdot SH \cdot K}{\pi \cdot D^2 \cdot ls \cdot \mu}$$

Formula for safety against sliding:

$$SH = \frac{Fkl \cdot i \cdot \pi \cdot D \cdot \mu}{2 \cdot KA \cdot T \cdot K}$$

Formula to calculate bending:

$$\sigma B = \frac{0{,}2 \cdot Fkl \cdot i \cdot ls}{2 \cdot Wb}$$

Slotted hub:

Formula for surface pressure:

$$pF = \frac{i \cdot Fkl \cdot l2}{D \cdot l \cdot l1}$$

Formula for safety against sliding:

$$SH = \frac{Fkl \cdot i \cdot D \cdot \mu \cdot l2}{KA \cdot T \cdot l1}$$

Formula to calculate bending:

$$\sigma B = \frac{0{,}3 \cdot Fkl \cdot i \cdot (l2 - l1)}{Wb}$$

Description of codes:

pF: Surface pressure [N/mm2]

KA: Application factor

T: Nominal torque [N]

SH: Safety against sliding

K: Correction factor surface pressure

l: Joint width [mm]

D: Joint diameter [mm]

lS: Distance bolt to shaft center [mm]

l1: Distance normal force to center of rotation [mm]

l2: Distance from clamp force to center of rotation [mm]

μ Coefficient of friction

σB: Bending stress [N/mm2]

Fkl: Clamp force per bolt [N]

i: Number of bolts

Wb: Moment of resistance [mm3]

34.2 Sizings

In these calculations you can size the torque, the clamp force per bolt, and the number of bolts, to suit a predefined required safety value.

34.3 Settings

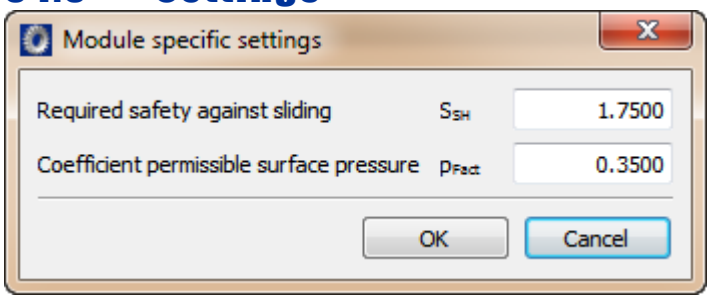


Figure 33.2: Settings Clamped connections

The required safety against sticking SSH is used to size the torque, the clamp force per bolt, and the number of bolts.

If the hub material is gray cast iron, this coefficient times the tensile strength is used to calculate the permitted pressure.

(pzul =pFact*Rm) (default value ~ 0.35 for an interference fit)

For all other materials, this coefficient times the yield point is used to calculate the permitted pressure.

(pzul =pFact*Rp) (default value ~ 0.35 for an interference fit)

34.4 Materials

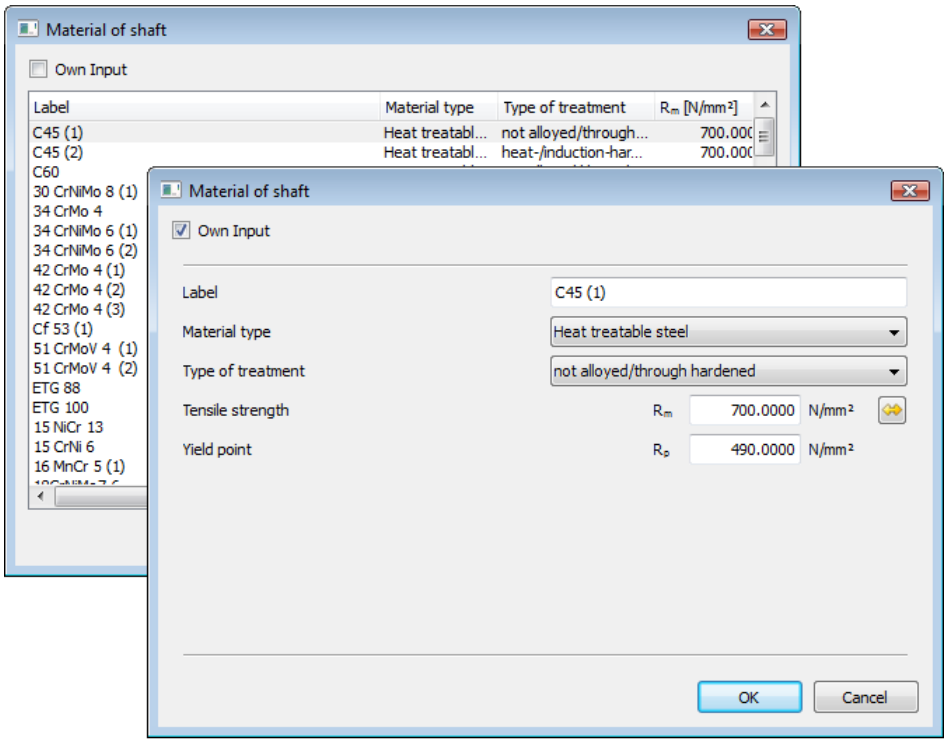


Figure 33.3: Materials Clamped connections

In the selection list, you can select materials in accordance with the standard.
If you have set the `Own Input` flag, a new dialog appears here. This displays the material data used in the calculation which you can define to suit your own purposes. You can also define your own materials directly in the database (see page I-114) so that these can also be used in future calculations.

Chapter 34
Key

Keys are by far the most commonly used shaft-hub connections. In particular, they help to transmit the torque. Their geometry has long been standardized in accordance with DIN 6885 [26]. However, to ensure adequate safety levels are achieved when transmitting torque, you always had to refer to secondary sources of technical literature [64]. The DIN standard 6892 [27] documents the different calculation methods that can be used for key connections.

You must perform two checks for keys:

1. Check the torque transmission by monitoring the surface pressure on the shaft, hub, and key.
2. Check the fatigue limit of the shaft due to the notch effect caused by the keyway. This effect is already described in DIN 743 [9]. We recommend you use this standard to verify the shaft strength rather than DIN 6892.

Special features of calculations in accordance with DIN 6892:

- Key connections are mostly used with light interference fits. The calculation takes into account the decrease in torque on the key due to the interference fit.
- The calculation proves the nominal torque as well as the actual pitch torque over the entire operating period. The fatigue strength calculation based on the nominal torque also includes the number of load changes, which experience has shown to have a significant and damaging effect on the key.
- The type of load has a considerable effect on the service reliability of keys. This effect is taken into account by using a wide range of load distribution coefficients.
- The permissible pressure values are derived from the yield point. As a result, you can derive this for common and more unusual materials in accordance with the standard. The hardness influence coefficient is used to take the surface treatment into account.

Calculation Method B as defined in DIN 6892 recommends you use a differentiated calculation to prove the service reliability of key connections. Method C has been greatly simplified.

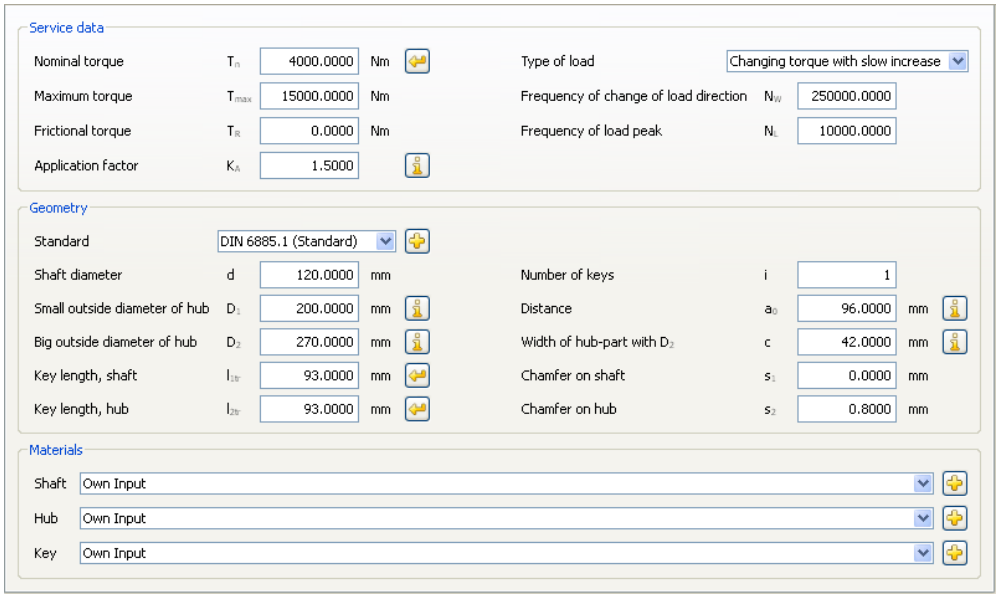


Figure 34.1: Basic data: Key connection

35.1 Main screen

For keys as defined in

- DIN 6885.1 (ISO/R 773, VSM 15161) Standard
- DIN 6885.1 Form G, H, J
- DIN 6885.2
- DIN 6885.3
- ANSI B17.1 Square
- ANSI B17.1 Rectangular
- Own Input

you can calculate the load on shaft, hub and key (surface pressure) and the key (shearing) to determine the safeties.

The following calculation methods are available: DIN 6892 B/C [27].

The calculation takes into account the tolerances of the key radii and the direction of force. You can also enter your own value for the number of keys and the application factor.

Explanations for Figure 34.2:

→	Application or removal of torque
o	Start of key
F_u	Center of force application point on hub

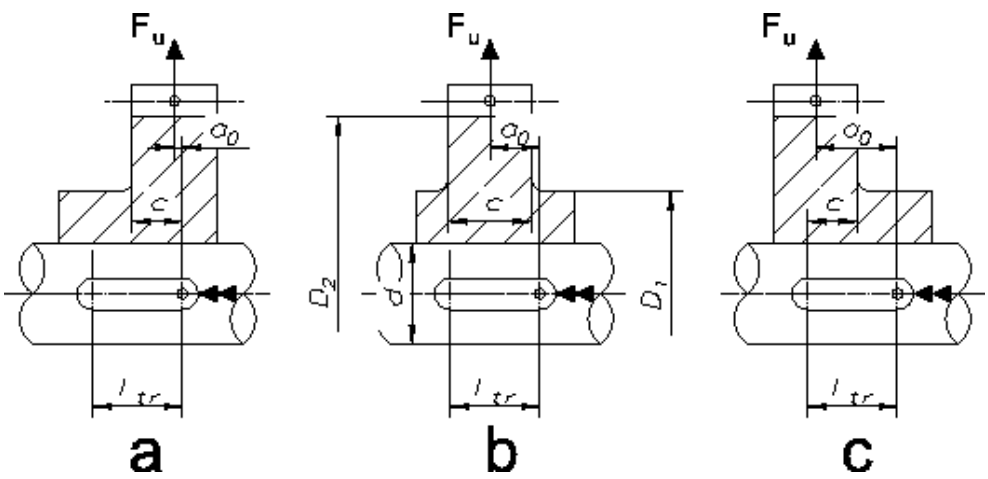


Figure 34.2: Key: Load application.

- Supporting key length
  The supporting key length is defined in accordance with DIN 6892:
  Helical key form (A, E, C in accordance with DIN 6885) $l_{tr} = l_{eff} - b$
  Straight key form (B, D, F, G, H, J in accordance with DIN 6885)$l_{tr} = l_{eff}$

l_{eff}	actual key length
l_{tr}	supporting key length
b	key width

- Frictional torque
  Key connections are usually combined with a light interference fit. The calculation takes into account the decrease in torque on the key due to the interference fit. This effect is only relevant if you are performing the calculation as defined in DIN 6892 B.
- Frequency of load peak
  To determine the safety regarding the maximum torque, you must enter the approximate number of load peaks. This effect is only relevant if you are performing the calculation as defined in DIN 6892 B.

35.1.1 Additional inputs for DIN 6892 Method B

If you select the calculation method specified in with DIN 6892 B, you can enter the following data:

- Chamfer on shaft
- Chamfer on hub
- Small hub external diameter D_1
- Large hub external diameter D_2
- Width c for external diameter D_2
- Distance a_0 (→ see Figure on page IV-821)
- Torque curve: indication of whether this is alternating torque.
  If alternating torque is present, you can also define the backwards torque here.
  If this backwards torque is greater than the minimum effective frictional torque ($T_{maxR} > T_{Rmin}*q$; q=0.8), the load direction changing factor f_w is set to = 1.
  If ($T_{maxR} > T_{Rmin}*q$ and $T_{max} > T_{Rmin}*q$; q=0.8), the maximum torque is therefore also greater than the minimum effective frictional torque. In this case,

the frequency of the changes in load direction is taken into account when defining the load direction changing factor (from diagram; $f_w<1$)

To perform the calculation with the equivalent torsional moment, use factor q=0.5. The formula is the same as the one used for the maximum torsional moment, therefore ($T_{eqR} > T_{Rmin}*q$; q=0.5).

- Frequency of the changes in load direction

  In this situation you input the number of torque changes throughout the entire service life (but only if alternating torque is present).

35.2 Application factor

You define the application factor here in the same way as in the cylindrical gear calculation:

Operational behavior of the driving Machine	**Operational behavior of the driven machine**			
	equal-moderate	moderate shocks	medium shocks	strong shocks
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 34.2: Suggestions for the application factor in calculations in accordance with DIN 6892. You will find more detailed comments in DIN 3990, DIN 3991 and ISO 6336.

Suggestions for the application factor from other sources: See tables 34.4 and 34.6.

Type of machine	**characteristic Operational behavior**	**Type of Shocks**	**working Factor**
Turbines, blowers	uniformly rotating movements	slight	1.0 . . . 1.1
Internal combustion engine	reciprocating movements	medium	1.2 . . . 1.5
Presses, saw frames	reciprocating, impacting movements	big	1.6 . . . 2.0
Hammers, stone crushers	impacting movements	very heavy	2.1 . . . 3.0

Table 34.4: Application factor in accordance with Roloff/Matek [61].

	surfaces pressed together	**surfaces sliding against each other without load**	**surfaces sliding against each other under load**
constant load	1.0	2.0	6.0
pulsating load medium shocks	1.5	3.0	9.0

alternating load medium shocks	3.0	6.0	18.0
pulsating load heavy shocks	2.0	4.0	12.0
alternating load heavy shocks	6.0	8.0	36.0

Table 34.6: Application factor that takes into account the load behavior in accordance with Professor Spinnler [72].

35.3 Load factor

Contact coefficient as defined in DIN 6892, [27]:

$K_v=1/(i*\phi)$

$\phi = 1$ for one spring

$\phi = 0.75$ for two springs to calculate the equivalent surface pressure

$\phi = 0.9$ for two springs to the calculate the maximum surface pressure

more than two springs is unusual

KISSsoft calculates the contact coefficient on the basis of the number of springs.

35.4 Own inputs

In the `Own inputs` option you can enter your own geometry data for keys, that differs from the values given in DIN 6885.

NOTE

If you already know the upper and lower allowance, you must enter the mean value for the chamfer and the two groove depths. The value for the peak incline a is only to be defined for key forms G, H, and J, in accordance with the DIN 6885.1 standard.

35.5 Permissible pressure

The permitted values are calculated on the basis of the yield point (or fracture in the case of brittle materials).

35.6 Materials

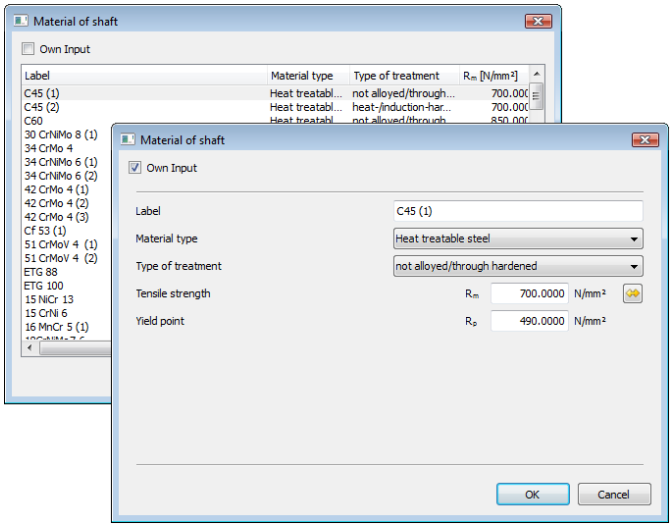


Figure 34.3: Materials dialog: Keys

In the selection list, you can select materials in accordance with the standard.
If you have set the `Own Input` flag, a new dialog appears here. This displays the material data used in the calculation which you can specify to suit your own purposes. You can also define your own materials directly in the database (see page I-114) so that these can also be used in future calculations.

35.7 Settings

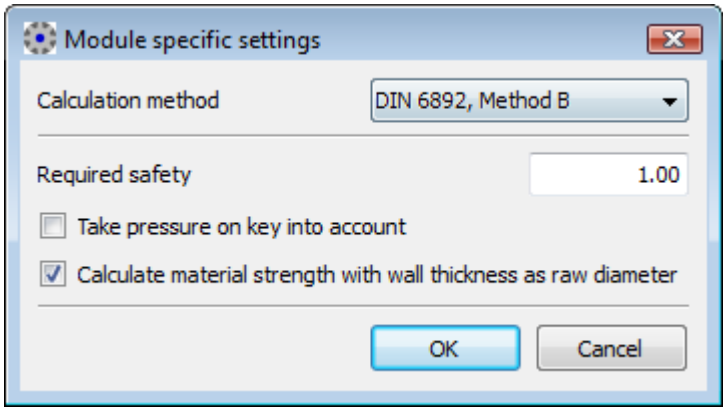


Figure 34.4: Settings: Key connection

- `Calculation method`
  Here you can select either DIN 6892 Method B or Method C. The default setting is method B, because Method C has been greatly simplified.
- `Take pressure on key into account`
  With this flag set the pressure on the key is taken into account when sizing the permissible pressure by clicking the button.
- `Calculate material strength with wall thickness as raw diameter`
  When the strength values for the hub are being set, either the external diameter (hub was turned from solid) or the wall thickness of the hub (hub was heat treated as a ring) is used.

35.8 Sizings

During the sizing process, the required value is defined such that the required safety factor (specified in `Calculations/ Settings`) is only just achieved. To view the results in the lower part of the main window, you must perform the calculation immediately after the sizing.

Possible sizings:

- transmissible torque
- necessary length of key way in shaft and hub

The "Keys" tutorial has been created specially to describe how you verify these keys.

Chapter 35
Spline shaft

Spline shaft connections are often used for adjustable, form-closed shaft-hub connections. Main areas of use: Vehicle gear trains, machine tools.

KISSsoft calculates the load on the shaft and hub (surface pressure) for spline shafts. The load placed on the shaft and hub (surface pressure), together with determining the safeties, is calculated as described in classic technical literature [64]). The calculation defined by Niemann forms the basis of DIN 6892 (key calculation).

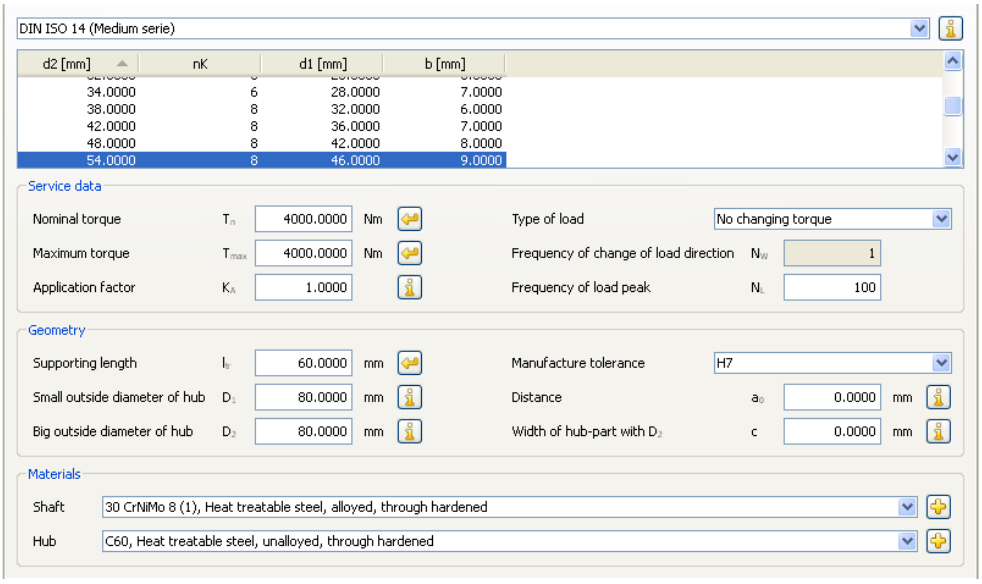


Figure 35.1: Basic data: Spline shafts

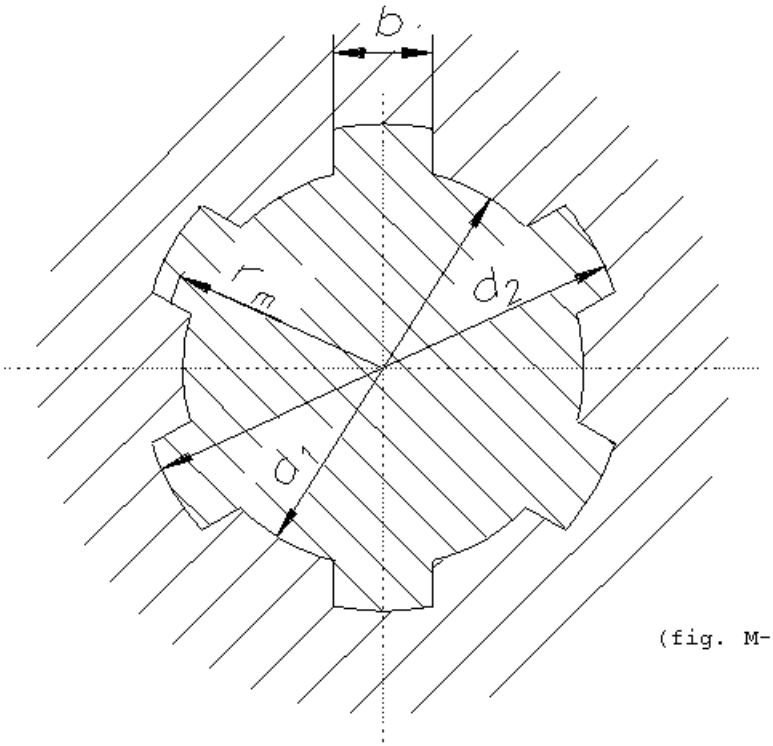
36.1 Standard profiles

You can select one of these standards from the selection list:

- DIN ISO 14 (light series)
- DIN ISO 14 (medium series)
- DIN 5464 (heavy, for vehicles)
- DIN 5471 (for machine tools)
- DIN 5472 (for machine tools)
- Own Input

In a spline shaft connection, after you select a standard, the program displays the corresponding external and inside diameters, and number of keys, along with their width.

Own Input: select the `Own Input` option to define your own spline shaft profile.



(fig. M-013)

36.2 Application factor

The application factor is defined in the same way as in the key calculation:

Operational behavior of the driving Machine	**Operational behavior of the driven machine**			
	equal-moderate	moderate shocks	medium shocks	strong shocks
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 35.2: Application factor in accordance with DIN 6892

36.3 Torque curve/Frequency of change of load direction

When you select the torque curve you can choose one of three positions:

1. `No alternating torque`
2. `Alternating torque, slow increase`
3. `Alternating torque, fast increase`

If you select positions 2) and 3), the calculation also defines a frequency of change of load direction factor f_w as defined in DIN 6892 Figure 6, as well as the frequency of change of load direction. In the case of position 1) the factor will be set to 1.0.

36.4 Occurring flank pressure

This formula is used to calculate occurrences of flank pressure. The formula is used both for the equivalent load and for the maximum load:

$p(eq,max)=k_{\varphi\beta}(eq,max) * k_1 * T * 2000/(d_m * l_{tr} * h * z)$

$k_{\varphi\beta}$: Share factor

k_1: Length factor

T Torque

d_m: average diameter

l_{tr}: supporting length

h spline height

z Number of keys

36.5 Length factor

A length factor, k_1, is multiplied by the loading that takes into account how the load is distributed across the bearing length as a consequence of the torque action of the shaft and hub. The length factor depends on the equivalent diameter derived from the bearing length, the small and the large outside hub diameter, and the width c to the external diameter. The distance a_0 is also used to determine the length factor. This coefficient is shown in a diagram in Niemann.

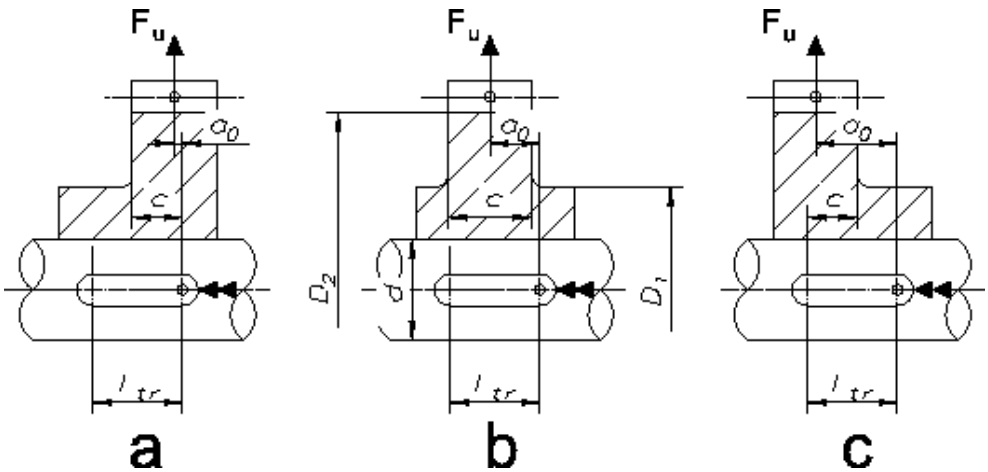


Figure 35.2: Spline shafts: Load application.

36.6 Share factor

To calculate the occurring flank pressure, a share factor of $k_{\varphi\beta}$ is taken into account. This is then multiplied by the load. Interim sizes not shown in the table are interpolated linearly.

Form-closure connection	**spline connection with involute flank** **Tolerance zones in accordance with DIN 5480**					
	H5/IT4	**H7/IT7**	**H8/IT8**	**H9/IT9**	**H11/IT11**	**Maximum value**
$k_{\varphi\beta}$eq	1.1	1.3	1.5	2	4	z/2
$k_{\varphi\beta}$max	1	1.1	1.3	1.7	3	z/2

Table 35.4: Share factor in accordance with Niemann

36.7 Permissible pressure

The permitted values are calculated on the basis of the yield point (or fracture in the case of brittle materials). For continuous stress with T_{eq}:

- for ductile materials:

$p_{eq}=f_s * f_H * R_p$

- for brittle materials:

$p_{eq}=f_s * R_m$

Structural steel	Material	f_s
Shaft	Structural steel, heat treatable steel, case-hardened steel, GJS, GS GJL	1.2 1.0
Hub	Structural steel, heat treatable steel, case-hardened steel, GJS, GS GJL	1.5 2.0

Table 23.6: Support factor as specified by Niemann

The support factor f_s takes into account the support effect that appears in components subjected to a compression load.
The hardness influence coefficient, f_H, is derived from the ratio of surface to core strength for surface hardened components.
The hardness influence coefficient for case-hardened steel is 1.15, otherwise it is 1.0. The values used for this coefficient are defined in DIN 6892.

For calculation with peak torque:

$p_{max}=f_L * p_{eq}$

f_L is the frequency of load peak factor, which depends on the material type and the frequency of load peak. This coefficient is shown in a diagram in DIN 6892.

36.8 Materials

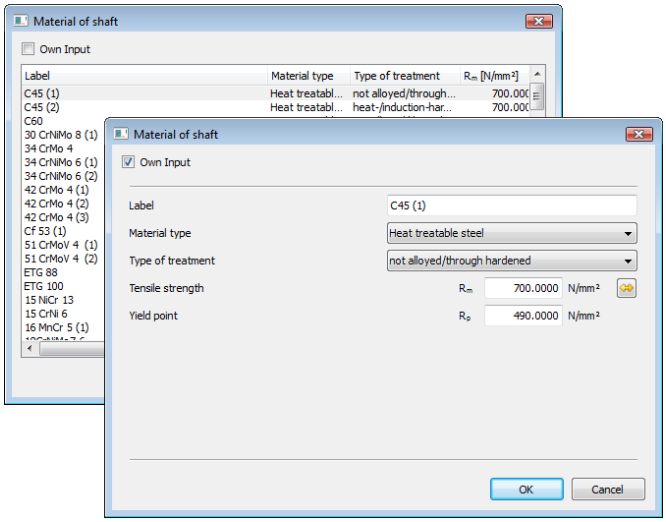


Figure 35.3: Materials screen: Spline shaft

In the selection list, you can select materials in accordance with the standard.
If you have set the `Own Input` flag, a new dialog appears here. This displays the material data used in the calculation which you can specify to suit your own purposes. You can also define your own materials directly in the database (see page I-114) so that these can also be used in future calculations.

36.9 Settings

In Settings you can specify the required safety for the connection. The values that are being searched for are defined on the basis of the required safety during sizing.

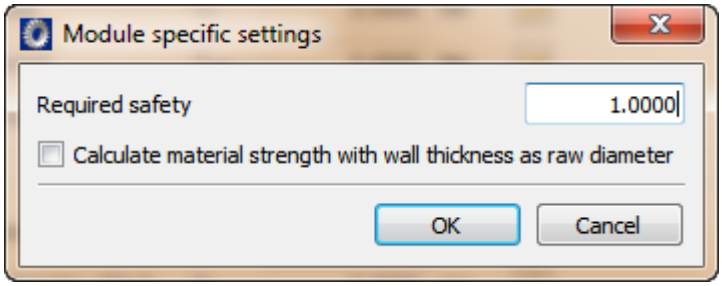


Figure 35.4: Settings: Spline shaft

If you selected `Calculate material strength with wall thickness as raw diameter`, the strength of the hub material is calculated using the wall thickness instead of the raw diameter.

36.10 Sizings

During the sizing process, the required value is defined such that the required safety factor (specified in `Calculations/ Settings`) is only just achieved. To display the results in the lower part of the main window, you must perform the calculation after the sizing. Possible sizings:

- transmissible nominal torque T_n
- transmissible maximum torque T_{max}
- supporting length l_{tr}

Chapter 36
Spline (strength)

Splines are spur gear toothings that have a shortened tooth depth and a large pressure angle (usually 30°). In KISSsoft, you can use one of two different calculation modules to calculate splines. The geometry and tolerances required for manufacture, and the strength calculation, are described in the Splines chapter (Geometry and Strength) (Z09a (see page IV-855)) under Connections.

For splines, you must calculate the load on shaft and hub (surface pressure). You can also add additional standards. Teeth data is defined in the database and therefore you can make the use of in-house profiles mandatory. You can also use the KISSsoft Spline (Geometry and Strength) module Z09a to calculate the manufacturing mass and the tolerances. The load placed on the shaft and hub (surface pressure), together with determining the safeties, is calculated as described in classic technical literature [64]).

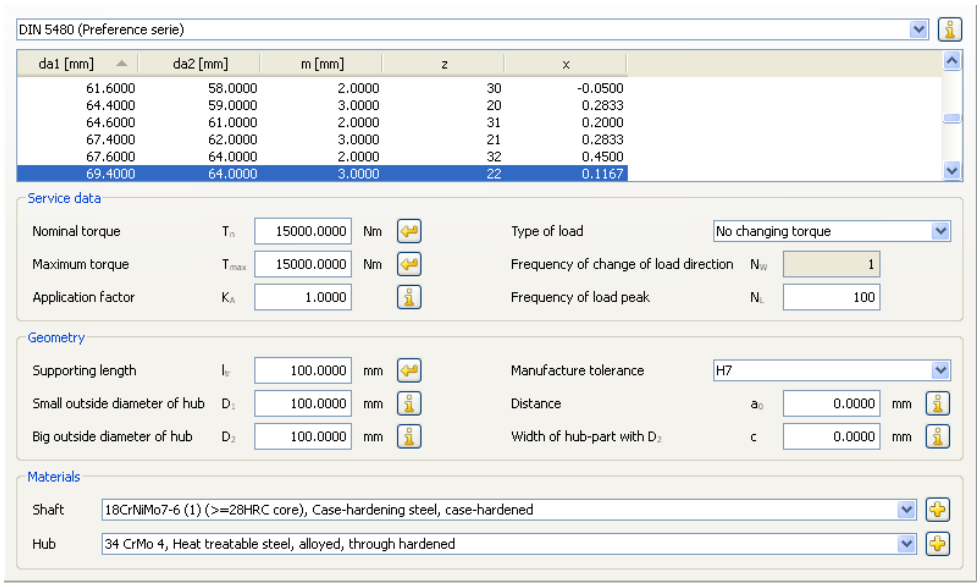


Figure 36.1: Basic data: Spline (strength)

37.1 Standard profiles

You can choose one of these standards from the selection list:

- DIN 5480
- DIN 5481
- DIN 5482
- ISO 4156
- ANSI B92.1
- ANSI B92.2M
- Own input (tip diameter of shaft and hub, module, number of teeth, profile shift coefficient)

For splines, the corresponding values are displayed in the list after the norm selection.

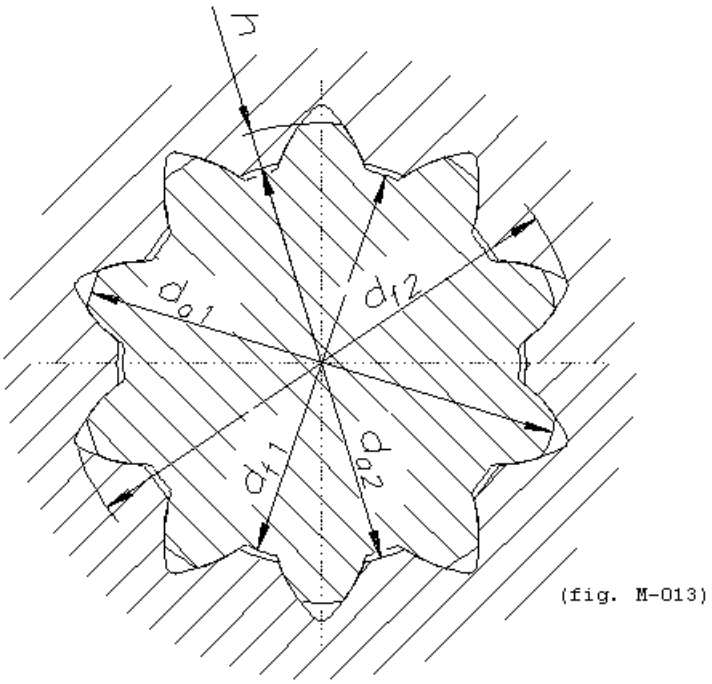
d_{a1}: Tip diameter of the shaft

d_{a2}: Tip diameter of the hub

m Module

z Number of teeth

x Profile shift coefficient



Own input: Select the `Own Input` option to enter your own data for the spline. The critical factor here is that the tip diameter of the shaft is greater than the tip diameter of the hub. If not, an error message is displayed.

37.2 Application factor

The application factor is defined in the same way as in the key calculation:

Operational behavior of the driving Machine	**Operational behavior of the driven machine**			
	equal-moderate	moderate shocks	medium shocks	strong shocks
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 36.2: Application factor in accordance with DIN 6892

37.3 Torque curve/Frequency of change of load direction

When you select the torque curve you can choose one of three positions:

1. `No alternating torque`
2. `Alternating torque, slow increase`
3. `Alternating torque, fast increase`

If you select positions 2) and 3), the calculation also defines a frequency of change of load direction factor f_w as defined in DIN 6892 Figure 6, as well as the frequency of change of load direction. In the case of position 1) the factor will be set to 1.0.

37.4 Occurring flank pressure

This formula is used to calculate occurrences of flank pressure. The formula is used both for the equivalent load and for the maximum load:

$p(eq,max)=k_{\varphi\beta}(eq,max) * k_1 * T * 2000/(d_m * l_{tr} * h * z)$

$k_{\varphi\beta}$: Share factor

l_{tr}: supporting length

k_1: Length factor

h spline height

T Torque

z Number of keys

d_m: average diameter

37.5 Length factor

A length factor, k_1, is multiplied by the loading that takes into account how the load is distributed across the bearing length as a consequence of the torque action of the shaft and hub. The length factor depends on the equivalent diameter derived from the bearing length, the small and the large outside hub diameter, and the width c to the external diameter. The distance a_0 is also used to determine the length factor. This factor is shown in a diagram in Niemann.

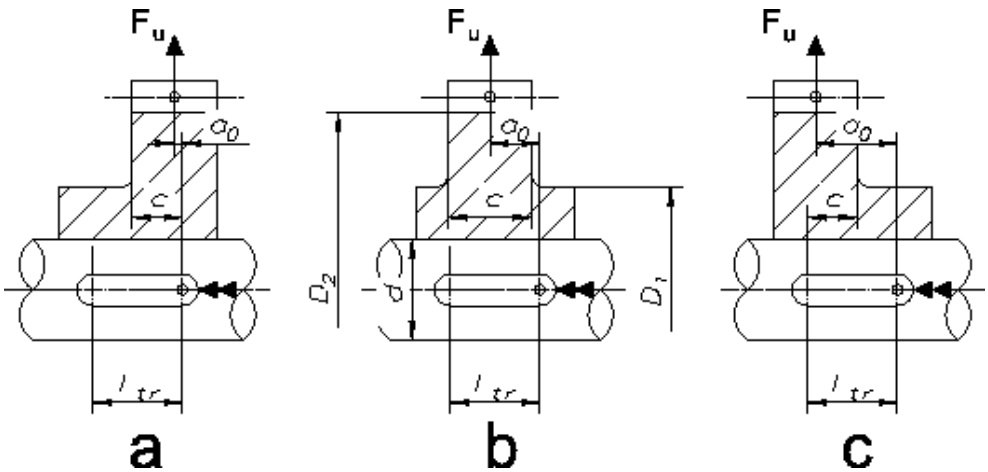


Figure 36.2: Splines: Load application.

37.6 Share factor

To calculate the occurring flank pressure, a share factor of $k_{\varphi\beta}$ is taken into account. This is then multiplied by the load. Interim sizes not shown in the table are interpolated linearly.

Form-closure connection	**spline connection with involute flank**					
	Tolerance zones in accordance with DIN 5480					
	H5/IT4	**H7/IT7**	**H8/IT8**	**H9/IT9**	**H11/IT11**	**Maximum value**
$k_{\varphi\beta}$eq	1.1	1.3	1.5	2	4	z/2
$k_{\varphi\beta}$max	1	1.1	1.3	1.7	3	z/2

Table 36.4: Share factor in accordance with Niemann

37.7 Permissible pressure

The permitted values are calculated on the basis of the yield point (or fracture in the case of brittle materials). For continuous stress with T_{eq}:

- for ductile materials:

$p_{eq}=f_s * f_H * R_p$

- for brittle materials:

$p_{eq}=f_s * R_m$

Structural steel	**Material**	**f_s**
Shaft	Structural steel, heat treatable steel, case-hardened steel, GJS, GS	1.2
	GJL	1.0
Hub	Structural steel, heat treatable steel, case-hardened steel, GJS, GS	1.5
	GJL	2.0

Table 36.6: Support factor as specified by Niemann

The support factor f_s takes into account the support effect that appears in components subjected to a compression load.
The hardness influence coefficient, f_H, is derived from the ratio of surface to core strength for surface hardened components.
The hardness influence coefficient for case-hardened steel is 1.15, otherwise it is 1.0. The values used for this coefficient are defined in DIN 6892.

For calculation with peak torque:

$p_{max}=f_L * p_{eq}$

f_L is the frequency of load peak factor, which depends on the material type and the frequency of load peak. This coefficient is shown in a diagram in DIN 6892.

37.8 Materials

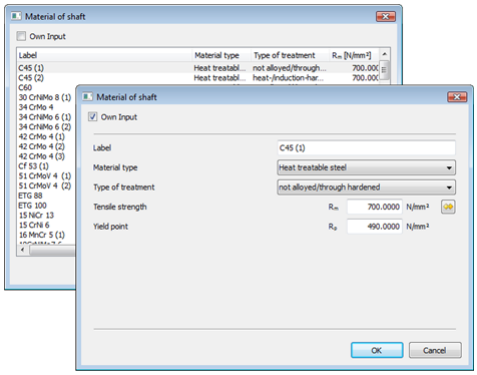


Figure 36.3: Materials screen: Spline

In the selection list, you can select materials in accordance with the standard.
If you have set the `Own Input` flag, a new dialog appears here. This displays the material data used in the calculation which you can specify to suit your own purposes. You can also define your own materials directly in the database (see page I-114) so that these can also be used in future calculations.

37.9 Settings

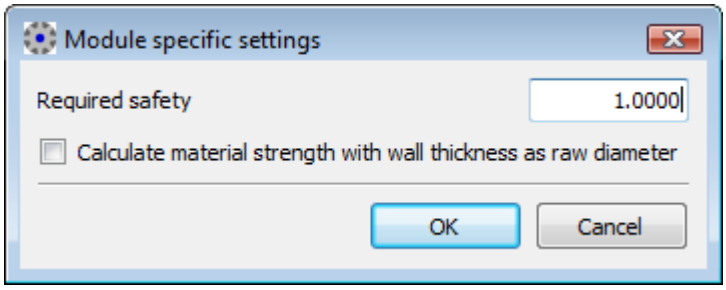


Figure 36.4 Settings for splines

In Settings you can specify the required safety for the connection. The values that are being searched for are specified on the basis of the required safety during sizing.

If you selected `Calculate material strength with wall thickness as raw diameter`, the strength of the hub material is calculated using the wall thickness instead of the raw diameter.

37.10 Sizings

During the sizing process, the required value is defined such that the required safety factor (specified in `Calculations/ Settings`) is only just achieved. To display the results in the lower part of the main window, you must perform the calculation after the sizing.

Possible sizings:

- transmissible nominal torque T_n
- transmissible maximum torque T_{max}
- supporting length l_{tr}

Chapter 37
Spline (geometry and strength)

You can calculate the geometry and the control mass of splines and hub in accordance with DIN 5480 (1986 Edition), ISO 4156, ANSI B92.1 or ANSI B92.2M. The tolerance system has been completely implemented in accordance with DIN 5480, Sheet 14. It also includes strength calculation in accordance with Niemann or DIN 5466.

The geometry profiles in accordance with DIN 5481 (2005) and in accordance with DIN 5482(1973) are saved in files.

When you open the file for the profile you require, the KISSsoft screens are filled with all the necessary geometry settings.

38.1 Underlying principles of calculation

38.1.1 General

Involute short cut teeth are often used for couplings. Teeth with large pressure angles $\alpha_n = 30^o$ are very common and, to increase strength, they have a tooth depth that is half the size of normal cylindrical gears. Couplings with teeth as defined in DIN 5480 are very widespread, and are precisely described with regard to geometry and tolerance calculation. The strength calculation is performed in accordance with the usual methods described in technical literature [5],[42].

NOTE

The moment of inertia is calculated as follows: the inside diameter of the shaft is $d_i = 0$, and the hub external diameter is the rounded result of $d_i = d_f + 4 \cdot m_n$. The moment of inertia is then determined for the cylinder between d_i and $(d_a + d_f)/2$.

38.1.2 Calculation of spline connections as described in DIN 5480 with diameter centering

Diameter centered connections are centered in the outside or inside diameters. The hub root diameter with the shaft tip diameter is used for outside centering, and the pinion center root diameter and the hub tip diameter is used for inside centering. Here, the gear toothing is only used for rotational synchronization. The connection must therefore have sufficient flank clearance to prevent the centering intersecting. Due to the small tolerances of the centering diameter, diameter-centered connections require more manufacturing effort to limit the central displacement. This is why they are only used in exceptional circumstances.

To calculate diameter-centered connections:

1. In the `Connections > Splines (Geometry and Strength)` calculation module, open the `Reference profile` input window by clicking on its tab. Here, select the `DIN5480 Major diameter fit` option in the `Reference profile` drop-down list in the `Shaft` and `Hub` area.
2. Click the `Tolerances` tab to open the Tolerances input window. Check that no flag has been set in the checkbox to the right of `Tip diameter deviation (upper/lower)` and `Root diameter deviation (upper/lower)` input fields, for `Shaft` or for `Hub`. The program then prompts with values from the DIN 5480 recommendations. For the tip circle, the following apply:

- for outside centering, H6 for the shaft tip diameter and H11 for the hub tip diameter
- for inside centering, h11 for the shaft tip diameter and H7 for the hub tip diameter

For the root circle, the following apply:

- for outside centering h14 for the shaft root diameter and H7 for the hub root diameter
- for inside centering, h6 for the shaft root diameter and H14 for the hub root diameter

9H/9e is recommended for the tooth thickness allowance.

38.2 Basic data

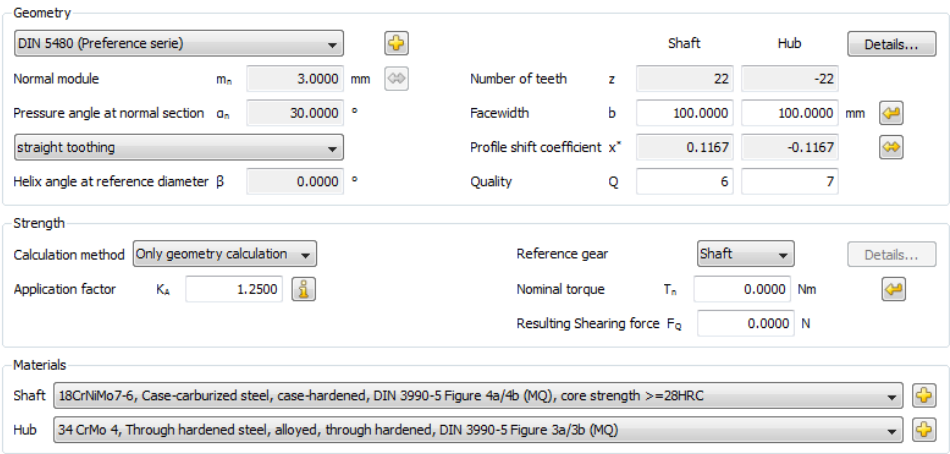


Figure 37.1: `Basic data` input window in the `Splines (Geometry and Strength)` module

38.2.1 Geometry standards

In the drop-down list in the upper left-hand part of the `Geometry` area, you see a list of the available geometry standards. To view a specific standard, click the button to the right of the drop-down list to open the `Define profile` view dialog window. The complete standard and preference sequences are also available for most of the standards in this list. Use the database tool (see page I-114) to add your own standards to the list or extend existing guidelines. For example, the preference sequence for DIN 5480 is stored in the `M02C-001.dat` file in the `dat` folder of your KISSsoft installation folder. Each line corresponds to an entry in the `Define profile` list and uses the following syntax:

d_{a1} | d_{a2} | m_n | z | i^*

where

d_{a1} Tip diameter, shaft

d_{a2} Tip diameter, hub

m_n Normal module

z Number of teeth

x^* Profile shift coefficient shaft

EXAMPLE:

```
KISSsoft Protokoll          27. February 2008, 09:35:12
#Units:mm|mm|mm||
3.50|2.72|.5|6|0.00
4.00|3.18|.5|7|0.00
4.50|3.66|.5|8|0.00
5.00|4.14|.5|9|0.00
5.50|4.62|.5|10|0.00
6.00|5.11|.5|11|0.00
6.50|5.60|.5|12|0.00
```

Figure 37.2: Example entry in `M02C- 001.dat`

The marked entry in `KISSedit` (see Figure 37.2) stands for d_{a1} = 5.5mm, d_{a2} = 4.62mm, m_n = 0.5mm, z = 10 and x^* = 0.

NOTE

You can only edit the Normal module, Number of teeth, and Profile shift coefficient, input fields if you have first selected `Own Input` in the drop-down list for geometry standards.

38.2.2 Normal module

Enter the normal module. However, if you know the pitch, transverse module, or diametral pitch, instead of this, click the button to open a dialog window in which you can perform the conversion. If you want to transfer the diametral pitch instead of the normal module, you can select `Input normal diametral pitch instead of normal module` by selecting `Calculation > Settings > General`.

38.2.3 Pressure angle at normal section an

The normal pressure angle at the reference circle is also the flank angle of the reference profile. For splines the pressure angle is usually $\alpha_n = 30^o$.

38.2.4 Number of teeth

For internal toothed gears, you must enter the number of teeth as a negative value as stated in DIN 3960. The shaft and the hub must have the same number of teeth, but with different signs.

38.2.5 Profile shift coefficient

The tool can be adjusted during manufacture. The distance between the production pitch circle and the tool reference line is called the profile shift. To create a positive profile shift, the tool is pulled further out of the material, creating a tooth that is thicker at the root and narrower at the tip. To create a negative profile shift the tool is pushed further into the material, with the result that the tooth is narrower and undercut may occur sooner. For pinion and gear factors:

$$x_1^* = -x_2^*$$

Click the button and KISSsoft will determine whether the profile shift coefficient is to be taken from measured data or from values given in drawings.

The following options are available here:

- **Base tangent length**
  Here you must enter the base tangent length (span) and the number of teeth spanned. This option cannot be used for (internal) helical gear teeth because their base tangent length cannot be measured.
- **Measurement over two balls**
  To do this, enter this dimension and the diameter of the ball. In a gear with helical gear teeth and an odd number of teeth, the measurement over balls is not the same as the measurement over two pins, see Measurement over pins.
- **Measurement over 2 pins**
  To do this, enter this dimension and the diameter of the pin. For helical gear teeth and gears with an odd number of teeth, you must also enter a minimum span. This measurement cannot be calculated in internal helical gear teeth.
- **Tip circle**
  This is a rather imprecise calculation because the tip circle does not always depend solely on the profile shift.
- **Tooth thickness at reference circle**
  Here, you specify the tooth thickness. You can also enter the arc length or chordal length, and whether the value is in transverse or normal section.

NOTE

The profile shift coefficient of the shaft and hub must be the same value.

38.2.6 Quality

In this input field you specify the accuracy grade in accordance with the standard shown in brackets. To change the standard used for this calculation, select `Calculation > Settings > General> Input of quality`. The accuracy grade specified in ISO 1328 is approximately the same quality as in DIN 3961 or BS 436/2.

Achievable qualities are shown in Table 37.1.

Manufacturing process	**Quality in accordance with DIN/ISO**		
Grinding	2	. . .	7
Shaving	5	. . .	7
Hobbing	(5)6	. . .	9
Milling	(5)6	. . .	9
Shaping	(5)6	. . .	9
Punching, Sintering	8	. . .	12

Table 37.1: Quality values for different manufacturing processes

38.2.7 Geometry details

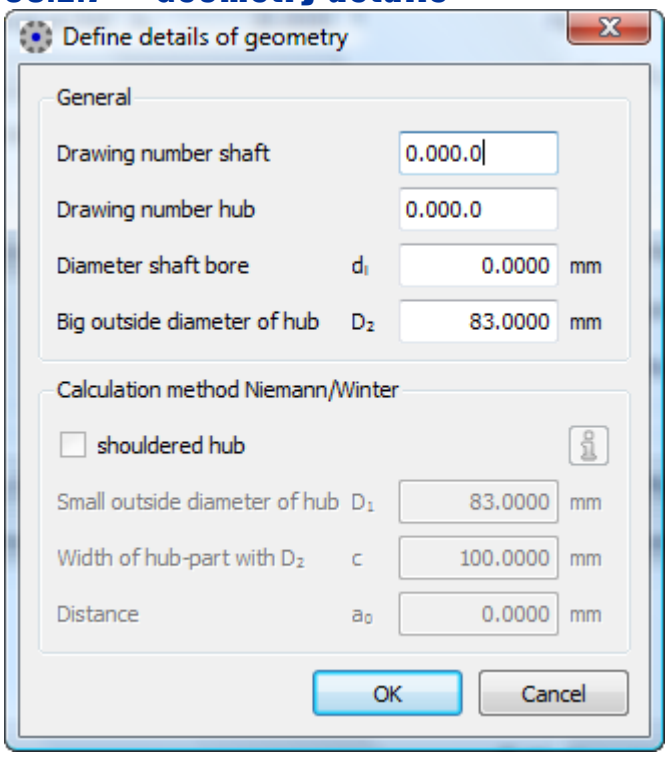


Figure 37.3: `Define details of geometry` dialog window

To open the `Define geometry details` window, click the `Details...` button in the upper right-hand part of the Geometry area. Here you can change the values for:

- Shaft and hub center drawing numbers
- Shaft bore diameter d_I
- Large hub outside diameter D_2

If you perform the calculation as defined in Niemann, you must also enter additional values. Depending on the position of the load, you can enter the value a0. If a shouldered hub is present, you must also enter the small hub external diameter D_1 and the width of the center part c (with D_2). The following diagram shows how to define these values:

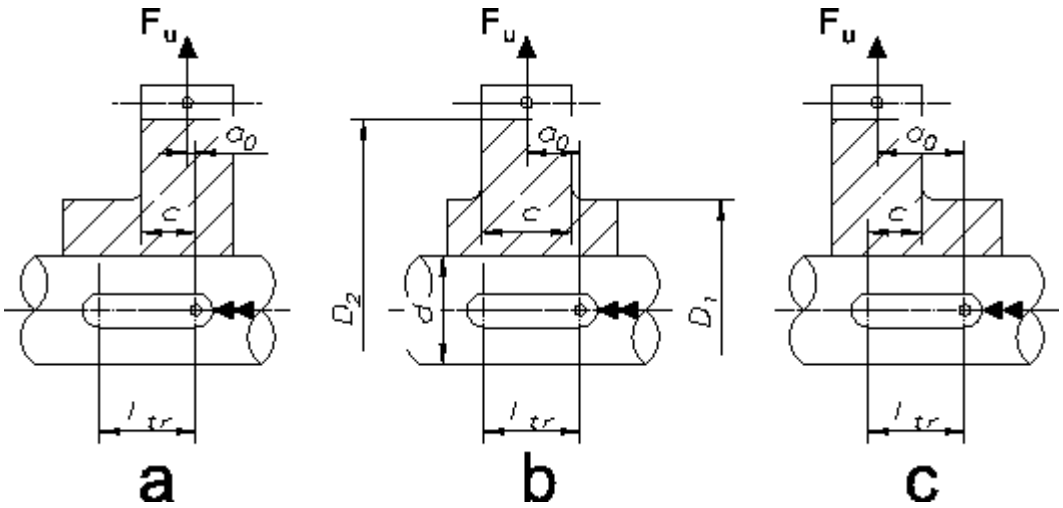


Figure 37.4: Niemann parameter definition

38.2.8 Strength calculation method

You can calculate strength either as defined in Niemann [64] or in accordance with DIN 5466. As DIN 5466 is still being developed, it is not described in any further detail. To perform the calculation in accordance with DIN 5466 and Niemann, you must make additional entries in the `Define details of strength` (see page **IV-865**) dialog window.

38.2.9 Application factor

The application factor compensates for any uncertainties in loads and impacts, whereby $K_A \geq 1.0$. Table 37.2 illustrates the values that can be used for this factor. You will find more detailed comments in ISO 6336, DIN 3990 and DIN 3991.

Operational behavior of the driving	**Operational behavior of the driven machine**			
	equal-	moderate	medium	strong

Machine	moderate	shocks	shocks	shocks
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 37.2: Assignment of operational behavior to application factor

38.2.10 Resulting shearing force

Shearing forces vertical to the shaft axis cause flank contact on both sides of the opposing side of the contact point. (DIN 5466)

38.2.11 Define details of strength

Click the `Details...` button in the `Strength` group to open the `Define details of strength` window in which you can change the following parameters.

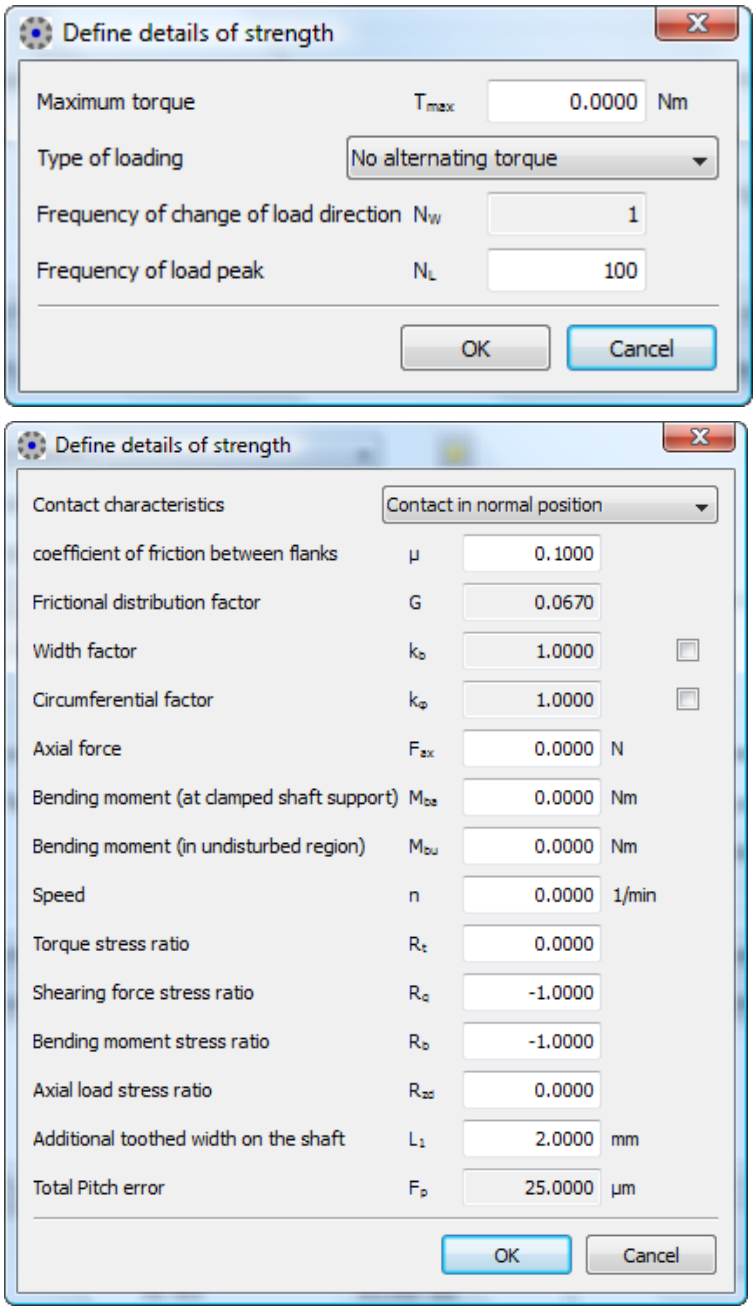


Figure 37.4: `Define details of strength` dialog window for calculation methods described in Niemann (left) and DIN 5466 (right)

The strength method as described in Niemann is described in more detail in the Spline (see page IV-843) chapter.

38.2.11.1 Type of loading/Frequency of change of load direction

If you open the `Type of loading` list, you can then select one of the three items shown in it:

1. `No alternating torque`
2. `Alternating torque, slow increase`
3. `Alternating torque, fast increase`

If you select positions 2) and 3), the calculation also defines the `frequency of change of load direction NW` as defined in DIN 6892 Figure 6, and a frequency of change of load direction factor f_w. In the case of position 1) the coefficient will be set to 1.0. This data is only used for calculations as described in Niemann.

$p_{eq}=f_w * p_{zul}$

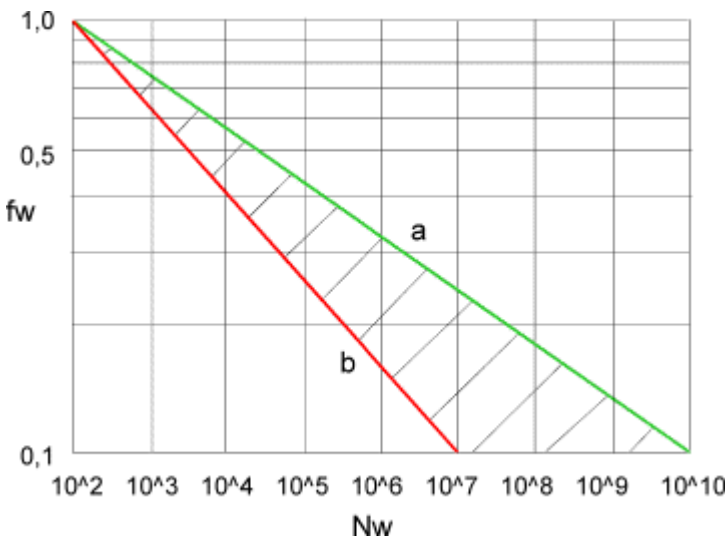


Figure 37.5: Graphic as described in DIN 6892 Figure 6: Load direction changing coefficient for reciprocal load

- a: Alternating torque, slow increase
- b: Alternating torque, fast increase

38.2.11.2 Number of load peaks

fL is the frequency of load peak factor, which depends on the material type and the `frequency of load peak NL`. This coefficient is shown in a diagram in DIN 6892. This value is needed for calculations as described in Niemann.

For calculation with peak torque:

$p_{max}=f_L * p_{zul}$

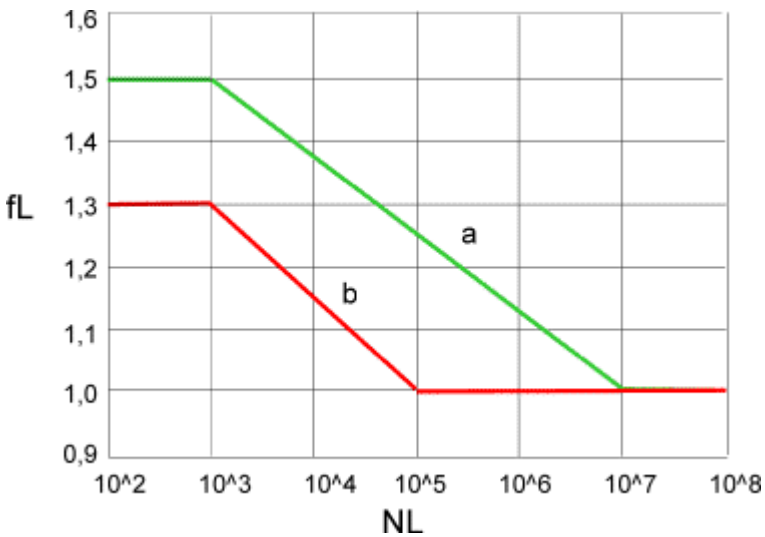


Figure 37.6: Graphic as described in Niemann (DIN 6892 Figure 7): Load peak coefficient

- a: ductile material
- b: brittle material

38.2.11.3 Stress ratio R

Stress ratios are the ratios between under and over stress with regard to a particular type of load, such as torque. Here R = -1 and defines a pure alternating stress ratio, R = 0 defines a pure pulsating stress ratio.

38.2.11.4 Width and circumferential factor

Click on the checkbox to the right of the input field for one of these factors to enter a value for that factor. Otherwise, this value is calculated automatically and may vary within the range [3, 5]. As these are multiplied together to define the load increase, you can achieve safeties of up to 20 times smaller than is possible with the calculation method defined in Niemann.

38.2.12 Materials

The materials displayed in the drop-down lists are taken from the materials database. If you cannot find the material you require in this list, you can either select `Own Input` from the list or enter the material in the database (see page I-114) first. Click the button to open the `Material hub/shaft` window in which you can select your material from a list of materials that are available in the database. Select the `Own Input` option to enter specific material properties. This option corresponds to the `Create a new entry` window in the database tool.

38.3 Tolerances

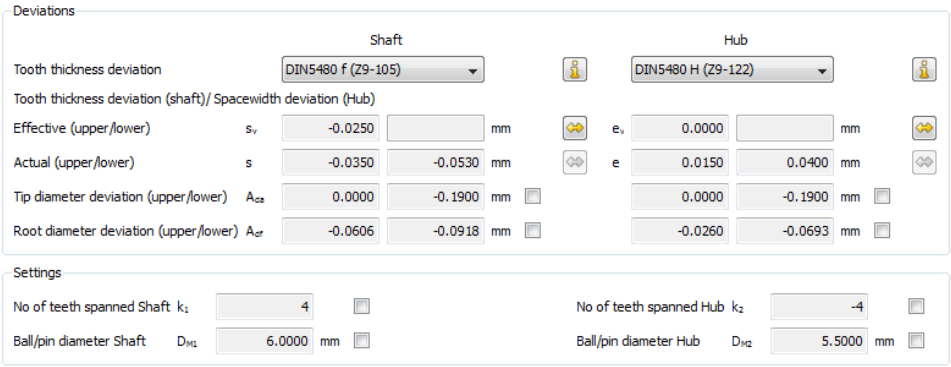


Figure 37.5: `Tolerances` input window in the `Splines calculation module`

38.3.1 Tooth thickness tolerance

Select one of the options from the `Thickness tolerance` drop-down list.

The allowances for "Actual" (smax, smin, emax, emin) correspond to the individual measurements (base tangent length or measurement over pin measured on the teeth). The deviations for "Effective" correspond to the measurement with templates (all teeth checked together). The gear tooth play of a spline connection is therefore derived from the "Effective" deviations. The effective allowances include not only the tooth thickness allowances of individual teeth but also a pitch and form error component. The "Effective" allowances are therefore theoretical values, and are smaller (the tooth is thicker) than for the "Actual" allowances.

NOTE

In accordance with the standard, the allowances for tooth thickness (smax, smin) are predefined for the shaft. In contrast, for the hub, the allowances apply to the tooth space (emax, emin).

If the tooth thickness tolerance has been set to your own specific value, you can input svmax for the shaft ("Effective" maximum deviation) to calculate svmin because the relationship applies in this case:

svmin – smin = svmax – smax

In addition, you can then use the flag to predefine the individual measurement allowances for "Actual". However, if this flag is not set, the difference svmax–smax (pitch and form error component), and the tolerance interval smax-smin are set according to the standard for the selected quality.

The same also applies to the hub.

38.3.1.1 DIN 5480

Unlike ISO 4156 or ANSI 92.1, DIN 5480 has the special feature that sveffmin = svmax always applies to the shaft and eveffmax = svmin to the hub. For this reason, sveffmin and eveffmax are not displayed.

NOTE

The tolerance widths for template entries are larger because of Taylor's formula [25].

38.3.1.2 ANSI 92.1 and ISO4156/ANSI 92.2M

If you have entered your `own tooth thickness tolerance value`, you must take the following points into account:
You must enter the tooth thickness allowance s_v for the `Effective` tooth thickness for the overall measurement (caliber) to suit the tolerance system that you are using to calculate cylindrical gears. The `Actual` tooth thickness s for single measurements is defined using these equations.

$$s_{\max} = s_{v,\max} - \lambda \tag{37.1}$$

$$s_{\min} = s_{v,\min} - \lambda \tag{37.2}$$

These equations apply to the shaft tooth thickness or to the tooth space of the hub.

38.3.2 Effective/Actual

Click the button to open the `Convert total deviation of tooth thickness Effective(Actual) for shaft` window which uses the corresponding screen to convert the Effective/Actual tooth thickness allowance

deviation. Here you can enter values either for the base tangent length, ball or roller measurement or the tooth thickness. (see Figure)

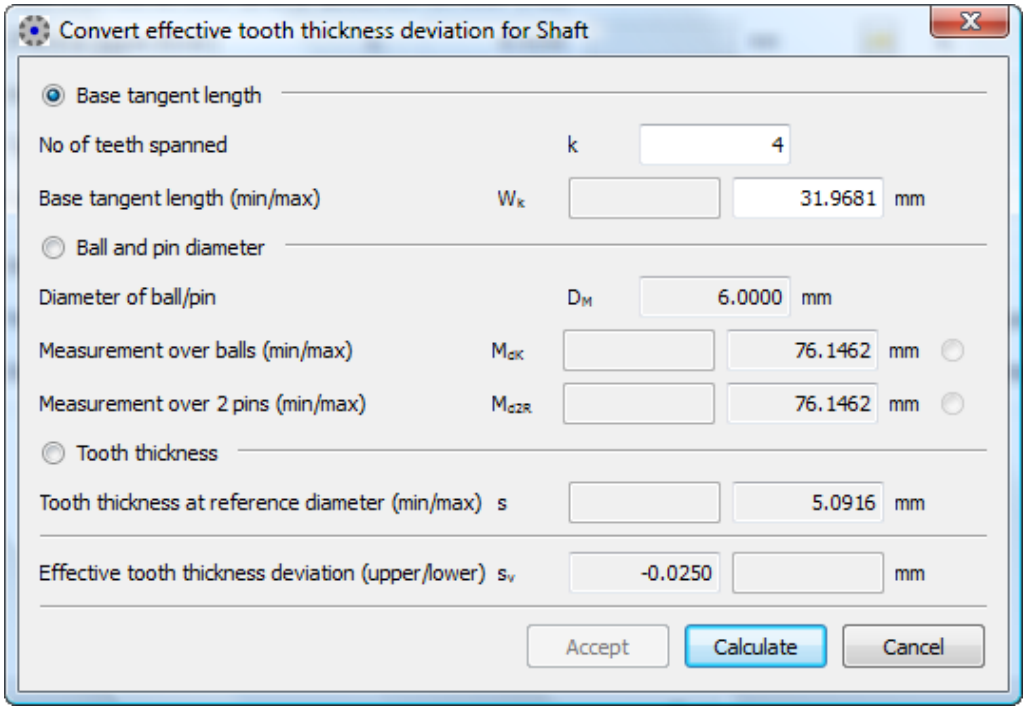


Figure 37.6: `Convert total Effective(Actual) deviation of tooth thickness for shaft` dialog window

38.3.3 Ball/pin diameter shaft/hub

The implemented DIN 5480, Part 1, contains an extract of the measuring roller diameter as specified in DIN 3977 that must be used here. You can decide whether to extend the list of available roller diameters in the `Z0Rollen.dat` file in the `dat` directory in your KISSsoft installation folder.

38.4 Templates

Spline connections are often checked using templates.

Go limits are always fully toothed (teeth all around the perimeter) and are used to test the **effective** tolerance limit. For hubs this is the **min. effective** tooth space and for shafts this is the **max. effective** tooth thickness.

No-go gauges are always toothed by sector (depending on the number of teeth of the test piece, 2 to 7 teeth located opposite to each other) and are used to test the **actual** tolerance limit. For hubs this is the **max. actual** tooth space and for shafts this is the **min. actual** tooth thickness. The externally located flanks of each sector are given sufficient clearance (flank relief, see 1 in the Figure), as they cannot be measured exactly.

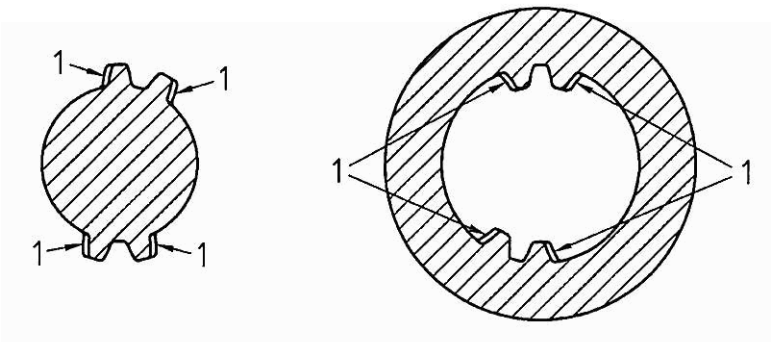


Figure 37.7: Displaying templates

The KISSsoft system can calculate all the gauge allowances specified in ISO 4156. To do this, select "Reports" and then call "Construction of gauges". The system does not automatically calculate the gauge dimensions for profiles that comply with DIN or ANSI. However, you can do this by simply following the specifications given in DIN 5480-15.

38.5 Tooth form

If you want to generate the tooth forms of a splined joint, you can select the data you require from the six different tolerance fields (actual, effective) and the three different diameter tolerances displayed in the `Tooth form` tab.

The default settings here are the average allowances for tooth thickness and diameter. After the calculation has been performed, the resulting diameter and tooth thickness are output in the tooth form report.

NOTE

To ensure the selected diameter tolerance is applied to the root diameter, set the flag next to the root diameter allowance in the `Reference profile` tab. If you do not set this flag, only the tooth thickness tolerance will be included in the calculation of the root diameter.

If this flag is not set, the calculation uses a default tolerance for the root diameter as defined in the standard. In other words, the diameter tolerance you selected will not be used to calculate the root diameter.

Chapter 38
Polygon

You use polygon connections to create shaft-hub connections that can withstand very heavy loads. In particular, the low notch effect present in this connection does not reduce shaft strength.

For polygon shafts you must calculate the load on the shaft and hub (surface pressure). You can also add additional standards.

You can use one of these two methods to calculate the load on the shaft and hub (surface pressure) and to define the safeties:

- Niemann, Volume I (4th Edition) [64].
- DIN 32711-2 (P3G profile) [84]/ DIN 32712-2 (P4C profile) [85]

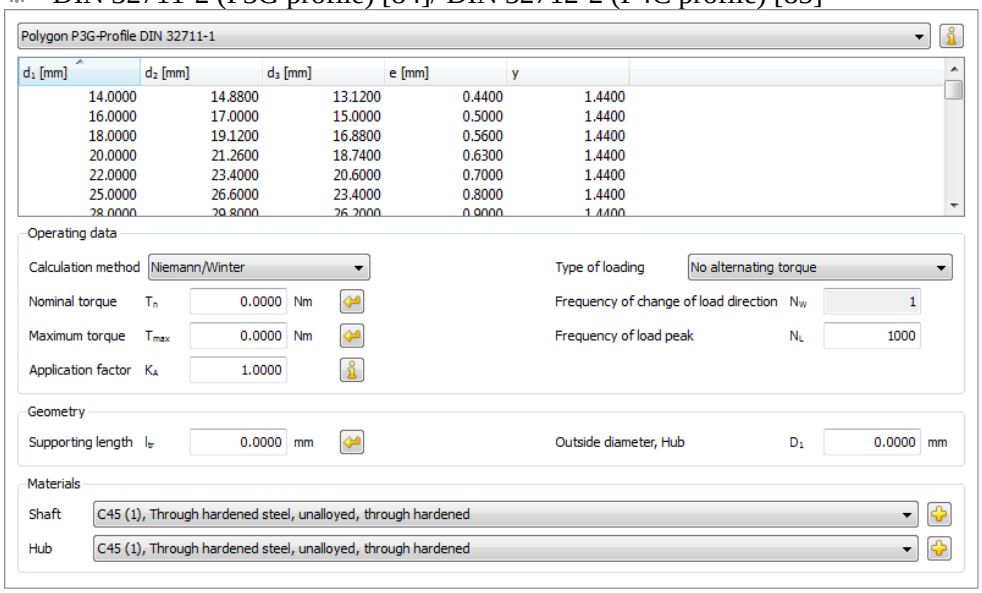


Figure 38.1: Basic data: Polygon

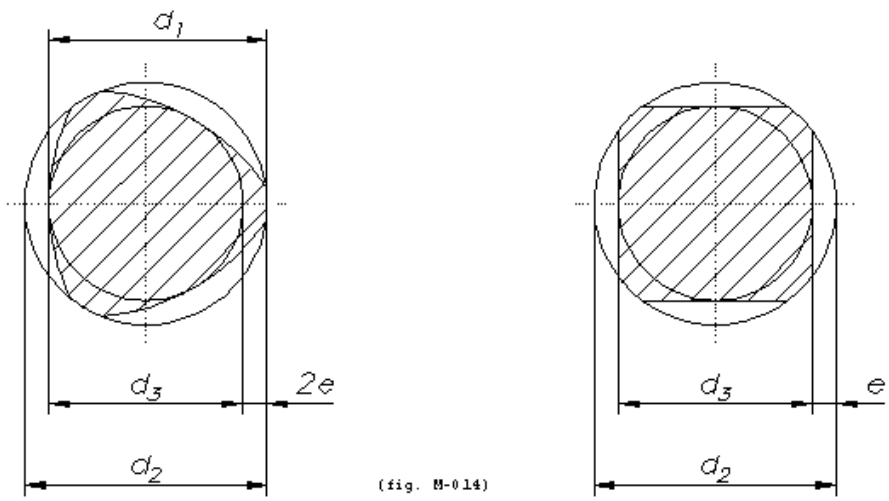
In the calculation in accordance with DIN, only the static load is observed. In the method in accordance with Niemann, the influence of alternating torque can be observed or

load peaks can also be calculated.

39.1 Standard profiles

You can select one of these standards from the selection list:

- DIN 32711-1 (P3G profile)
- DIN 32712-1 (P4C profile)



In a P3G profile, which varies according to which standard you select, the list displays the diameter of mean circle, d1, the diameter of outer circle, d2, the diameter of inner circle, d3, the eccentricity e and the factor y.
In a P4C profile, the diameter of outer circle d2, the diameter of inner circle, d3, the eccentricity e, and the factor y, are displayed in the list.

39.2 Application factor

The application factor is defined in the same way as in the key calculation:

Operational behavior of the driving Machine	**Operational behavior of the driven machine**			
	equal-moderate	moderate shocks	medium shocks	strong shocks
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 38.2: Application factor in accordance with DIN 6892

39.3 Torque curve/Frequency of change of load direction

This influence can only be made to apply using the Niemann calculation method.

When you select the torque curve you can choose one of three positions:

1. `No alternating torque`
2. `Alternating torque, slow increase`
3. `Alternating torque, fast increase`

If you select positions 2) and 3), the calculation also defines a frequency of change of load direction factor f_w as defined in DIN 6892 Figure 6, as well as the frequency of change of load direction. In the case of position 1) the factor will be set to 1.0.

39.4 Occurring flank pressure

Method in accordance with Niemann:

This formula is used to calculate occurrences of flank pressure. The formula is used both for the equivalent load and for the maximum load:

Profile P3G:

p(eq,max)=T * 1000/(l_{tr} * d1 * (0.75 * π * e + 0.05 * d1))

Projection area = l_{tr} * n * 2 * e; (n = 3)

d_1: Diameter of mean circle T Torque

l_{tr}: supporting length e Eccentricity

Profile P4C:

er = (d2 - d3) / 4; dr = d3 + 2 * e

p(eq,max)=T * 1000/(l_{tr} * (π *dr* er + 0.05 * d2^2))

Projection area = l_{tr} * n * 2 * er; (n = 4)

d_2: Diameter of outer circle T Torque

l_{tr}: supporting length e Eccentricity

d_r: Mathematically theoretical diameter e_r: Mathematical eccentricity

d_3: Diameter of inner circle

Method in accordance with DIN:

The following formula is used to calculate the occurrence of flank pressure:

Profile P3G:

p=T * 1000/(l_{tr} * d1 * (0.75 * π * e + 0.05 * d1))

d_1: Diameter of mean circle T Torque

l_{tr}: supporting length e Eccentricity

Profile P4C:

er = (d2 - d3) / 4; dr = d3 + 2 * e

p=T * 1000/(l_{tr} *dr (π * er + 0.05 * dr))

d_2: Diameter of outer circle

l_{tr}: supporting length

d_r: Mathematically theoretical diameter

d_3: Diameter of inner circle

T Torque

e Eccentricity

e_r: Mathematical eccentricity

39.5 Permissible pressure

Method in accordance with Niemann:

The permitted values are calculated on the basis of the yield point (or fracture in the case of brittle materials). For continuous stress with T_{eq}:

- for ductile materials:
$p_{eq}=f_s * f_H * R_p$

- for brittle materials:
$p_{eq}=f_s * R_m$

Structural steel	Material	f_s
Shaft	Structural steel, heat treatable steel, case-hardened steel, GJS, GS	1.2
	GJL	1.0
Hub	Structural steel, heat treatable steel, case-hardened steel, GJS, GS	1.5
	GJL	2.0

Table 38.6: Support factor as specified by Niemann

The support factor f_s takes into account the support effect that appears in components subjected to a compression load.
The hardness influence coefficient, f_H, is derived from the ratio of surface to core strength for surface hardened components.
The hardness influence coefficient for case-hardened steel is 1.15, otherwise it is 1.0. The values used for this coefficient are defined in DIN 6892.

For calculation with peak torque:

$p_{max}=f_L * p_{eq}$

f_L is the frequency of load peak factor, which depends on the material type and the frequency of load peak. This coefficient is shown in a diagram in DIN 6892.

Method in accordance with DIN:

The permissible surface pressure on the shaft or hub for polygon profiles P3G and P4C is:

$p_{zul} = 0.9 * R_{p0.2}$

39.6 Materials

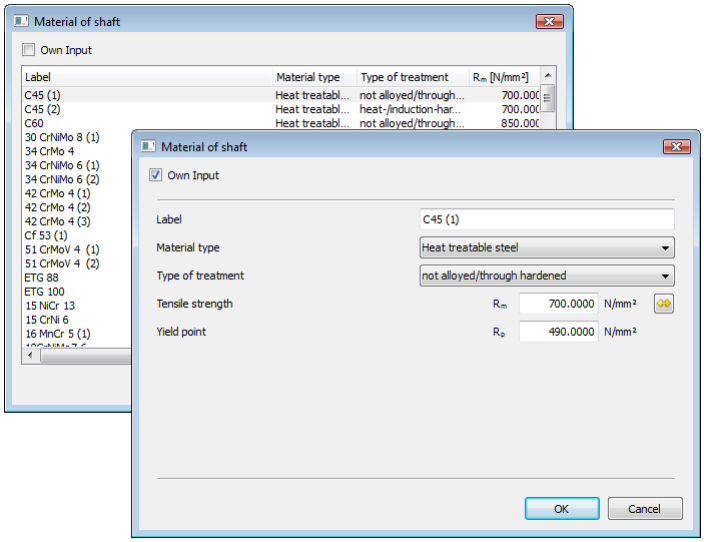


Figure 38.2: Materials screen: Polygon

In the selection list, you can select materials in accordance with the standard.
If you have set the `Own Input` flag, a new dialog appears here. This displays the material data used in the calculation which you can specify to suit your own purposes. You can also define your own materials directly in the database (see page I-114) so that these can also be used in future calculations.

39.7 Settings

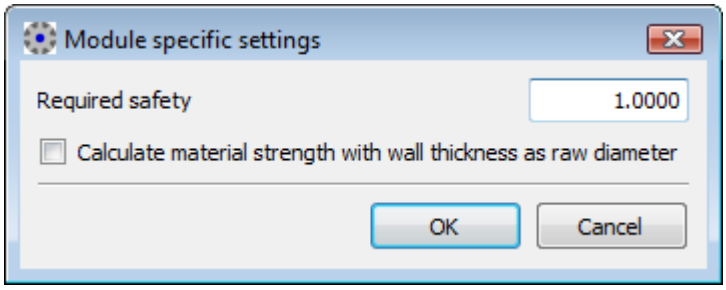


Figure 38.3: Settings: Polygon

In Settings you can specify the required safety for the connection. The values that are being searched for are defined on the basis of the required safety during sizing.

If you selected `Calculate material strength with wall thickness as raw diameter`, the strength of the hub material is calculated using the wall thickness instead of the raw diameter.

39.8 Sizings

During the sizing process, the required value is defined such that the required safety factor (specified in `Calculations/ Settings`) is only just achieved. To display the results in the lower part of the main window, you must perform the calculation after the sizing. Possible sizings:

- transmissible nominal torque T_n
- Transmissible maximum torque T_{max} (only for Niemann)
- supporting length l_{tr}

39.9 Graphics

The polygon form is defined using the formulae in the relevant DIN standard (32711-1/ 32712-1) and is displayed as a graphic which can be exported either as a picture file or as a DXF file.

Polygon curve equation (profile P3G, DIN 32711-1)

$$x(\alpha) = \left[Dm/2 - e \cdot \cos(3 \cdot \alpha)\right] \cdot \cos(\alpha) - 3 \cdot e \cdot \sin(3 \cdot \alpha) \cdot \sin(\alpha)$$

$$y(\alpha) = \left[Dm/2 - e \cdot \cos(3 \cdot \alpha)\right] \cdot \sin(\alpha) - 3 \cdot e \cdot \sin(3 \cdot \alpha) \cdot \cos(\alpha)$$

Polygon curve equation (profile P4C, DIN 32712-1):

$$x(\alpha) = \left[Dm/2 - e \cdot \cos(4 \cdot \alpha)\right] \cdot \cos(\alpha) - 4 \cdot e \cdot \sin(4 \cdot \alpha) \cdot \sin(\alpha)$$

$$y(\alpha) = \left[Dm/2 - e \cdot \cos(4 \cdot \alpha)\right] \cdot \sin(\alpha) - 4 \cdot e \cdot \sin(4 \cdot \alpha) \cdot \cos(\alpha)$$

Chapter 39
Woodruff Key

Connections that use Woodruff keys are no longer commonly used, because the deep keyway in these keys causes too great a notch effect. However, this connection is still used widely in precision mechanics.

For Woodruff keys you calculate the load on shaft and hub (surface pressure). You can also add additional standards. The load placed on the shaft and hub (surface pressure), together with determining the safeties, is calculated as described in classic technical literature [64]. The calculation defined by Niemann forms the basis of DIN 6892 (key calculation).

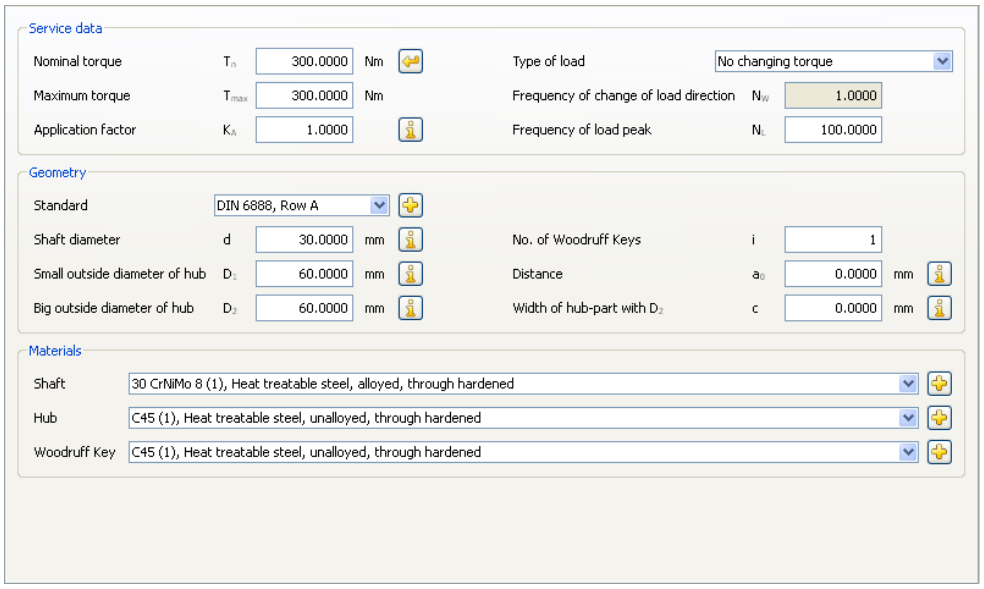


Figure 39.1: Basic data: Woodruff key

40.1 Standard profiles

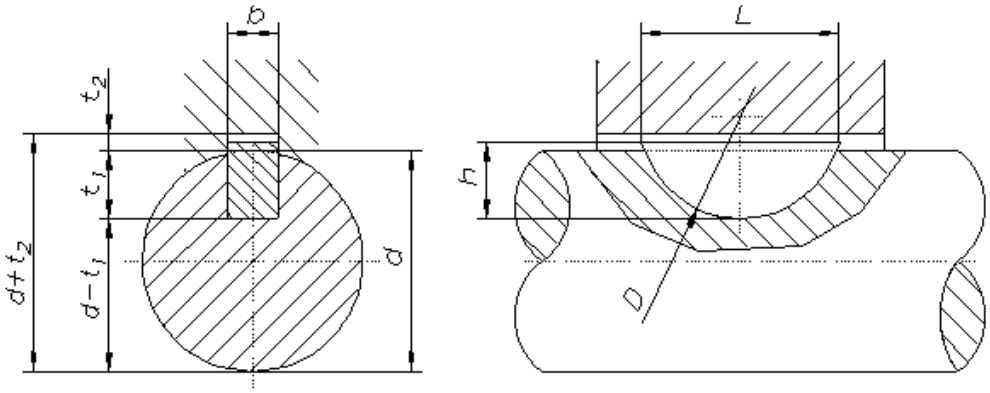
You can select one of these standards from the selection list:

- DIN 6888, series A (high pinion groove)
- DIN 6888, series B (low pinion groove)
- Own Input

After you select the standard for calculating the Woodruff key, a list of corresponding values is displayed.

b: Width d: Diameter

h: Height t_1: Shaft groove depth



(fig. M-015)

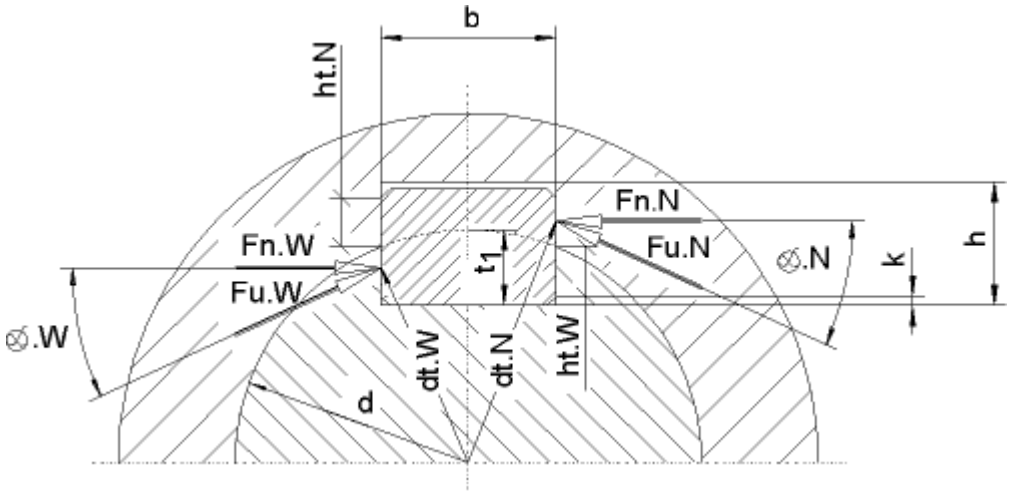


Figure 39.2: Woodruff key with circumferential and normal forces for the calculation as defined in Niemann

Own input: Select the `Own Input` option to define your own Woodruff keys.

40.2 Application factor

The application factor is defined in the same way as in the key calculation:

Operational behavior of the driving Machine	**Operational behavior of the driven machine**			
	equal-moderate	moderate shocks	medium shocks	strong shocks
uniform	1.00	1.25	1.50	1.75
light shocks	1.10	1.35	1.60	1.85
moderate shocks	1.25	1.50	1.75	2.00
heavy shocks	1.50	1.75	2.00	2.25

Table 39.2: Application factor in accordance with DIN 6892

40.3 Torque curve/Frequency of change of load direction

When you select the torque curve you can choose one of three positions:

1. `No alternating torque`
2. `Alternating torque, slow increase`
3. `Alternating torque, fast increase`

If you select positions 2) and 3), the calculation also defines a frequency of change of load direction factor f_w as defined in DIN 6892 Figure 6, as well as the frequency of change of load direction. In the case of position 1) the factor will be set to 1.0.

40.4 Occurring flank pressure

This formula is used to calculate occurrences of flank pressure. The formula is used both for the equivalent load and for the maximum load:

$$p(eq,max)=k_{\varphi\beta}(eq,max) * k_1 * T * 2000/(d * l_{tr} * h_{tw} * z)$$

$k_{\varphi\beta}$: Share factor

k_1: Length factor

T Torque

d Shaft diameter

l_{tr}: supporting length

h_{tw}: supporting height (shaft)

z Number of Woodruff Keys

40.5 Length factor

A length factor, k_1, is multiplied by the loading that takes into account how the load is distributed across the bearing length as a consequence of the torque action of the shaft and hub. The length factor depends on the equivalent diameter derived from the bearing length, the small and the large hub outside diameter, and the width c to the external diameter. The distance a_0 is also used to determine the length factor. This factor is shown in a diagram in Niemann.

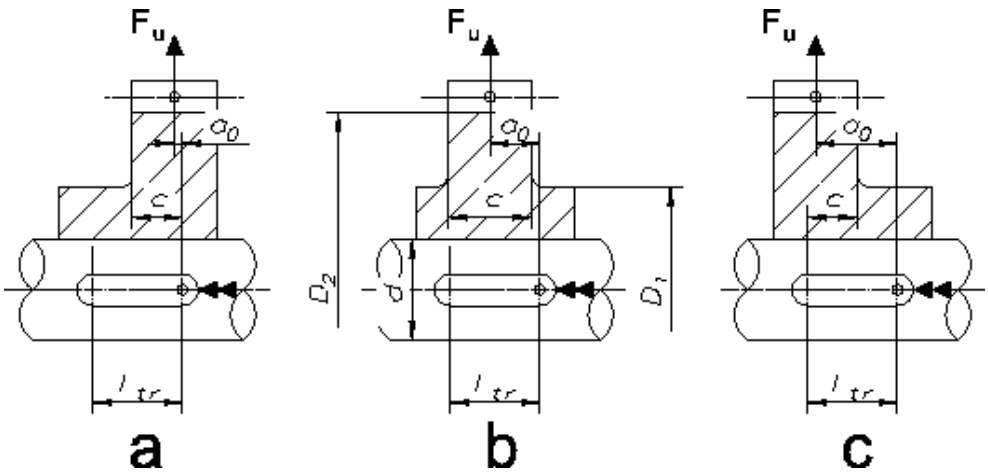


Figure 39.3: Woodruff key: Load application.

40.6 Share factor

To calculate the occurring flank pressure, a share factor of $k_{\varphi\beta}$ is taken into account. This is then multiplied by the load. Interim sizes not shown in the table are interpolated linearly.

Form-closure connection	**spline connection with involute flank** **Tolerance zones in accordance with DIN 5480**					
	H5/IT4	**H7/IT7**	**H8/IT8**	**H9/IT9**	**H11/IT11**	**Maximum value**
$k_{\varphi\beta}$eq	1.1	1.3	1.5	2	4	z/2
$k_{\varphi\beta}$max	1	1.1	1.3	1.7	3	z/2

Table 39.4: Share factor in accordance with Niemann

40.7 Permissible pressure

The permitted values are calculated on the basis of the yield point (or fracture in the case of brittle materials). For continuous stress with T_{eq}:

- for ductile materials:

$p_{eq}=f_s * f_H * R_p$

- for brittle materials:

$p_{eq}=f_s * R_m$

Structural steel	**Material**	**f_s**
Shaft	Structural steel, heat treatable steel, case-hardened steel, GJS, GS GJL	1.2 1.0
Hub	Structural steel, heat treatable steel, case-hardened steel, GJS, GS GJL	1.5 2.0

Table 1.6: Support factor as specified by Niemann

The support factor f_s takes into account the support effect that appears in components subjected to a compression load.
The hardness influence coefficient, f_H, is derived from the ratio of surface to core strength for surface hardened components.
The hardness influence coefficient for case-hardened steel is 1.15, otherwise it is 1.0. The values used for this coefficient are defined in DIN 6892.

For calculation with peak torque:

$p_{max}=f_L * p_{eq}$

f_L is the frequency of load peak factor, which depends on the material type and the frequency of load peak. This coefficient is shown in a diagram in DIN 6892.

40.8 Materials

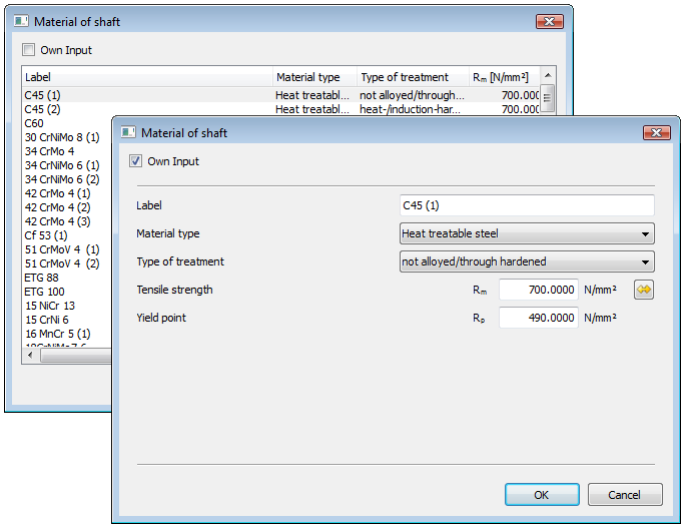


Figure 39.4: Materials screen: Woodruff key

In the selection list, you can select materials in accordance with the standard.
If you have set the `Own Input` flag, a new dialog appears here. This displays the material data used in the calculation which you can specify to suit your own purposes. You can also define your own materials directly in the database (see page I-114) so that these can also be used in future calculations.

40.9 Settings

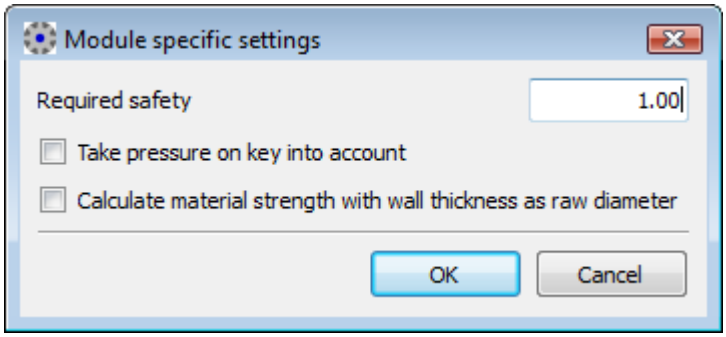


Figure 39.5: Settings: Woodruff key

In Settings you can specify the required safety for the connection. The values that are being searched for are specified on the basis of the required safety during sizing.

If the `Take pressure on key into account` flag is set, the values of the Woodruff key are included in the sizing. Otherwise the sizing procedure will be carried out on the basis of the shaft and hub.

If you selected `Calculate material strength with wall thickness as raw diameter`, the strength of the hub material is calculated using the wall thickness instead of the raw diameter.

40.10 Sizings

During the sizing process, the required value is defined such that the required safety factor (specified in `Calculations/ Settings`) is only just achieved. To display the results in the lower part of the main window, you must perform the calculation after the sizing.

Possible sizings:

- transmissible nominal torque T_n

Chapter 40
Bolts and pins

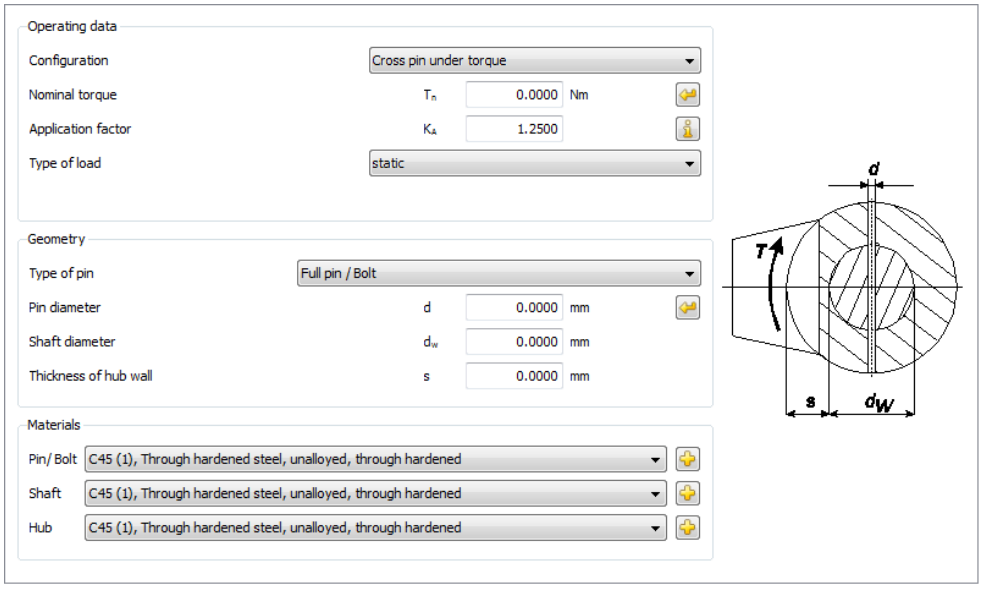


Figure 40.1: Basic data: Bolts and Pins

The bolt/pin connections are divided into four types of calculation depending on where they are used:

- `Cross pin under torque`
  With cross pin connections where large forces are in play, the surface pressure of the shaft and hub, and the shearing of the pin, will be checked.
- `Longitudinal pin under torque`
  Longitudinal pin connections are subject to surface pressure in the shaft and hub and shearing force on the pin.
- `Guide pin under bending force`
  Guide pin connections are subject to bending stress due to torque and to shearing force caused by transverse forces. The shearing force, surface pressure, bending of the pin, and surface pressure on the element. are calculated here.
- `Bolt connection subjected to shearing action (in double shear)`

The bolt is subject to bending and shear stress and to surface pressure in this arrangement. You can use different calculation methods, depending on the fit of the rod/bolt and fork/bolt connections. Experience shows that the limiting factor in sizing non-sliding surfaces is the bending stress and in sliding surfaces it is the surface pressure.

- `Bolts in a circular array (in single shear)`
  In this arrangement, the effective torque is distributed uniformly across the individual bolts/pins and therefore the shaft and hub are subject to surface pressure from the individual bolts/pins, creating a shearing force. The maximum shear stress and the minimum safety for the bolts are also output.

The load placed on the bolts, shaft and hub (or part) is calculated, and the safeties are determined, as described in classic technical literature (Niemann, Maschinenelemente I, 4th Edition 200564]),

apart from bolts in a circular layout.

The cross-sectional area and moment of resistance to bending in the spring dowel and coiled spring pins (bushes) is calculated in accordance with Decker [86]. In configurations in which the bolts, spring dowels and coiled spring pins (bushes) are only subjected to shearing, the permitted shearing force specified in the relevant DIN standard can be applied to the pins.

41.1 Influence factors

When calculating individual connections you must include a number of influencing factors which are defined depending on the load, stress, type, etc.:

- Application factor
- Dynamic factor:
  fixed load: Cd = 1; pulsating load: CD = 0.7; alternating load: CD = 0.5;
  for coiled spring pins and spiral pins (bushes)
  fixed load: Cd = 1; pulsating load: CD = 0.75; alternating load: CD = 0.375;
- Reduction factors for full/grooved dowel pin
  Full pin: CK = 1; grooved dowel pin (bending, shear): CK = 0.7; grooved dowel pin pressure: Ckp = 0.8;

Since the permissible stress values in the literature are very low, other material factors have been added to obtain the values in the table.

41.2 Materials

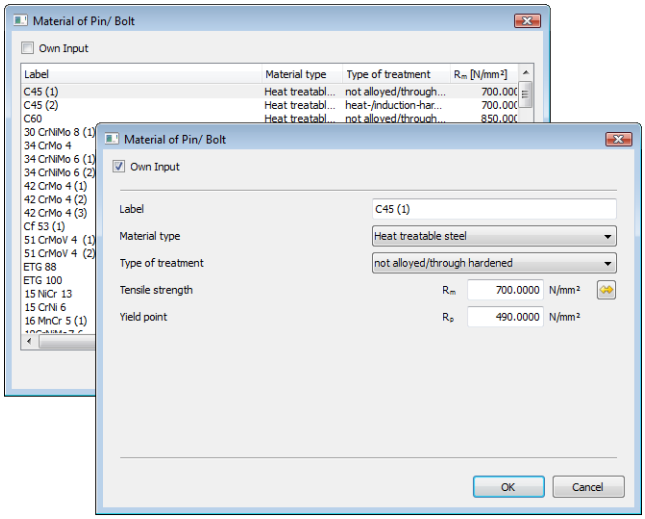


Figure 40.2: Materials screen: Bolts and Pins

In the selection list, you can select materials in accordance with the standard.
If you have set the `Own Input` flag, a new dialog appears here. This displays the material data used in the calculation which you can specify to suit your own purposes. You can also define your own materials directly in the database (see page I-114) so that these can also be used in future calculations.

41.3 Settings

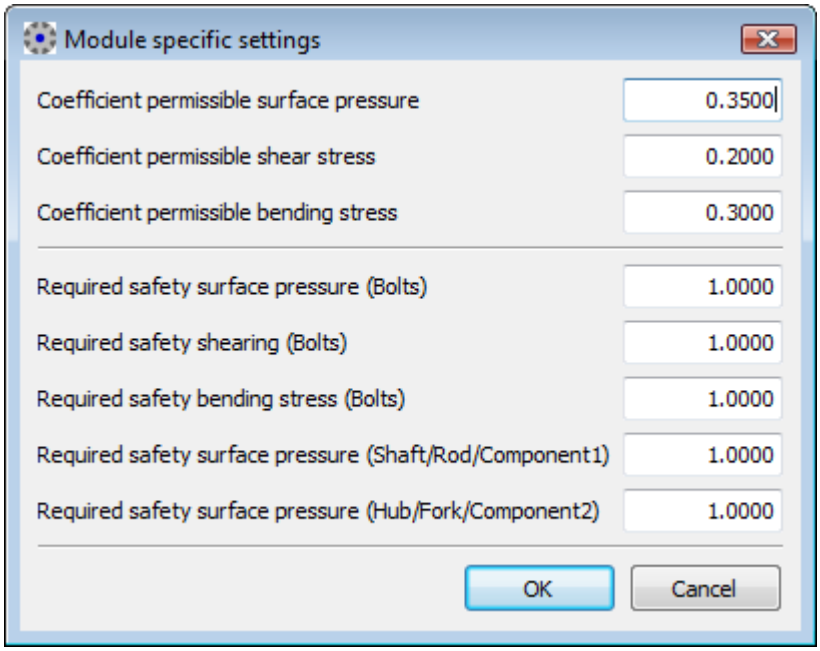


Figure 40.3: Settings: Bolts and Pins

In this sub window you can view and change the material factors and required safeties for each calculation.

This factor is multiplied by the tensile strength Rm for all parts/bolts and pins apart from coiled spring pins and spring dowel pins (bushes) to calculate the permitted value.

In the case of coiled spring pins and spring dowel pins (bushes), the permitted values are taken directly from the file and do not depend on tensile strength Rm.

41.4 Permitted values

Parts/Full pins/Bolts/Grooved dowel pins

For each part/bolt and pin, depending on the load, the coefficient you find under `Calculation/Settings` is multiplied with the tensile strength Rm to define the permitted value.

Coiled spring pins and spiral pins (bushes)

The permitted values for coiled spring pins and spiral pins (bushes), are imported from a file.

The permitted values for transverse force, for configurations that are only subject to shearing, can be taken from the relevant DIN standard for the pins.

The permitted values for shear and bending moment under different loads can be taken from the technical documentation provided by Decker:
Bending Stress: $\sigma b = 380 N/mm^2$
Shear stress: $\tau = 160 N/mm^2$
Surface pressure: $p = 208 N/mm^2$

Half the permitted values from other arrangements are used for the arrangement "Longitudinal pin under torque".

(Recommendation in accordance with Decker)

41.5 Sizings

Click the buttons next to Diameter and Load to size the values that are beside them to suit the required safeties.

Chapter 41
Bolts

KISSsoft calculates bolted joint in accordance with VDI 2230 (2003). The bolt calculation functions help you find your way through the maze of tables and data defined in the standard. In addition to providing tables with standard values, the program also has a range of options that allow you to enter your own definitions for most of the constraint values (such as geometry and material data). Although the VDI 2230 standard does not have iteration functionality, i.e. it can be calculated manually, the flexible input and modification options give you a user-friendly software solution at your fingertips. However, you must be familiar with VDI 2230 before you can interpret the results and enter the required values correctly in the program.

VDI 2230 compares the permissible assembly preload (F_M and also, to some extent, F_{Mzul}) with the minimum and maximum assembly preload (F_{Mmax} and F_{Mmin}). Here the first is a value calculated with 90% of the bolt yield point and the last two are determined by the loads required to guarantee that the connection functions correctly. Assembly preload F_{Mzul} is therefore determined from the strength of the bolt, while assembly preloads F_{Mmin} and F_{Mmax} are determined from the function of the connection. The necessary assembly preload F_{Mmin} is calculated from the axial force F_A and the resilience of the parts and the bolt ϕ, the embedding loss F_Z, the thermal forces $F_{V\,th}$ and the required clamp load F_{Kerf}. F_{Mmax} can be calculated from Φ_{Mmin} while taking into consideration the coefficient of friction scatter and the tightening technique (tightening factor a_A).

$$F_{M\max} = \alpha_A \cdot F_{M\min} \tag{41.1}$$

$$F_{M\max} = \alpha_A \cdot \left[F_{Kerf} + (1-\phi)\cdot F_A + F_Z + \Delta F_{vth}\right] \tag{41.2}$$

The necessary assembly preload F_{Mmax} must now be smaller than the sustainable pretension of the bolt F_{Mzul}. Similar to this comparison is the comparison between the minimum required mounting pretension force F_{Mmin} and the minimum pretension force achieved by tightening at, for instance, 90% of the yield point F_{Mzul}/α_A:

$$F_{M\max} \overset{!}{<} F_M \qquad F_{M\max} \overset{!}{<} \frac{F_M}{\alpha_A} \tag{41.3}$$

42.1 Special features in KISSsoft

In VDI 2230, the values for pretension force F_M when using 90% of the yield point and for the tightening torque are to be found in Tables 1 to 4. These values are rounded (rounding off error <= 1%). However, KISSsoft calculates the values using the equations on which these tables are based. The results are therefore more general than the ones that use the values given in the tables and therefore may also differ slightly from them.

When analyzing bolts as defined in VDI 2230, the safety factor which is usually required in other strength calculations is missing. Despite this, and to provide information about the suitability of a bolt's design for its purpose, a "utilization" has been generated as a result. The utilization in % shows the ratio of the required pretension force F_{MMax} to the effective possible preload F_M. The formulae used here are not specified in VDI 2230 and are therefore included when a KISSsoft log is printed out.

42.2 Basic data inputs

The entries you make in the Basic Data tab form part of the service and bolt data, and include the type of bolting, washers, and tightening technique.

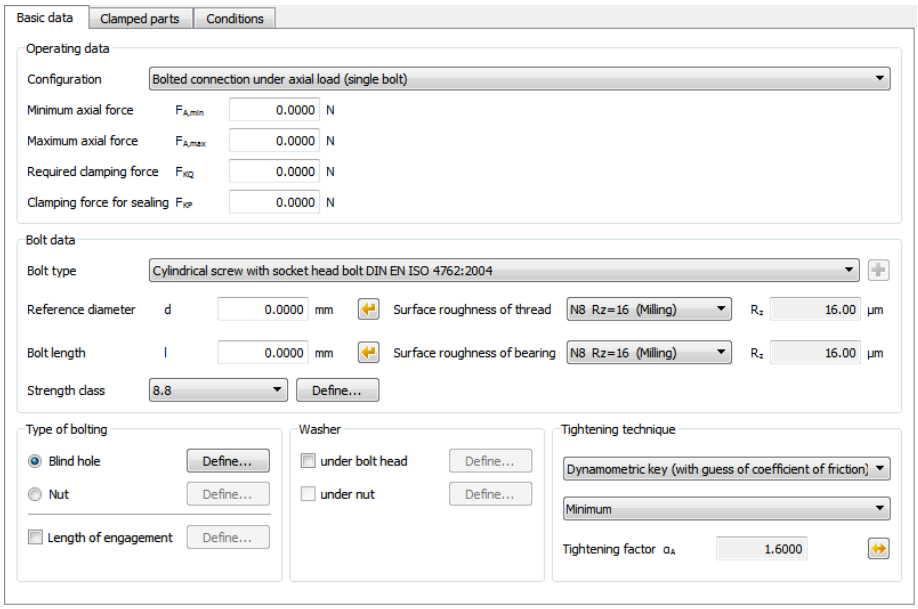


Figure 41.1: Basic data tab.

42.2.1 Operating data

You enter operating data in the `Basic data` tab. You can then use it for the following clamping configurations:

1. Bolted joint under axial load
2. Bolted joint under axial and shearing force
3. Flange connection with torque and forces
4. Multi-bolted joint with any bolt position
5. Proof for bolts with FEM results

An axial loading F_{Amax}, F_{Amin} and a required clamp load F_{Kerf} are determined each time, from the operating data.

External forces and torques, which must be transmitted via the bolted joint, are to be converted into axial force FA and into the required clamp load F_K. VDI 2230

assumes these values are known. In the KISSsoft system, you can input the appropriate configuration (bolt under shearing force and flange connection) as external forces and torques. These values are then used to calculate the axial load and preload on an individual bolt.

For a bolted connection under shear load, this shear load is represented by the friction between the bolted parts. The friction is determined by the coefficient of friction and the pretension force.

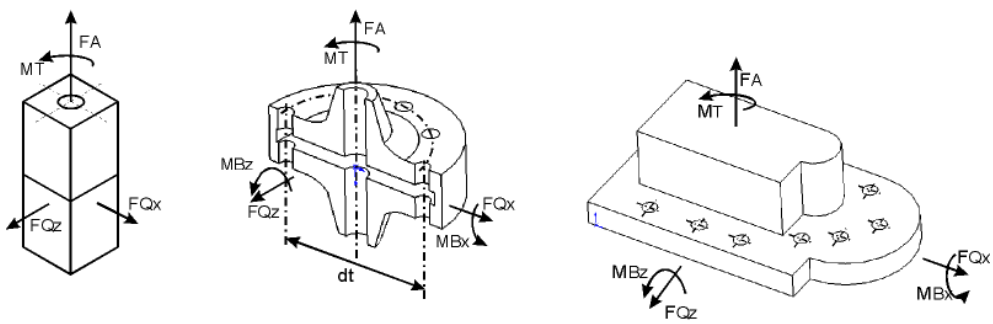


Figure 41.2: Bolting configurations: 1/2, 3 and 4

42.2.1.1 Bolted joint under axial and shearing force

In the second configuration, the required clamp load for axial load transmission is calculated from the shearing force F_Q, the torque M_T, the coefficient of friction m_T, the diameter d_a and the number of force transmitting parting lines q_T:

$$F_{KQ} = \frac{1}{\mu \cdot q_F} \cdot \left(F_Q + \frac{2000 \cdot M_T}{d} \right) \tag{41.5}$$

$$F_{Kerf} = \max\left(F_{KA} + F_{KD}, F_{KQ}\right) \tag{41.6}$$

F_{KQ} Required clamp load for transferring a shearing force and/or a torque through friction grip (e.g. for friction grip)

F_{KP} Clamp load required to guarantee a sealing function (required when internal pressure is present)

μ_T Interface coefficient of static friction (when shearing force or torques are present), → see Figure on page IV-909.

42.2.1.2 Bolted joint under axial load

The occurring axial forces F_{Amax} and F_{Amin} are entered directly. The necessary clamp load F_{Kerf} is defined in accordance with

$$F_{Kerf} = \max\left(F_{KA} + F_{KP}, F_{KQ}\right) \tag{41.4}$$

and the required clamp load for axial load transmission F_{KQ} and the sealing function F_{KP} are calculated. F_{KA} is present to prevent gaping in the required clamp load and is calculated by the program.

42.2.1.3 Flange connection with torque and forces

The forces on the single bolt in the case of flanged joints (with stress from torque and/or shearing force and/or bending moment and/or axial force) are calculated in accordance with [63], and also partially in accordance with [61], Example. 8.4:

$$F_{KQe} = \frac{1}{n \cdot \mu_T}\left(\frac{2000 \cdot M_T}{dt} + F_Q\right) \tag{41.7}$$

$$F_{KPe} = \frac{F_{KP}}{n} \tag{41.8}$$

$$F_{Kerf} = \max\left(F_{KA} + F_{KPe}, F_{KQe}\right) \tag{41.9}$$

$$F_{Bo} = \frac{1}{n}\left(F_{A\max} + \frac{4000 \cdot M_B}{dt}\right) \tag{41.10}$$

$$F_{Bu} = \frac{1}{n}\left(F_{A\min} - \frac{4000 \cdot M_B}{dt}\right) \tag{41.11}$$

dt Bolt pitch diameter

n	Number of bolts
μ_T	Coefficient of friction between the parts, see Figure 41.3
F_Q	Shearing force on configuration
F_{Amax}	Axial force on configuration (maximum)
F_{Amin}	Axial force on configuration (minimum) Resultant operating force on the bolt that is subject to the highest stress:
F_{Bo}	Upper limiting value
F_{Bu}	Lower limiting value
F_{KP}	Configuration sealing load
$F_{Kerf?}$	Required clamp load
M_B	Bending moment on configuration
M_T	Torque on configuration
F_{Kerf}	Required clamp load
F_{KQe}	Required clamp load (e.g. for friction grip)
F_{KPe}	Required clamp load to ensure sealing (for internal pressure)
F_{KA}	Required clamp load to prevent gaping under eccentric load

If you select a flanged joint configuration, we strongly recommend that you define the geometry of the stressed parts as `individual annulus segments`. The program then automatically generates a suggested value for the pitch (t_t) and the screw radius (t_{rs}).

Experience shows that the results of VDI 2230 are usually very conservative for flanged connections. In order to achieve realistic results, you should increase the coefficient of friction between the parts.

Material combination	Static friction coefficient in the state	
	dry	lubricated
Steel - steel/cast steel	0.1 to 0.23	0.07 to 0.12
Steel - gray cast iron	0.12 to 0.24	0.06 to 0.1
Gray cast iron - gray cast iron	0.15 to 0.3	0.2
Bronze - steel	0.12 to 0.28	0.18
Gray cast iron - bronze	0.28	0.15 to 0.2
Steel - cooper alloy	0.07	
Steel - aluminium alloy	0.1 to 0.28	0.05 to 0.18
Aluminium - aluminium	0.21	

Figure 41.3: Interface coefficients of static friction in accordance with [75]

42.2.1.4 Multi-bolted joint with any bolt position

In a multi-bolted joint you can define bolts in any position. These are then affected by shearing force, a bending moment in two directions, and a torsional moment. The load distribution on the bolts is calculated on the assumption that rigid plates are connected by springs at the bolt positions. Forces which do not affect the center of gravity must be moved to the center of gravity so they can be entered. By using a stiffness coefficient, you can model different bolt diameters (doubled diameter equals fourfold rigidity). Set the coefficient = 1 for the bolt with the smallest diameter. For larger bolts, use their diameter ratio to the smallest bolt as the coefficient. The forces are then distributed across the bolts. Only one bolt where the coefficient = 1 can be proved because otherwise the bolt data is incorrect.

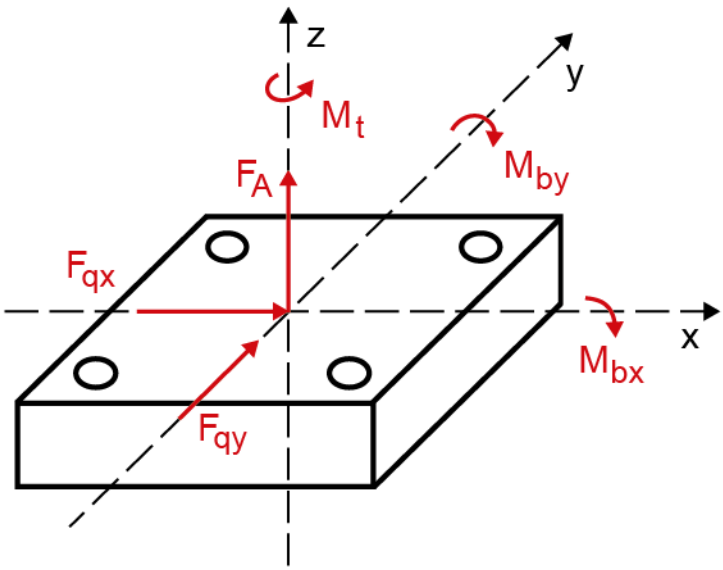


Figure: Definition of forces and torques in KISSsoft

Once you have entered the operating data in the `Basic data` tab, you can define the bolt positions in the `Position of bolt` tab. You can either enter the bolt positions in a table or import them from a file. The resulting axial forces, and also the clamp loads required to transmit shearing force, are also displayed in the table.

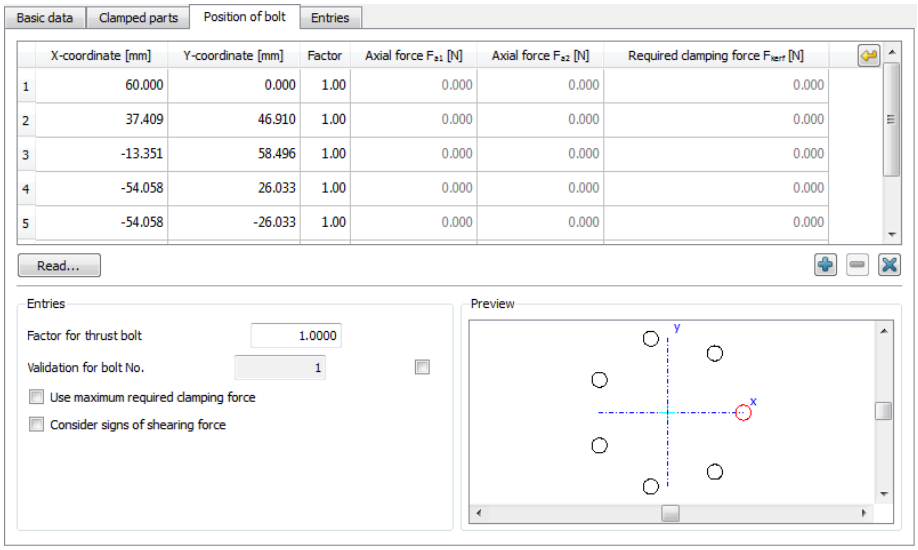


Figure 41.4: Position of bolt tab

Optionally, you can define an additional coefficient for set screws, in which it is assumed that compression is transmitted directly via the plates. However, use of this coefficient assumes you have expertise in this area. In [62], under the keyword Multi-bolted Plate Joint, for example, an average pressure point of 1/4 plate height is assumed. You use the coefficient for set screws to set this status.

The program then automatically selects the bolt with the highest axial force for the calculation. As a rule, and to ensure that the calculation results are on the safe side, the maximum required clamp load is used for all bolts. However, you can select a further option to deactivate this function.

When you calculate the necessary clamp load, you can also take the prefix of the shearing force into account. Shearing forces caused by torsion and shearing force are then added at specific points and subtracted at other points. You should only include the prefix if you know the direction of the shearing force and if this force is constant.

To save you having to specify the arrangements in which the incremental distances between the bolts repeat themselves, for every single bolt, you can define the position of particular bolts in the Sizing function. Click the button in the table (above, on the right) in the "Position of bolt" tab to open this window, in which you can enter different configurations.

You can enter these values here:

- line (values for: starting point, end point, number of bolts)
- circle (values for: center point, radius, number of bolts)
- circle segment (values for: radius, starting angle, end angle, number of bolts)

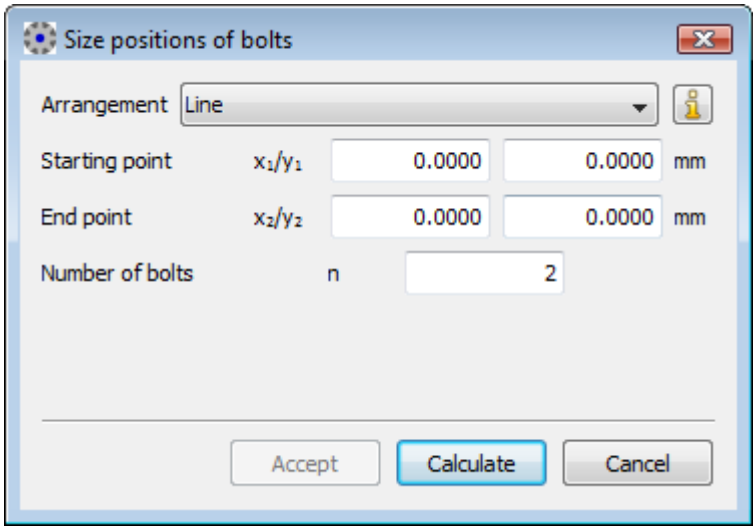


Figure 41.5: Sizing the position of a bolt

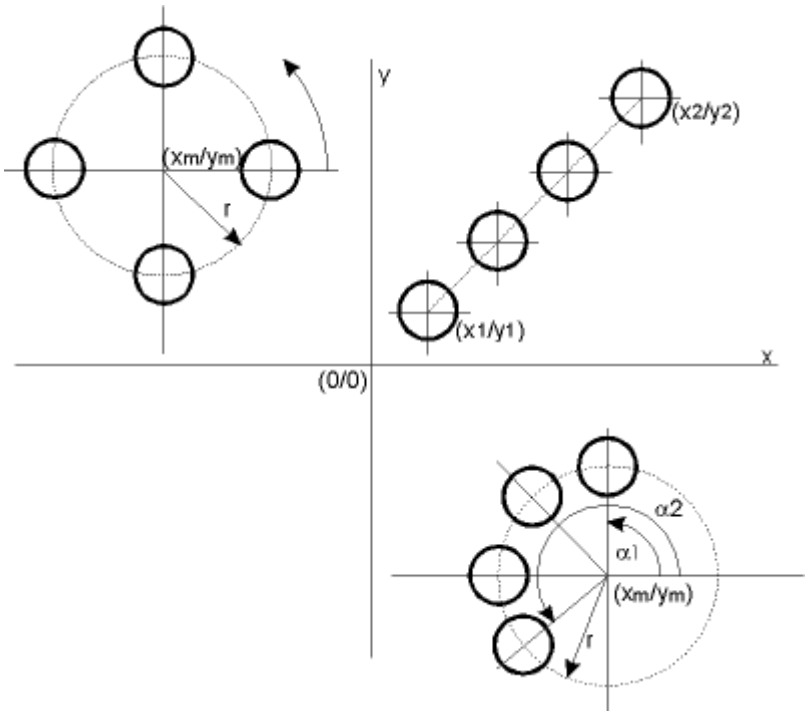


Figure 41.6: Position sizing options

You can add positions by transferring existing positions. However, if you only want to use positions shown in the configuration, you must delete all the others.

42.2.1.5 Results from FEM calculation

The following behavior can be taken into account in an FEM calculation (instead of calculation in accordance with VDI 2230 Sheet 1):

- Non linear material behavior (for example, plastified or yield)
- Non linear boundary conditions (for example, load dependency of the contact surfaces),
- Geometric non linear behavior (for example, significant deformation occurs when the tightening process is simulated)

As specified in VDI2230 Sheet 2, Multi-bolt joints (draft version), the FEM modeling is subdivided into 4 model classes:

Model class I:

The bore is not represented, the interface is either completely integrated or connected by a rigid coupling and a reference node for every bolt.

Model class II:

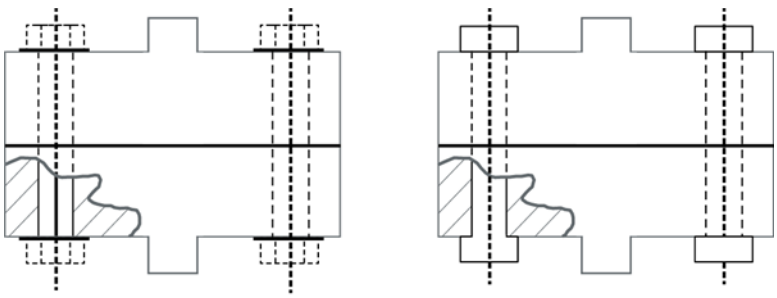
The bolt is represented as a bar or a spring element (with the ideal translational and rotational degrees of freedom for each section) Interface with contact definition.

Model class III:

The bolt is shown in 3D as a cylinder with large cylinders at its ends. The interface and support areas are assigned contact definitions in the FEM calculation. The cylinder has the same core diameter as the thread.

Model class IV:

The bolt is modeled with the exact thread geometry. The assembly and operating states are treated separately.

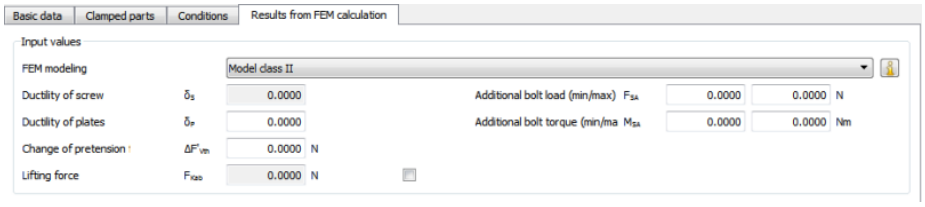


Modellklasse / model class II Modellklasse / model class III

Using VDI 2230 Sheet 1 to prove FEM results is really only a sensible idea for model classes II and III. Model class I does not supply enough informative results to create a proof in accordance with VDI 2230 Sheet 1. If you are using model

class IV, you already have all the necessary results and should use a different method to the one in VDI 2230 Sheet 1 for the proof.

Implementation in KISSsoft – VDI 2230 Sheet 2 (draft version) FEM results



You can select these model classes directly from the list. The main difference between the models is that bolt resilience can only be defined in model class III.

The integration of FEM results in the calculation in accordance with VDI 2230 Sheet 1, so that a proof can be performed, is implemented as follows in KISSsoft (sub-divided as given in VDI 2230 Sheet 1):

R0, R1	in accordance with VDI 2230 Sheet 1
R2	You can input the clamp force for the sealing function FKP directly in the basic data, if it has already been determined in the FEM calculation. The lifting force can be defined directly in FEM and specified here. If the results have not been defined, the program determines them in accordance with VDI 2230 Sheet 1. The axial load transmission FKQ is determined as specified in VDI 2230 Sheet 1.
R3	In model classes II and III, the resilience of the plates is taken from the results of the FEM calculation. In model class III, the bolt resilience is also taken from the results of the FEM calculation. In model class II, the bolt resilience is derived as specified in VDI 2230 Sheet 1. The amounts of embedding are estimated in accordance with VDI 2230 Sheet 1.
R4	Amounts of embedding defined as specified in VDI 2230 Sheet 1, Fv’th can be input directly as result from FEM.
R5	Fv’th can be input directly as result from FEM, Fkerf, φen, Famax, Fz from VDI 2230 Sheet 1 - >determination of FMmin.
R6	Fmmax = αA*FMmin.
R7	In KISSsoft, assembly stress is calculated in accordance with VDI 2230 Sheet 1. This would take an exceptional amount of time and effort if FEM were used instead.
R8	FV'th is included in the calculation of Fsmax (total bolt load). If FV'th > 0, it is set to 0, as specified in R8/1, VDI 2230 Sheet 1. All others σz, τmax, σredB are calculated in accordance with VDI 2230

	Sheet 1.
R9	FSA, MSA as input from the FEM results. You can specify the upper and lower limit. Proof of dynamic strength in accordance with VDI 2230 Sheet 1 (R9/2). The permitted values are defined in accordance with VDI 2230 Sheet 1. The effects of temperature are included directly in the FEM results.
R10	pBmax can be derived from the FEM results if model class III is being used, otherwise it is calculated in accordance with VDI 2230 Sheet 1. You should only calculate the values for permissible surface pressure pGmax directly in FEM if these values are not already available.
R11	The minimum length of engagement cannot be represented realistically in model classes I to III. It would take a great deal of time and effort to model this in model class IV. The calculation is performed in accordance with VDI 2230 Sheet 1.
R12	The calculation In KISSsoft is performed in accordance with VDI 2230 Sheet 1. The values you need to input here are determined from the FEM calculation.
R13	not applied.

42.2.2 Bolt data

The type, geometry, surface roughness and strength class of a bolt can all be defined as bolt data.

`Bolt type:` You can use the following standard bolt descriptions from the database to define the bolt type:

DIN EN ISO 4762/ DIN 912	Hexagon socket head cap screw Standard thread M1.6 to M64
DIN 7984	Hexagon socket head cap screw with low head Standard thread M3.0 to M24.0
DIN EN ISO 4014/ DIN EN 24014	Hexagon headed bolts with shank (formerly DIN 931 T1) Standard thread M1.6 to M64
DIN EN ISO 4017/ DIN EN 24017	Hexagon headed bolt with thread to head (formerly DIN 933) Standard thread M1.6 to M64

DIN EN ISO 1207/ DIN 84	Slotted cheese head screw Standard thread M1.0 to M10
DIN EN ISO 8765	Hexagon headed bolt with shank Fine thread M8.0 to M64
DIN EN ISO 8676	Hexagon headed bolt without shank Fine thread M8.0 to M64
DIN EN 1662	Hexagon headed bolt with flange, light series form F Standard thread M5.0 to M16
DIN EN 1662	Hexagon headed bolt with flange, light series form U Standard thread M5.0 to M16
DIN EN 1665	Hexagon headed bolt with flange, heavy series form F Standard thread M5.0 to M20
DIN EN 1665	Hexagon headed bolt with flange, heavy series form U Standard thread M5.0 to M20
ASME B18.2.1	Square bolts, UNC thread, 0.25 to 1.5in
ASME B18.2.1	Hex bolts, UNC thread, 0.25 to 4in
ASME B18.2.1	Heavy hex bolts, UNC thread, 0.5 to 3 in
ASME B18.2.1	Hex cap bolts, UNC thread, 0.25 to 3 in
ASME B18.2.1	Heavy hex bolts, UNC thread, 0.5 to 3 in

`Reference diameter:` You can either input your own value for the reference diameter or click the button to calculate an approximate value after you input the operating data. This sizing function usually produces bolt diameters that are too large. We therefore recommend you input a value that is 1 or 2 standard sizes less than the system's proposed value.

`Bolt length:` You can input any bolt length if you are inputting your own bolt geometry. Otherwise, after you input the bolt length, the system sets it to the next standard length.

`Surface roughness of thread/bearing surface:` The surface roughnesses influence the amount of embedding and therefore also the preload loss of the bolted joint.

`Strength class:` After the entry for standard strength grades you can click the button to define your own strength values.

`Own definition of bolt geometry:` To define your own bolt geometry, you must set the `Bolt type` selection list to `Own input`. This

activates the `Define...` button which you can click to input your own values for bolt geometry.

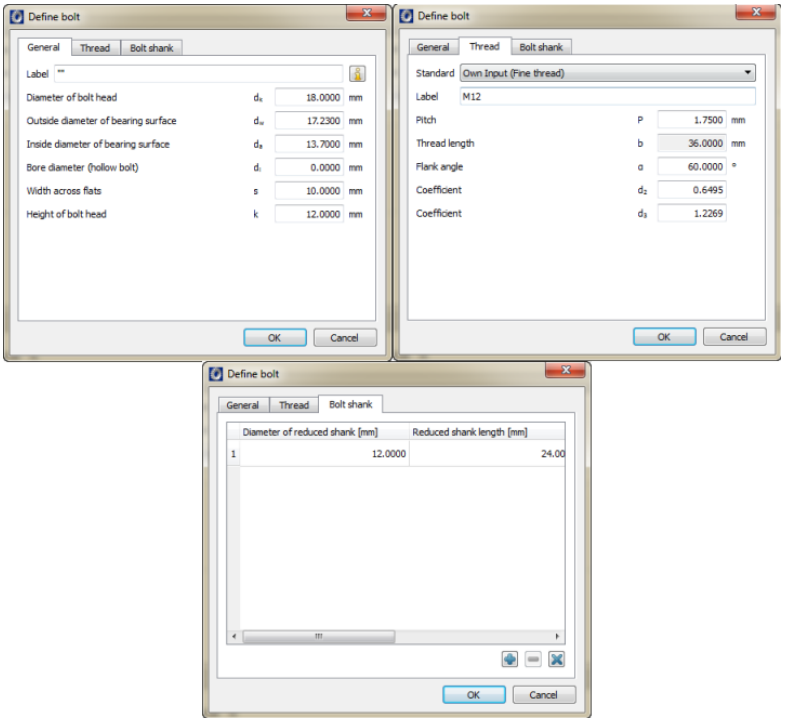


Figure 41.7: Dialog with three tabs for defining your own bolt geometry.

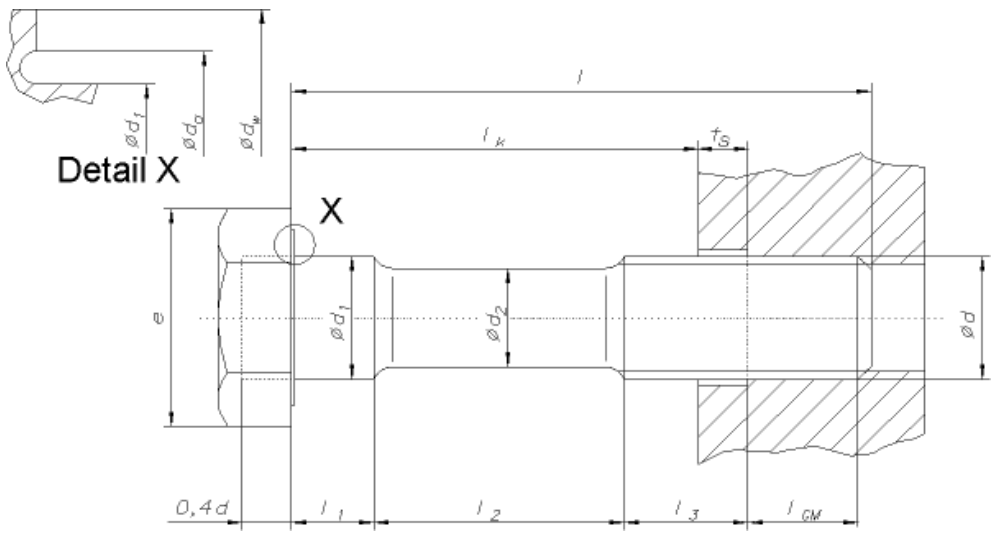


Figure 41.8: Bolt geometry

- **General:** tab Input the dimensions of the bolt head, as well as the bore diameter if you are using a bolt with a bore.
- **Thread:** tab Data from the standard, the size of the thread, the lead, and the thread length. This is where you define the factors used to calculate the flank diameter d_2 and the core diameter d_3 (d_2 = d - $d_2$factor*P; d_3 = d - $d_3$factor*P).
- **Bolt shank:** tab Data about the individual bolt cross sections. The button adds a new cross section, and the button removes the selected one. Click the button to delete all the cross sections.

42.2.3 Type of bolted joint

To define the type of bolting, enable either `Nut` or `Blind hole`. This corresponds to the difference between Through-bolt and Single-bolted, (tapped thread) connections as defined in VDI. Click on the appropriate `Define...` button to open the corresponding input dialog for additional data about the nut or the threaded part.

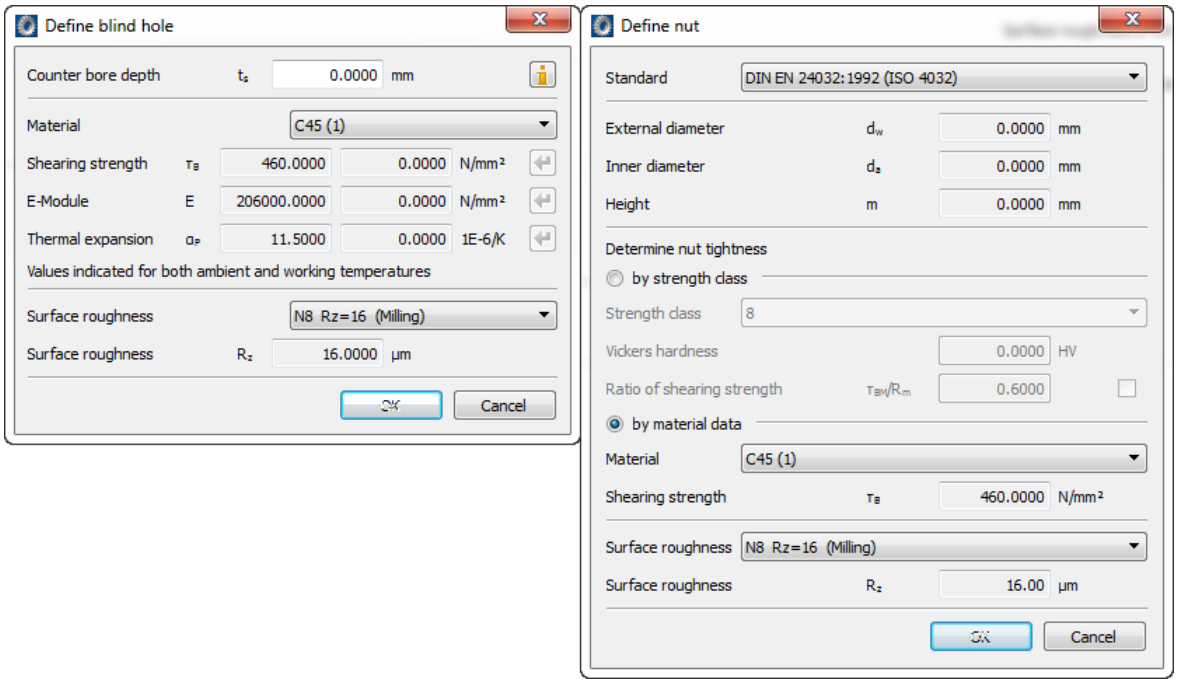


Figure 41.9: Input dialog for data about cut threads and nuts

Blind hole

For cut threads, the counter bore depth t_s describes milling without thread that is primarily used to extend the clamp length (see also Figure on page IV-917).

Nut

In the nut definition screen you can either select a standard for the geometry or define one yourself.

For example, when calculating the length of engagement, you can either define the hardness from the strength class (as specified in DIN EN ISO 898-2) or define the shearing strength directly from the material.

The "Own Input" option is also available in both variants. However, when you input the strength class you must also define the ratio of shearing strength to tensile strength (tauBM/Rm).

The system then converts the hardness value you enter here into tensile strength as part of the hardness conversion process. The tensile strength Rm is then multiplied with the ratio (tauBM/Rm) to calculate the shearing strength (tauBM).

The minimum hardness value for nuts with a standard thread (including UNC) is taken from the strength class in Table 6 in DIN EN ISO 898-2. The minimum hardness value for fine threaded nuts (including UNF) is taken from the strength class in Table 7 of the same standard.

NOTE

If the dimension of the interface area DA is only slightly larger than the bearing diameter of the bolt head dw, it must be calculated as a through-bolt connection (note the deformation cone).

(DA to ~1.4*dw)

42.2.4 Washers

If this flag is set, a washer is inserted between the nut and the part and/or the head and part.

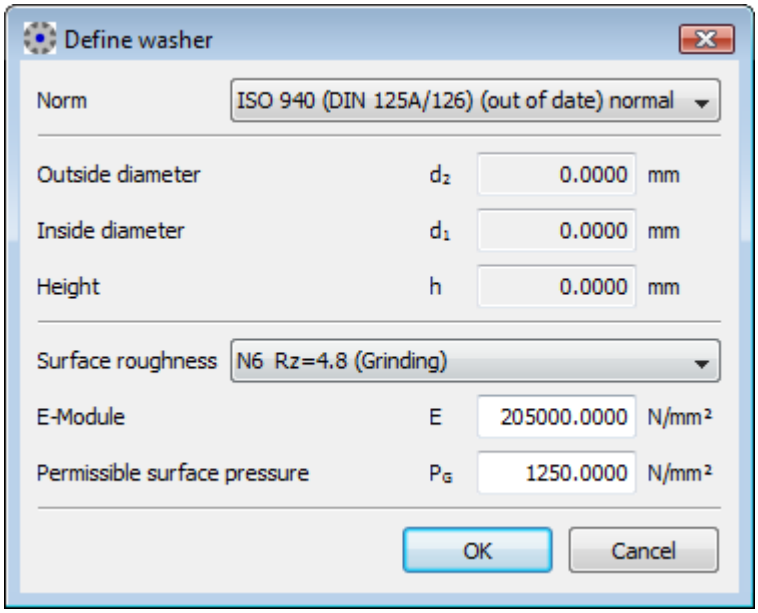


Figure 41.10: Defining washers.

In the `Calculation/Settings` tab, if you select the "Determine specific thermal expansion of washers" flag, you can also define the thermal expansion coefficients that are used to calculate the difference in pretension force. You will find a more detailed description in the "Settings" section.

42.2.5 Tightening technique

Uncertainties such as, for example, the scatter of the friction factors, tightening techniques that differ in their precision, and instrument, operating and reading errors, result in variation in the achievable assembly pretension force. For this reason, oversizing the bolt is necessary, and is expressed by the tightening factor $\alpha_A = F_{Mmax}/F_{Mmin}$. If the required minimum preload Φ_{Mmin} remains constant, then an increasing tightening factor a_A means that the bolt must be sized for a larger maximum assembly preload F_{Mmax} (due to the greater scatter). Tightening technique and associated tightening factors:

Tightening factor α_A	**Tightening technique**	**Adjusting technique**
1.0	Yield point-determined tightening mechanically or	

	manually	
1.0	Angle of rotation-controlled tightening mechanically or manually	Experimental determination of the preload moment and angle of rotation
1.2 to 1.6	Hydraulic tightening	Adjustment by means of measuring length or pressure
1.4 to 1.6	Torque-controlled tightening with a torque wrench, torque indicating wrench or a precision torque wrench with dynamic torque measurement	Experimental determination of the required tightening torques on the original bolting part, e.g. by measuring the length of the bolt
1.6 to 1.8	ditto	Defining the nominal tightening torque by estimating the friction factor (surface and lubrication ratios)
1.7 to 2.5	Torque-controlled tightening with a bolt installation spindle	Torque wrench adjustment with a tightening-up moment, set to the required tightening moment (for an estimated friction factor) plus a supplement.
2.5 to 4	Pulse controlled tightening with an impact wrench	Torque wrench adjustment with tightening-up moment as described above

42.3 Clamped parts inputs

The `Clamped parts` screen displays data about the materials and geometry of the clamped parts, the distances involved for eccentric load/clamping, and data about the load application factor.

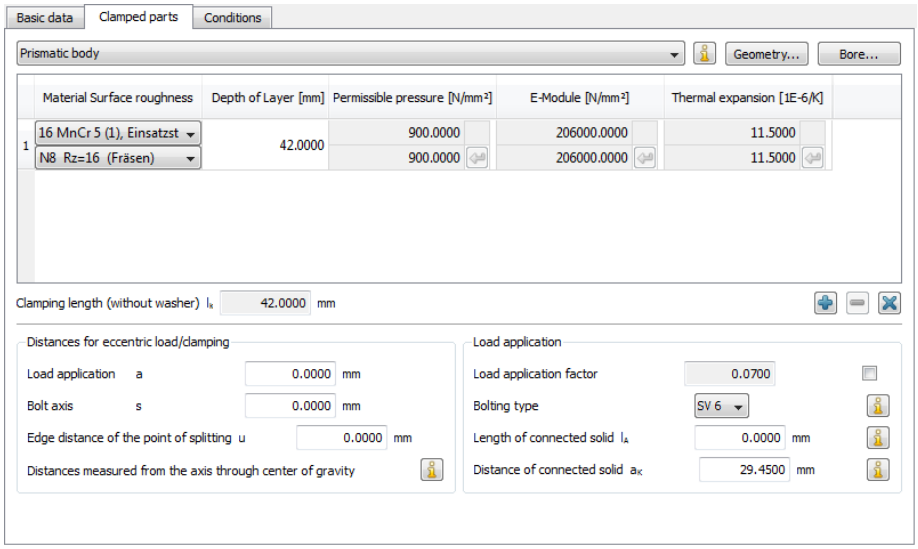


Figure 41.11: Clamped parts inputs tab

42.3.1 Geometry of clamped parts

There are several basic types of clamped parts:

- Plates
- Cylinder
- Prismatic body
- Annulus segment

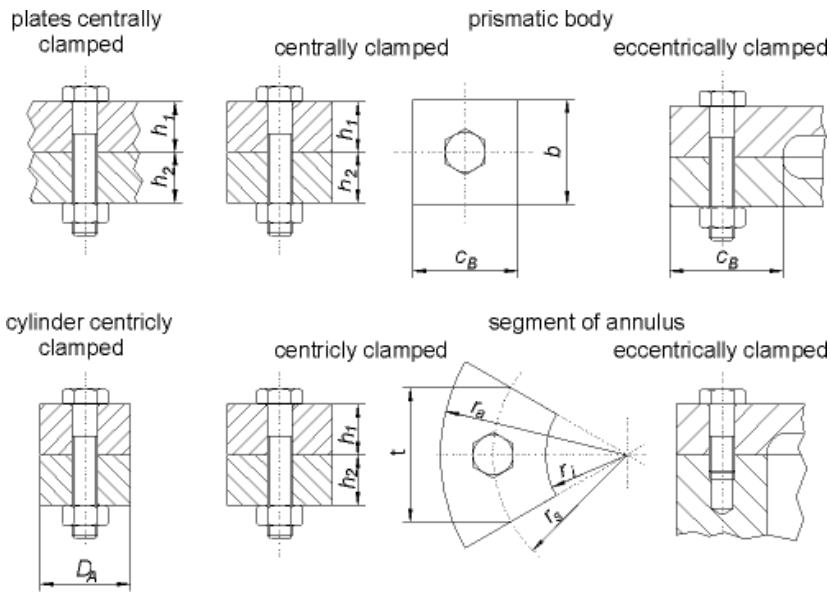


Figure 41.12: Clamped parts.

If you select Plates, it is assumed that the clamping deformation cone will be able to expand sidewise freely. For all the other selection options, click the `Geometry` button to enter the type of clamped part you want to use in the calculation.

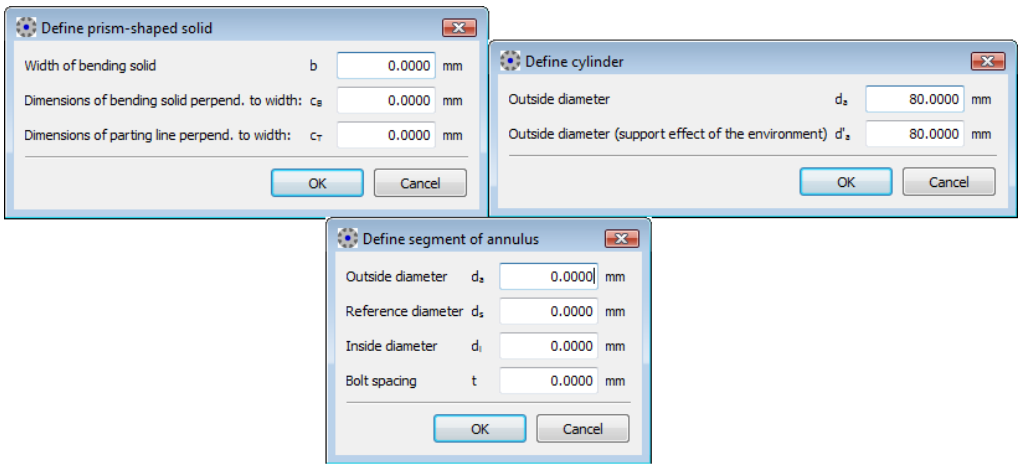


Figure 41.13: Geometry inputs for the cylinder, prismatic body, and annulus segment.

Click the `Bore` button to define a threadless through-bore in the part. You can also define chamfers at the head and or nut here. These chamfers are then included when the bearing areas are calculated. The chamfer reduces the outside radius of the bearing area and therefore increases the surface pressure.

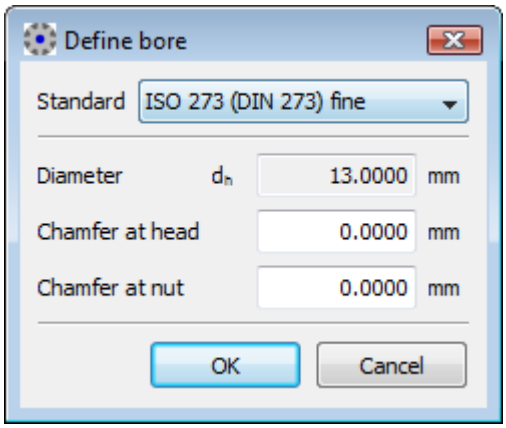


Figure 41.14: Defining through-bores and chamfers under head and nut.

You simply enter the different material situations in the list. The upper values for permissible pressure, e-module and thermal expansion are material values that apply to room temperature and, unless they are values you have entered, are always shown with a gray background. If the "Calculate temperature dependent material data automatically with estimation formulae" flag is set in `Calculations/Settings`, the values for the operating temperature are calculated empirically and displayed in the lower half of the particular material. You cannot edit these values. If the flag is not set, you must input your own values. Click the buttons to call the particular empirical formulae so they can be applied in the calculation. Click the button to add a material and the button to delete the selected element. Click the button to delete all the elements. The calculated clamp length is displayed in the l_k field.

42.3.2 Distances for eccentric clamping/load

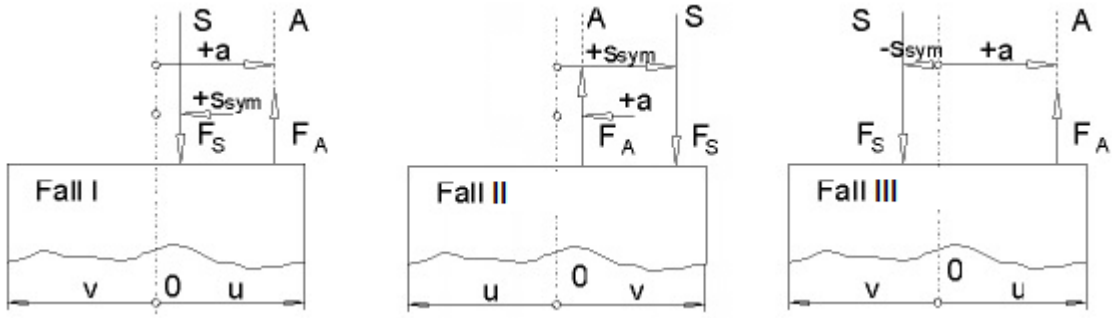


Figure 41.15: Possible load cases in the case of eccentric clamping.

As you can see in Figure 41.15, the axis of the center of gravity of the clamping solid 0 - 0 determines the null point (origin) of the X-axis. The distance between load line of action A - A and the center of gravity axis 0 - 0 is always positive. The distance s between bolt axis S - S and center of gravity axis 0 - 0 is set as positive, if the bolt axis S - S and the load line of action A - A lie on the same side as the center of gravity axis 0 - 0. If not, this value is negative.

The dimension u defines the distance of the center of gravity axis 0 - 0 to the point at which gaping first occurs. In Figure 41.15 this is the distance to the right-hand side in cases I and III, but the distance to the left-hand side in case II. In cases I and II, u must be positive, and in case II it must be negative. The guidelines governing the use of prefixes specified in VDI 2230 Sheet 1 are applied here.

42.3.3 Load application

The VDI guideline issued in 2003 defines equations for calculating the load application factor. Here, you must select a configuration in accordance with Figure 41.13. The interface must lie within the range shown in gray. The length of the clamped parts h, the distance to the connection piece $a_k$and the length of the connected solid l_A as shown in Figure 41.14 define the position of the application of load point and therefore also the load application factor.

In single-bolted, (tapped thread) connections, only configurations SV1, SV2 and SV4 are available. You must use the height h_{ESV} up to the interface as the height h.

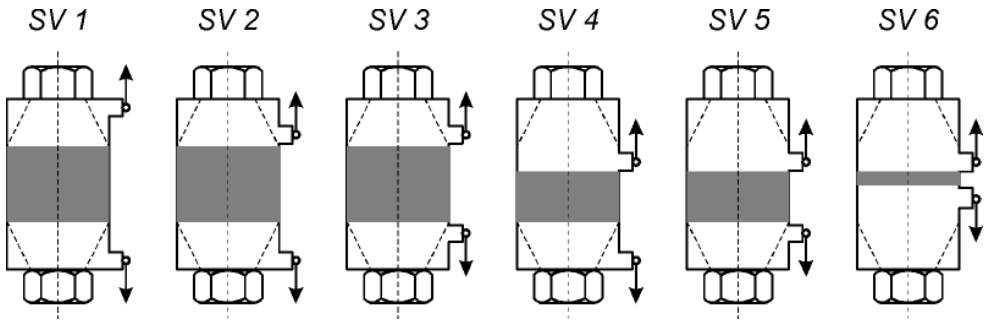


Figure 41.16: Configurations for defining the load application factor as shown in VDI 2230 (2003 edition).

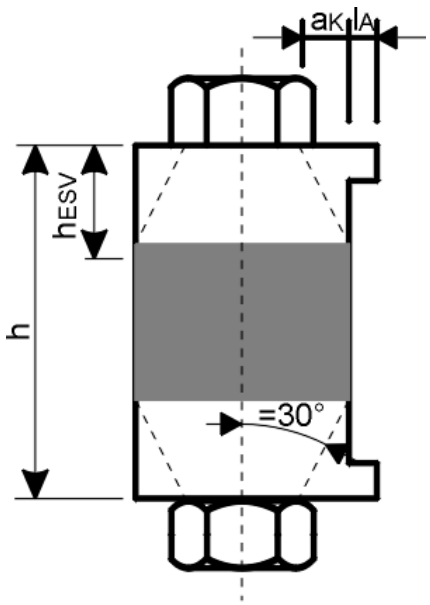


Figure 41.17: Inputs for defining the load application factor as shown in VDI 2230 (2003 edition).

42.4 Constraints data

In this calculation you can define the utilization of the yield point, the maximum assembly preload, the maximum tightening torques, or both tightening torques, as constraints. If you define the maximum and minimum tightening torque as constraints, the tightening factor is then calculated from this torque variation and the friction coefficient variation. You can also enter values for the number of load cycles, amount of embedding, preload loss, and temperatures for the bolted joint in this window.

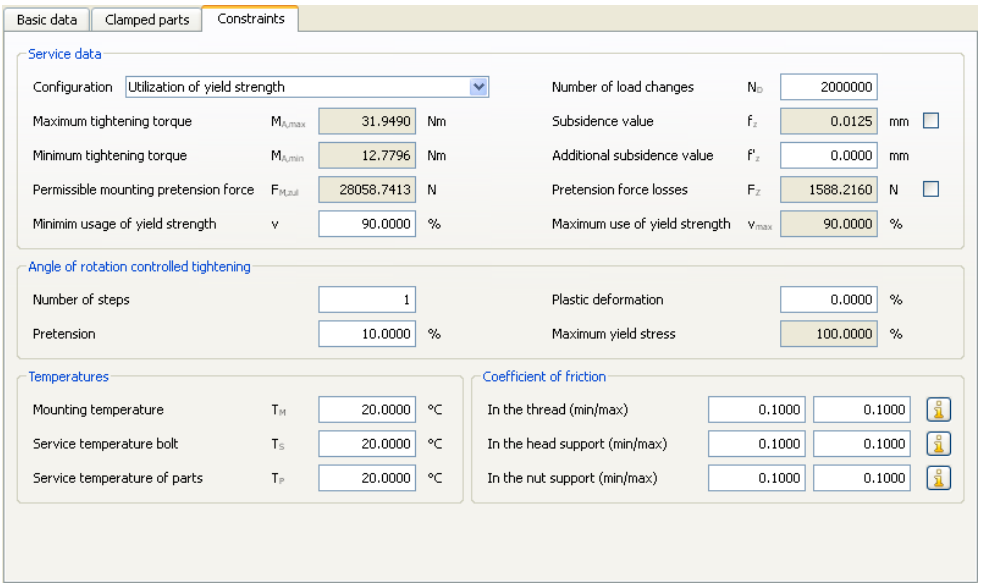


Figure 41.18: Preset values inputs tab

Use of the yield point

In usual bolt layouts, the bolt is tightened to 90% of its yield point to calculate the preload force. However, if you use yield point or angle-of-rotation controlled tightening, you can increase this utilization value up to 100%.

Number of load cycles

If this number of load cycles is ND >= $2*10^6$ the fatigue lives for final heat treated and final rolled bolts are calculated in accordance with the formulae specified in VDI (VDI 2230, 5.5/20 and 5.5/21). If smaller values are involved (ND < $2*10^{6)}$, a limited life fatigue strength sizing is performed for the connection (5.5/22 and 5.5/23).

Amount of embedding

The amount of embedding is calculated according to which calculation method you use. You can also input an extra amount of embedding value for flat seals. In

addition, you can overwrite the calculated amount of embedding with your own value or input the preload loss directly. If you input your own preload loss, the amount of embedding is no longer taken into account.

Assembly and operating temperature
Now KISSsoft's bolt calculation function has been extended, it can be used for the calculation guideline specified in VDI 2230, which also calculates bolted joints for operating temperatures between -200 and +1000 degrees Celsius. You can specify different temperatures for the bolt and the clamped parts. You can also take into account the temperature-dependent changes in the Young's modulus, the thermal expansion coefficients, the yield point, and the pressures permitted for the materials. You can either use empirical formulae to calculate these temperature-dependent values or specify your own values. Since the empirical formulae for commonly used steels have already been determined, you should check the values for high temperature changes or, even better, enter your own values here.

All the criteria for the bolted joint are checked in the assembled state at ambient temperature as well as in a stationary or non-stationary state at operating temperature (in accordance with VDI 2230: preload, bolt load, fatigue life and surface pressure).

KISSsoft automatically performs the calculation for assembly and operating temperatures at the same time. This calculation should also be performed for a higher temperature difference between the bolt and the parts. The minimum temperature difference between the parts or the bolt and the assembly temperature must equal at least 30° C so that results appear in the report.

42.4.1 Technical Explanations

The critical influences of temperature on the operating properties of bolts are:

- Change in pretension force due to thermal expansion
- Change in pretension force due to relaxation (at high temperatures)
- Brittleness (at high and low temperatures)

The lack of sufficient general data for materials (bolt materials and clamped parts) means the number of calculation options is also limited. The change in pretension force due to thermal expansion can be calculated very accurately because, as a first approximation, the thermal expansion value can be viewed as linear (with the temperature) (in a temperature range: from -100 to +500°C). The other effects (relaxation and brittleness) can be minimized by selecting appropriate materials and taking precautionary measures (see the relevant literature).

The calculation of the change in pretension force due to thermal expansion is performed as specified by H. Wiegand in "Schraubenverbindungen, 4th Edition

1988, section 7.1.3.1 (with temperature-dependent thermal expansion value and Young's modulus). All other calculations are based on the equations in VDI 2230 with the appropriate values at operating temperature.

KISSsoft suggests sensible values for much of the data you can input (Young's modulus, thermal expansion value, yield point at operating temperature) which are based on current technical literature (DIN standards, technical documentation from the company Bosshard, in Zug, Switzerland). These suggestions are based on the Young's modulus for ambient temperature and, of course, also on the operating temperature. When calculating the suggestion for the permissible pressure at operating temperature, the proportional change to the yield point was assumed. The suggestions are average values for "commonly used steels". They do not refer to one specific material and must therefore be checked carefully in critical situations because the influence of temperature also varies according to the type of material involved. If you want to calculate material data automatically using empirical formulae, simply click on the `Calculation>Settings` tab.

42.4.2 Coefficients of friction

In KISSsoft you can specify an interval for friction coefficients. The minimum value is used for calculation with F_M, F_{Mmax} and the maximum value is used for calculation with F_{Mmin} and F_M/α_A. The maximum value therefore affects the scatter of the tightening torques.

μ_G	Threac			Outside Thread										
Thread	Material			Steel									A2	
	Surface			black heat-/ phosphate treated				electro-galvanised (Zn8)		cadmium-galvanised (Cd6)		Adhesive		
			Kind of Thread	rolled			cut	cut or rolled						
Thread	Material	Surface	Kind of Thread / Lubrication	dry	oiled	MoS_2	oiled	dry	oiled	dry	oiled	dry	dry	oiled
Inside Thread	Steel	bare	cut / dry	0,12	0,10	0,08	0,10	-	0,10	-	0,08	0,16	-	-
		electro-galvanised		0,10	-	-	-	0,12	0,10	-	-	0,14	-	-
		cadmium-galvanised		0,08	-	-	-	-	-	0,12	0,12	-	-	-
	GG/GTS	bare		-	0,10	-	0,10	-	0,10	-	0,08	-	-	-
	AlMg	bare		-	0,08	-	-	-	-	-	-	-	0,32	0,28
	A2			-	-	-	-	-	-	-	-	-	0,26	0,12

Figure 41.19: Friction coefficients in the thread.

Auflagefläche				Auflagefläche	Schraubenkopf / Mutter										
				Werkstoff	Stahl										A2
				Oberfläche	schwarz oder phosphatiert						galvanisch verzinkt (Zn6)		galvanisch cadmiert (Cd6)		
				Fertigung	gepresst			gedreht		geschliffen	gepresst				
Auflagefläche	Werkstoff	Oberfläche	Fertigung	Schmiedung	trocken	geölt	MoS_2	geölt	MoS_2	geölt	trocken	geölt	trocken	geölt	trocken	geölt
Gegenlage	Stahl	blank	geschliffen	trocken	-	0,16	-	0,10	-	0,16	0,10	-	0,08	-	-	-
			spanend bearbeitet		0,12	0,10	0,08	0,10	0,08	-	0,10	0,10	0,08	0,08	-	-
		galvanisch verzinkt			0,10	0,10	-	0,10	-	0,10	0,16	0,10	-	-	-	-
					0,08	0,08	0,08	0,08	0,08	0,08	-	-	0,12	0,12	-	-
	GG/GTS	blank	geschliffen		-	0,10	-	-	-	0,10	0,10	0,10	0,08	-	-	-
			spanend bearbeitet		-	0,14	-	0,10	-	0,14	0,10	0,10	0,08	-	-	-
	AlMg				-	0,08	0,08	0,08	0,08	-	-	-	-	-	0,35	0,08
	A2				-	-	-	-	-	-	-	-	-	-	0,08	0,08

Figure 41.20: Friction coefficients in head bearing area and nut bearing area.

You can also use the sizing according to friction classes A to E as specified in VDI 2230 Sheet 1, Appendix to Table A5 to define the values for the coefficients of friction. The minimum and maximum coefficients of friction for the thread, the bearing surface and the nut support are then imported into KISSsoft.

42.4.3 Swing angle controlled tightening

For the angle of rotation-controlled tightening, the report displays a preload torque and an angle of rotation split into a number of steps. Here you can enter the value for this preload torque and the number of steps. The angle of rotation is then calculated with the medium assembly preload force $(F_M + F_M/\alpha_A)/2$. If the yield point utilization is 100%, this force is applied up to the yield point. To calculate the tightening angle of rotation you can also enter the required plastic strain of the weakest cross section.

42.5 Stripping strength

Click on the `Length of engagement` flag to calculate the length of engagement and check the stripping strength of the thread according to VDI 2230 Chapter 5.

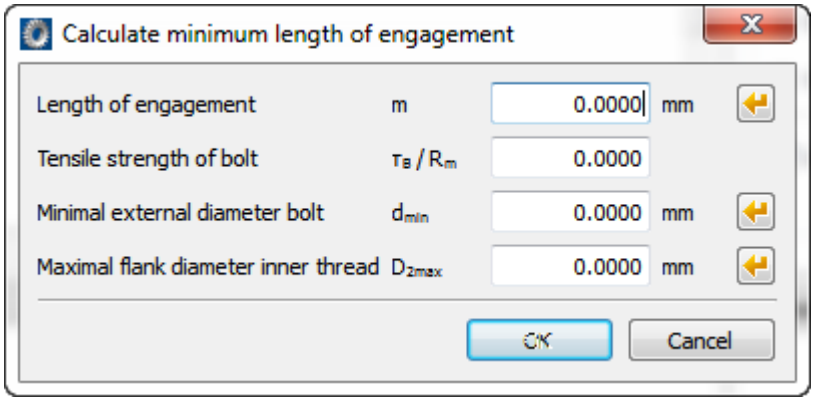


Figure 41.21: Entries for calculating the length of engagement and stripping strength

This is where you input data for the length of engagement, tensile strength of bolt, and the ratio of shearing strength/bolt. When the sizings are displayed they contain data from the bolt calculation. You can still change these values.

The length of engagement meffmin is calculated with the tensile strength Rm of the bolt material (theoretical), and the length of engagement meffmax is calculated with the worst case (with Rmmax, dmin and D2max according to VDI 2230 5.5/49), with the effect that the bolt would break before the thread is stripped (if the nut strength is relatively high).

The default value for the Rmmax/Rm coefficient is 1.2 (also used in VDI 2230 Figure 5.5/4, according to Prof. Dr. Ing. W. Lori, Zwickau, Germany).

You can change the Rmmax/Rm coefficient in `Calculation > Settings`.

To calculate the worst case (VDI 2230, formula 5.5/49), you must also take the thread tolerance into account. To define this influence, you can enter the minimum external diameter dmin of the bolt and the maximum flank diameter D2max of the internal thread in this window.

For a blind hole connection you should add 2*P to the minimum length of engagement because the first two threads on the bolt are not fully cut as specified in the standard.

The main report lists the stresses, the minimum length of engagement and the safety against shearing under load with the maximum preload force for the connection.

42.6 Settings

In `Calculations > Settings` you can enable additional settings:

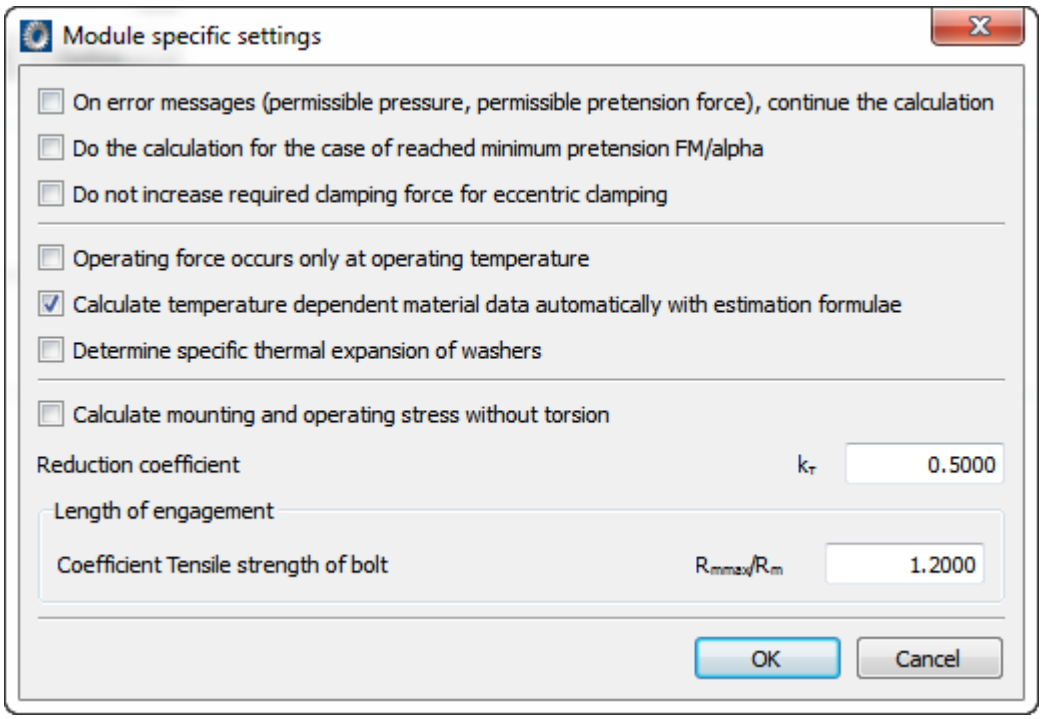


Figure 41.22: Settings for bolts.

- **Continue calculation if error messages appear**
  If you set this flag, the calculation will continue even if error messages are displayed, for example to warn you that the yield point or the permitted pressure has been exceeded.

- **Calculate minimum preload force F_M/α_A achieved**
  If this flag is set, the load F_M/α_A is also calculated. The preload force F_M/α_A is the minimum preload force that must be present, if the entered preload force F_M is included. α_A is the tightening factor. It describes the scatter of the preload force. If this option is set, the results overview in the main screen shows the results of the calculation with F_M, otherwise the results with F_{Mmax} are displayed.

- **Do not increase required clamp load for eccentric clamping**
  KISSsoft increases the required clamp load to prevent gaping for eccentric clamping. You can switch off this function here. You can then specify your

own required clamp load. Take care when using this option. The calculation assumes that gaping does not occur!

- **Operating force only at operating temperature**
  Normally, KISSsoft calculates the minimum preload force based on the required clamp load and loading at ambient and operating temperatures. This flag can be set when the operating force only occurs at operating temperatures. In this case, the minimum preload force is then only calculated at operating temperature. The mounting preload force F_M in the working state is reduced by the proportion $(1-\phi)*F_A$ of the axial bolt load.

- **Calculate temperature dependent material data automatically with estimation formulae**
  KISSsoft can automatically calculate material data at operating temperature by using empirical formulae. These formulae do not take into account the material data you entered: they use an average dependency for "commonly used steels"! Delete this flag if you want to enter your own materials data at the operating temperature.

- **Determine specific thermal expansion of washers**
  This opens the input field for thermal expansion values in the sub-window for (flat) washers. If this flag is not set, the difference in preload force is calculated using the average thermal expansion of the plates. In other words, the (flat) washer has the same thermal expansion as the plates. This is why you have the option of inputting this value. If you do so, the difference in preload force is calculated using the value you specified, but the resilience of the plates is still used in this calculation. VDI 2230 does not specify that a special thermal expansion calculation is to be used for washers.

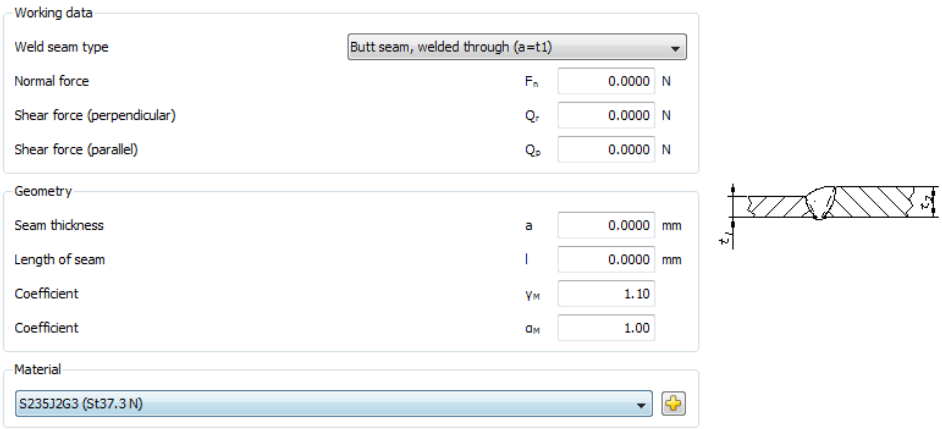
- **Calculate mounting and operating stress without torsion**
  Select this option if the connection is fully pretensioned to the extent that torsional stress no longer occurs. If you do this, the torsion is set to 0 when the system calculates the equivalent stresses that occur during mounting and working.

- **Reduction coefficient**
  The reduction coefficient is used to calculate equivalent stress when the machinery is in its working state. In many cases, the torsional stress in elastically pretensioned connections reduces by 50%. This is why VDI 2230 recommends the value 0.5 is used here.

- **Tensile strength of bolt coefficient**
  This coefficient is used to calculate the minimum length of engagement required to achieve a practical value for Rm (as in VDI 2230). You will find a more detailed description in the section on Stripping strength (see page IV-933).

Chapter 42
Welded joints

Underlying principles of calculation: DIN 18800, Part 1, Edition November 1990, in particular section 8.4 "Joints with arc welding"

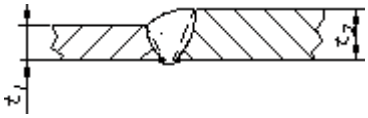


.

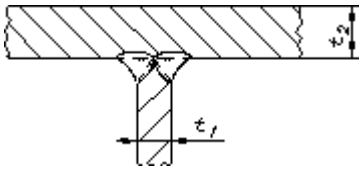
Figure 42.1: Basic data: Welded joints

43.1 Welded joints

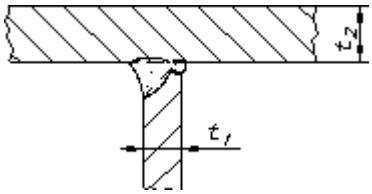
You can apply the calculation method defined in DIN 18800 to these welded seam types:



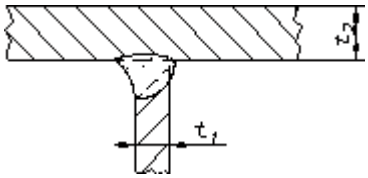
Butt seam through welded



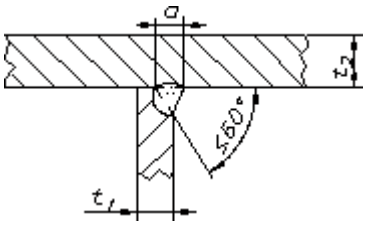
Double HV welded seam counter welded



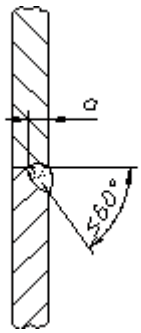
HV welded seam, cap position counter welded



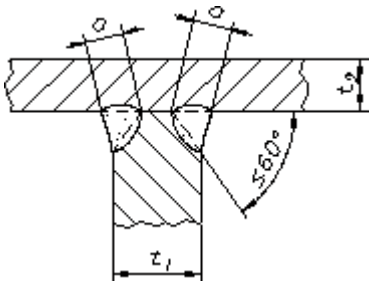
HV welded seam, root through welded



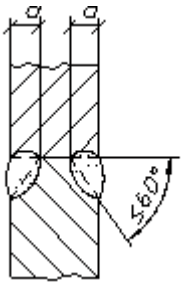
HY-seam with fillet weld, not through welded



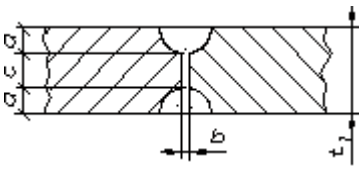
HY-seam, not through welded



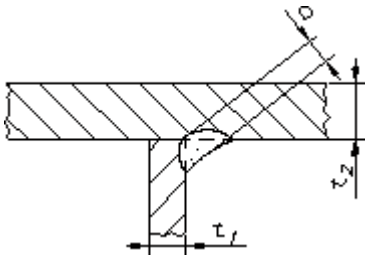
Double-HY-seam with fillet weld, not through welded



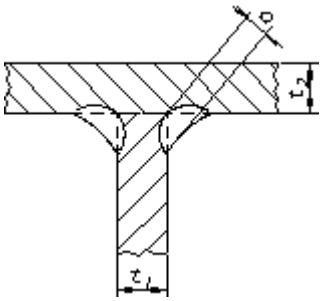
Double-HY-seam, not through welded



Double-I-seam, not through welded



Fillet weld, not through welded



Double-fillet weld, not through welded

43.2 Seam length

Table 20 in DIN 18800 shows various configurations that use welded seam length l.

43.3 Welded seam equivalent stress

Use the following formula to calculate the equivalent stress for butt and filled welded seams:

$$\sigma_{W,V} = \sqrt{\sigma_r^2 + \tau_r^2 + \tau_p^2} \tag{42.1}$$

$\sigma_{W,V}$: Equivalent stress [N/mm^2]
σ_r: Normal stress (vertical to the welded seam) [N/mm^2]
τ_r: Shear stress (vertical to the welded seam) [N/mm^2]
τ_p: Shear stress (parallel to the welded seam) [N/mm^2]

43.4 Weld seam boundary stress

The weld seam boundary stress $\sigma_{W,R,d}$ is calculated with:

$$\sigma_{W,R,d} = \alpha_W \cdot \frac{R_p}{\gamma_M} \tag{42.2}$$

$\sigma_{W,R,d}$: Weld seam boundary stress [N/mm^2]
α_W : Weld seam boundary coefficient [-]
R_p: Yield point [N/mm^2]
γ_M: Part safety coefficient [-]

43.5 Part safety coefficient

The part safety coefficient γ_m is usually 1.1 as specified in section 7.3 in DIN 18800. However, you can also use the value 1.0 to prove the suitability for use or reduced stiffness.

43.6 Boundary safety coefficient

The weld seam boundary coefficient α_W is defined as specified in Table 21 of the standard:

Weld seam type	Seam quality	Stress type	St37-2 and similar	St52-3 and similar
1 - 4	all seam quality	Compression	1.0	1.0
	Proven seam quality	Tension	1.0	1.0
	Unproven seam quality	Tension	0.95	0.85
5 - 15	All seam quality	Compression, tension	0.95	0.85
1 - 15	All seam quality	Shear	0.95	0.85

43.7 Materials

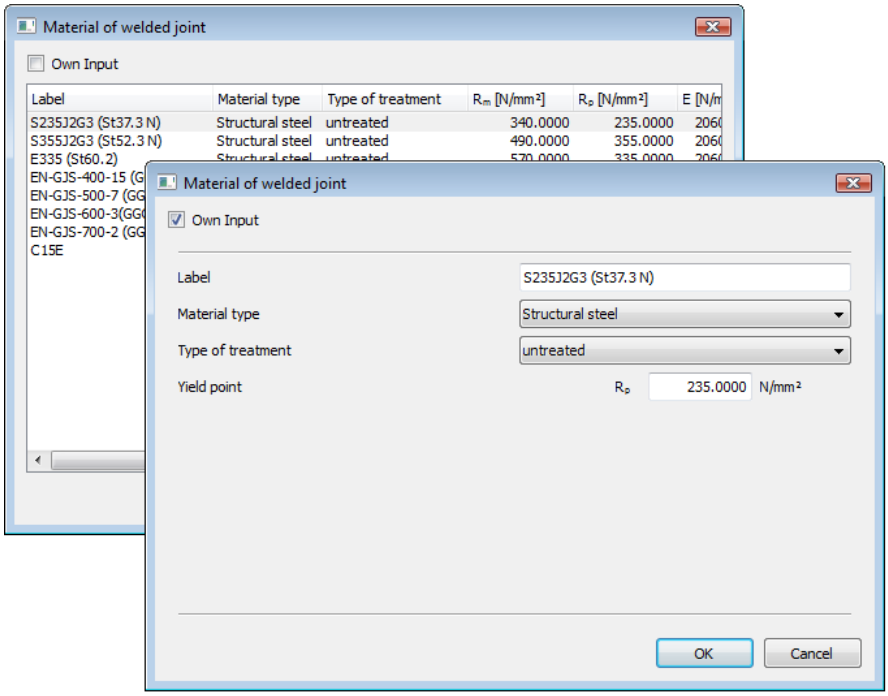


Figure 42.2: Materials screen: Welded joints

The selection list contains materials from the DIN 18800 standard.
If you have set the `Own Input` flag, a new dialog appears here. This displays the material data used in the calculation which you can specify to suit your own purposes. You can also define your own materials directly in the database (see page I-114) so that these can also be used in future calculations.

Chapter 43
Glued and soldered joints

Calculation basis: [64]. The calculation is performed for glued and soldered joints that are subject to shear.

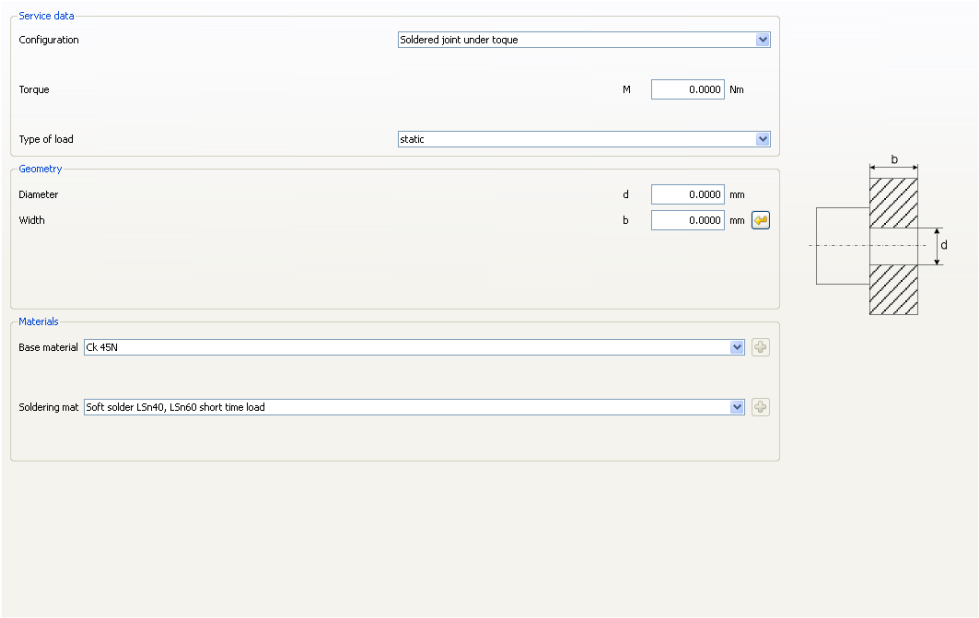


Figure 43.1: Basic data: Glued and soldered Joints

Two different load cases are described:

- Shear force: Transmission of shear force between two surfaces.
- Torque: shaft-hub connection with a torque load.

The connection can be subject either to static or dynamic (usually pulsating) load.

The guide values for the static strength of soldered joints are taken from [64], Table 8/8 (average values of resistance to fracture due to shearing). Guide values for glued joints are taken from Table 8/9. For the pulsating load on soldered joints, 50% of the static strength is assumed as the permitted limit (data not available: you must check these connections to ascertain the endurance limit of the base material.

This may reduce the element safety of the soldered joint by approximately 80%). For glued joints, 30% of the static strength is permitted (as defined in Table 8/9).

At present, the following materials can be used for glued joints:

- cured at ambient temperature.
- cured at higher temperature.

To calculate the shear strength value the program uses the mean value of the minimum and maximum value from the database. The value achieved by optimum implementation as defined in Niemann is not used.

At present, the following materials can be used for soldered joints:

- Soft solder LSn40, LSn60 for short-term loads
- Soft solder LSn40 at continuous load
- Brass solder: Steel NE heavy metals
- New silver solder-copper: Steel
- Silver solder: Steel NE heavy metals

There is no point calculating and sizing soldered joints with light Al-based metals because the strength of the underlying material is usually less than that of the joint. To calculate the shear strength value the program uses the mean value of the minimum and maximum value from the database.

44.1 Basic materials

These materials are only used to size the width, on the basis of the strength of the base material.

At present, you can select these materials:
Ck 45 N, Ck 60, CrNiMo, CrNi 4, CrNiMo, CrMo, St 37.3, St 52.3, St 60.2, Ganevasit, PA 12, PA 66, POM, laminated wood.

You must then still decide which material will be the best for your joint. For example, you should not select PA 12 if you are using a soldered joint.

44.2 Settings

In this window you can view the required safety value and the shear strength to be used in the sizing, you can change this value as required.

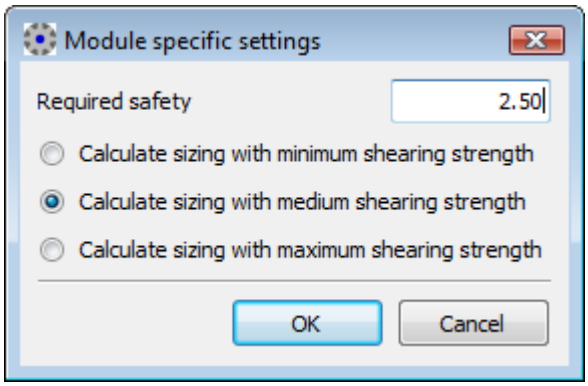


Figure 43.2: Settings: Glued and soldered joints

44.3 Sizings

- Sizing the width on the basis of the base material
  Sizing the adhesion width (for shaft hub), or the adhesion length (for brackets), on the basis on the strength of the base material. The tear resistance of the joint is set so that it corresponds to the tear resistance of the underlying material or the fatigue strength under pulsating stress of the shaft.
- Sizing the width on the basis of stress
  Sizing the adhesion width on the basis of stress. The tear resistance of the joint is sized so that it can withstand the forces it is subjected to without compromising the required safety.

44.4 Bracket connection

Calculating a glue or soldered joint with sheets or plates. You must specify the tension or compression force, the adhesion length, and the metal sheet or plate thickness.

44.5 Shaft connections

Calculating a glued or soldered joint for shaft-hub connections. You must specify the transferring torque in Nm, the joint diameter and the length of the adhesion point.

Chapter 44
Retaining rings (self-locking rings, Seeger rings)

This module is used to perform calculations for shaft or hub retaining rings. The calculations are carried out in accordance with the manual published by the company Seeger. To open the module, navigate to the module tree and double-click the appropriate module (see Figure 44.1).

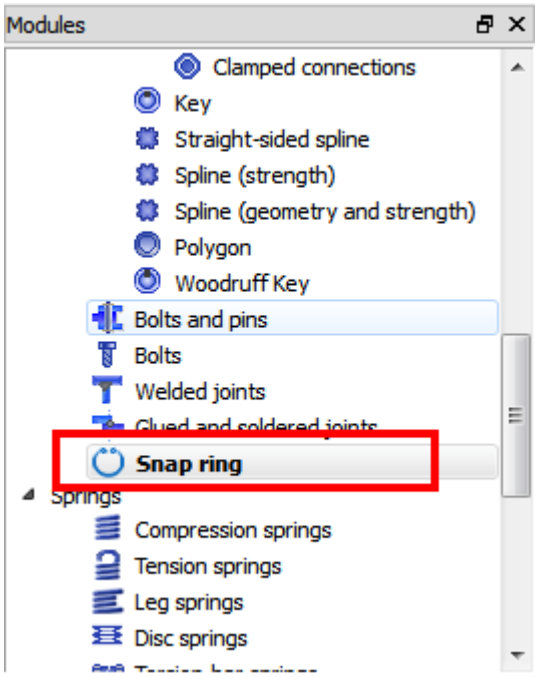


Figure 44.1 The Seeger ring calculation module

45.1 Basic data

Input the following data in the Basic data tab:

"Geometry" group

- Shaft/bore ring: specifies whether the calculation is to be performed for a shaft or for a bore ring
- Retaining ring/Circlip: specifies whether the calculation is to be performed for a circlip or a retaining ring
- d1: nominal length, the shaft diameter for a shaft ring, or the bore diameter for a bore ring
- d2: groove diameter
- d3: inside diameter of the Seeger ring for shafts or external diameter of Seeger ring for bores in the unstressed state
- b: the maximum radial width of the Seeger ring
- Measure l: see Figure 44.2
- s: the thickness of the ring
- ψ: permissible dishing angle of the Seeger ring (see Figure 44.3)
- sharp-cornered bearing area: Controls whether conditions with a chamfer, the corner distance or the radius of g should be considered (see Figure 44.3)
- g: the chamber or corner distance/radius

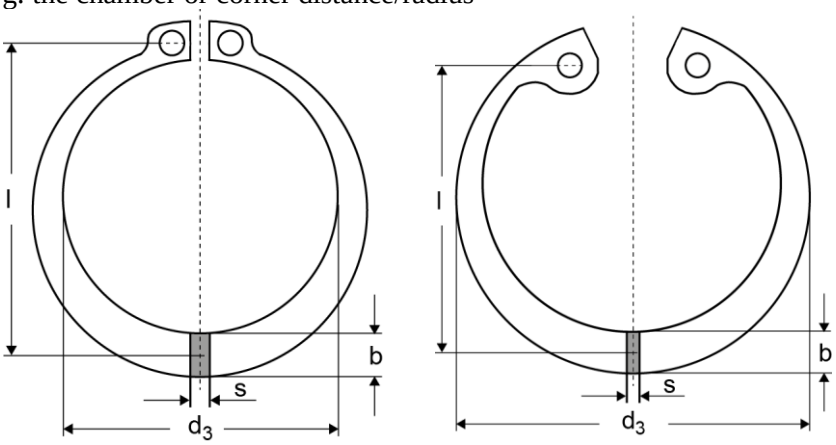


Figure44.2 Geometry of shaft ring (a) and bore ring (b)

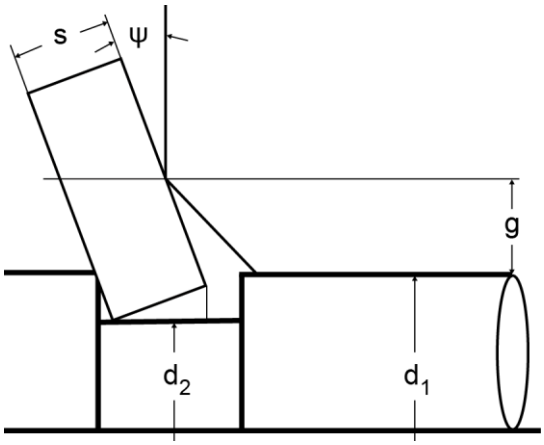


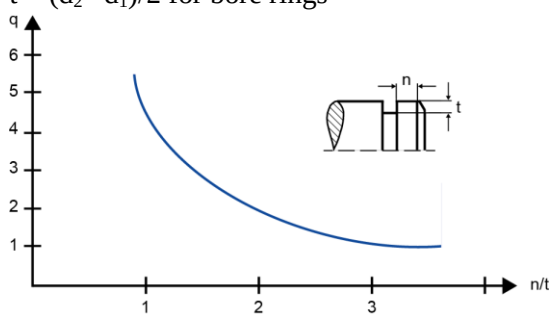
Figure 44.3 Definition of geometric values s, ψ, g

- "Operating data" group
  - q: the load factor, taking into consideration the effect of the shoulder length ratio (see Figure 44.4)
  - μ the coefficient of friction between the ring surface and the shaft/bore surface
- "Materials" group
  - In this group you can define the material of the ring and shaft/bore. The functionality is similar to the rest of the KISSsoft modules which are located in the "Connections" module group.

45.2 Automatic calculation of load factor q

If you click on the button next to q, the system displays a window in which you can calculate q, based on the ratio of the shoulder length n to the groove depth t (see Figure 44.4). The groove depth is defined as:

- $t = (d_1 - d_2)/2$ for shaft rings
- $t = (d_2 - d_1)/2$ for bore rings



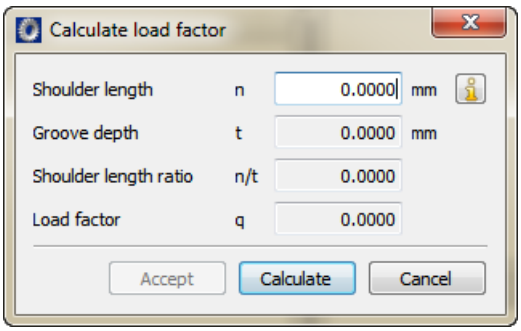


Figure 44.4 (a) Definition of load factor q, shoulder length n and groove depth t. (b) Sizing window for load factor q.

45.3 Automatic calculation of the dishing angle ψ

Use Figure 44.5 to calculate ψautomatically.

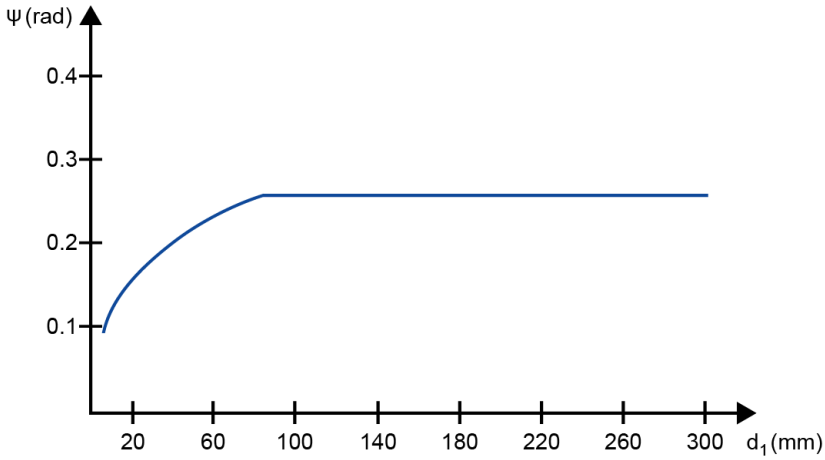
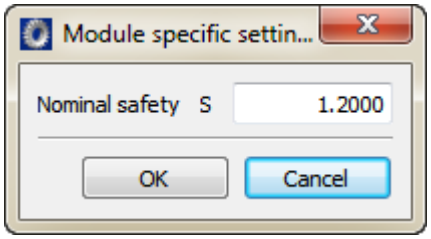


Figure 44.5 Calculation of ψ, based on d1.

45.4 Module specific settings



Here you can define the minimum required safety S.

Chapter 45
Answers to Frequently Asked Questions

46.1 Adding new bolt types to the database

The KISSsoft database includes the following bolt types:

- Hexagon socket head cap screws EN ISO 4762
- Hexagon headed bolts with shank (AB) EN ISO 4014
- Hexagon headed bolts without shank (AB) EN ISO 4017
- Slotted cheese head screw EN ISO 1207
- Hexagon headed bolts with shank, metric fine thread (AB) EN ISO 8765
- Hexagon headed bolts without shank, metric fine thread (AB) EN ISO 8676
- Hexagon headed screws with flange, light series, shape F EN 1662
- Hexagon headed screws with flange, light series, shape U EN 1662
- Hexagon headed screws with flange, heavy series, shape F EN 1665
- Hexagon headed screws with flange, heavy series, shape U EN 1665
- Own definition of screw geometry

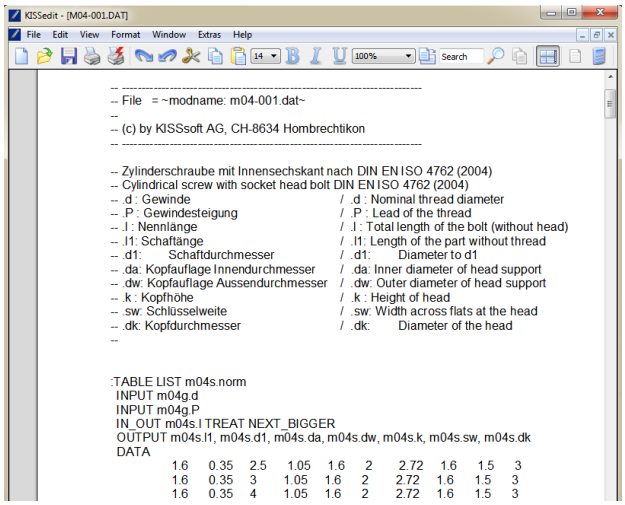
For each of these bolts types, a number of tables list the various bolts sizes (= bolts series). You will find the name of the file that contains the bolts series data in the database (see page I-114).

You enter a new size within an existing bolt type i.e. extend the bolt series (see page IV-960) or input a new bolt type (see page IV-962).

46.1.1 Extending an existing bolt series

Example: Enter the data for M8 bolts with a length of 100 mm in the "hexagon socket head cap screw EN ISO 4762" series.

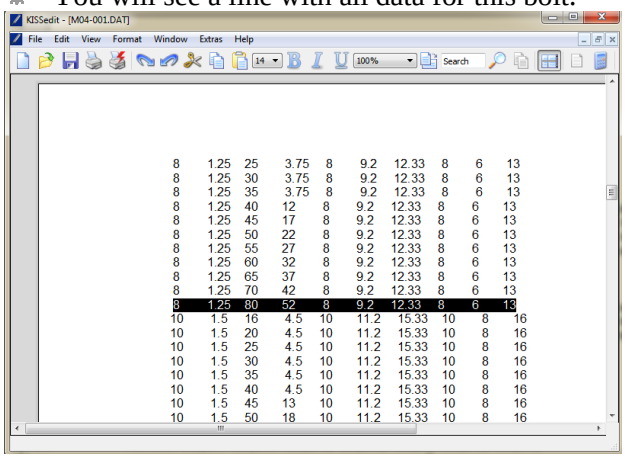
Then start the database tool. Open the `Screw Type M000.KDB, M040Typ` table. There, select the `Hexagon socket head cap screw EN ISO 4762` data record. In the `File name` field you will see the name of the file which contains the table with the bolt series data. Click the `Edit` button at the end of the input line to open the file in the Editor:



```
-- ------------------------------------------------------------------------
-- File   = ~modname: m04-001.dat~
--
-- (c) by KISSsoft AG, CH-8634 Hombrechtikon
-- ------------------------------------------------------------------------

-- Zylinderschraube mit Innensechskant nach DIN EN ISO 4762 (2004)
-- Cylindrical screw with socket head bolt DIN EN ISO 4762 (2004)
-- .d : Gewinde                          /  .d : Nominal thread diameter
-- .P : Gewindesteigung                  /  .P : Lead of the thread
-- .l : Nennlänge                        /  .l : Total length of the bolt (without head)
-- .l1: Schaftlänge                      /  .l1: Length of the part without thread
-- .d1:      Schaftdurchmesser           /  .d1:      Diameter to d1
-- .da: Kopfauflage Innendurchmesser     /  .da: Inner diameter of head support
-- .dw: Kopfauflage Aussendurchmesser    /  .dw: Outer diameter of head support
-- .k : Kopfhöhe                         /  .k : Height of head
-- .sw: Schlüsselweite                   /  .sw: Width across flats at the head
-- .dk: Kopfdurchmesser                  /  .dk:      Diameter of the head
--


:TABLE LIST m04s.norm
 INPUT m04g.d
 INPUT m04g.P
 IN_OUT m04s.l TREAT NEXT_BIGGER
 OUTPUT m04s.l1, m04s.d1, m04s.da, m04s.dw, m04s.k, m04s.sw, m04s.dk
 DATA
        1.6   0.35   2.5   1.05   1.6   2   2.72   1.6   1.5   3
        1.6   0.35   3     1.05   1.6   2   2.72   1.6   1.5   3
        1.6   0.35   4     1.05   1.6   2   2.72   1.6   1.5   3
```

To enter a new bolt:

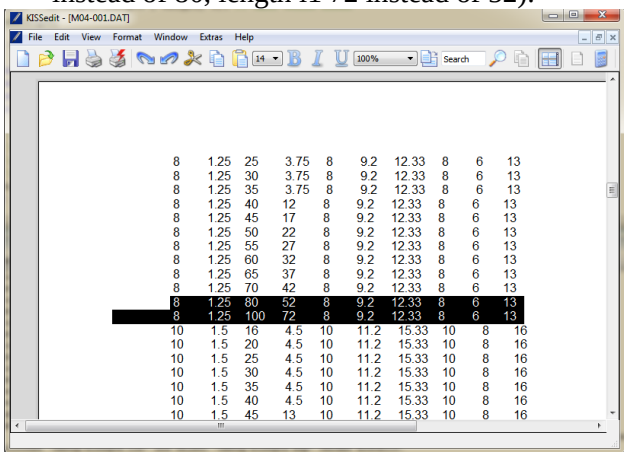
- Look for a similar bolt (M8, length 80mm).
- You will see a line with all data for this bolt.



```
8    1.25   25    3.75   8    9.2   12.33   8   6   13
8    1.25   30    3.75   8    9.2   12.33   8   6   13
8    1.25   35    3.75   8    9.2   12.33   8   6   13
8    1.25   40    12     8    9.2   12.33   8   6   13
8    1.25   45    17     8    9.2   12.33   8   6   13
8    1.25   50    22     8    9.2   12.33   8   6   13
8    1.25   55    27     8    9.2   12.33   8   6   13
8    1.25   60    32     8    9.2   12.33   8   6   13
8    1.25   65    37     8    9.2   12.33   8   6   13
8    1.25   70    42     8    9.2   12.33   8   6   13
8    1.25   80    52     8    9.2   12.33   8   6   13
10   1.5    16    4.5    10   11.2  15.33   10  8   16
10   1.5    20    4.5    10   11.2  15.33   10  8   16
10   1.5    25    4.5    10   11.2  15.33   10  8   16
10   1.5    30    4.5    10   11.2  15.33   10  8   16
10   1.5    35    4.5    10   11.2  15.33   10  8   16
10   1.5    40    4.5    10   11.2  15.33   10  8   16
10   1.5    45    13     10   11.2  15.33   10  8   16
10   1.5    50    18     10   11.2  15.33   10  8   16
```

- Copy this line. When you do so, note the exact sequence of the lines.

- Change the data in accordance with Table 1 in EN ISO 4762 (length 100 instead of 80, length l1 72 instead of 52).



8	1.25	25	3.75	8	9.2	12.33	8	6	13
8	1.25	30	3.75	8	9.2	12.33	8	6	13
8	1.25	35	3.75	8	9.2	12.33	8	6	13
8	1.25	40	12	8	9.2	12.33	8	6	13
8	1.25	45	17	8	9.2	12.33	8	6	13
8	1.25	50	22	8	9.2	12.33	8	6	13
8	1.25	55	27	8	9.2	12.33	8	6	13
8	1.25	60	32	8	9.2	12.33	8	6	13
8	1.25	65	37	8	9.2	12.33	8	6	13
8	1.25	70	42	8	9.2	12.33	8	6	13
8	1.25	80	52	8	9.2	12.33	8	6	13
8	1.25	100	72	8	9.2	12.33	8	6	13
10	1.5	16	4.5	10	11.2	15.33	10	8	16
10	1.5	20	4.5	10	11.2	15.33	10	8	16
10	1.5	25	4.5	10	11.2	15.33	10	8	16
10	1.5	30	4.5	10	11.2	15.33	10	8	16
10	1.5	35	4.5	10	11.2	15.33	10	8	16
10	1.5	40	4.5	10	11.2	15.33	10	8	16
10	1.5	45	13	10	11.2	15.33	10	8	16

- Save the file.
- Document any changes for other users.

46.1.2 Creating a new bolt type

Before you can add a new bolt type you must already be familiar with the table structure. You must know which value goes in which column (use the variable names from the descriptions in the table header).

Then, proceed as follows:

- In the database, open the data record most similar to the new bolt type.
- Copy this data record and rename it to suit the new bolt type.
- Click the `Edit` button at the end of the input line for the file name. This opens a file which still contains the "old" values.
- Overwrite these values with the new values. Note the variables structure (i.e. a specific variable is assigned to a number, depending on where the number appears) and the sorting of the lines.
- Save the updated file with a new name and close the Editor.
- Transfer the new file name to the database (to create the cross reference).
- Then save the new data record.

Part V

Springs

Chapter 46
Compression springs

The calculation of compression springs is based on DIN EN 13906-1 (2002)[30].

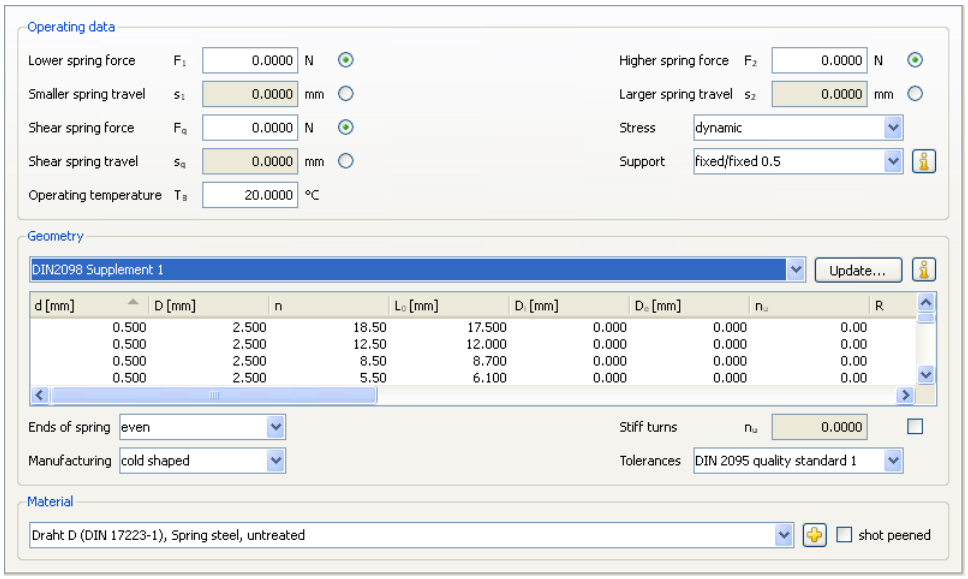


Figure 46.1 Basic data for compression springs

Operating data

When you specify a load, you can use your own value as the spring force or travel. You can also specify whether the spring is to be subject to static, quasistatic, or dynamic, force.

Geometry

You can select the geometry data according to DIN 2098 Part 1 directly from this table.

If you select `Own input`, you can either take selected values from the list or enter your own values. Select `Own input` to specify your own spring length and the diameter.

Instead of using the spring length in its non-stressed state L_0 you can also use a spring length in its stressed state L_1 or select L_2.

The choice of the `End of spring` and `Manufacture` affects the calculation of the block length L_c.

Click the `Update` button to calculate the block lengths and the resulting values of the current situation for individual springs and display them in a table.

47.1 Strength values

The material strengths are stored in different files, depending on diameter. The transverse strength is either saved in the tables, as in DIN EN 13906-1 for thermo-formed springs, or calculated from the predefined tensile strength as $\tau_{czul} = 0.56 \cdot R_m$.

To calculate the endurance limit, use either the Goodman diagram as defined in EN 13906-1 or an approximation. The approximation assumes a dynamic strength of $0.25 \cdot R_m$ and a gradient of the graph of the upper stress in the Goodman diagram of 0.75. For shot peened materials, the dynamic strength is increased by 20%. These values roughly correspond to the diagrams in the DIN EN 13906-1 standard, but you should regard the safeties more conservatively.

47.2 Shear stress values

The calculation of the highest shear stress also calculates the axial and shear spring travel.

$$\tau_{\max} = \frac{8}{\pi \cdot d^3} \cdot \left[F\left(D + s_Q\right) + F_Q \cdot \left(L - d\right)\right] \tag{46.1}$$

τ_{max}: highest shear stress [N/mm^2]
d: wire diameter [mm]
F: spring force [N]
D: average coil diameter [mm]
s_Q: shear spring travel [mm]
F_Q: shear spring force [N]
L: spring length [mm]

The highest corrected shear stress is calculated by:

$$\tau k_{\max} = k \cdot \tau_{\max} \tag{46.2}$$

τk_{max}: highest corrected shear stress [N/mm^2]
τ_{max}: highest shear stress [N/mm^2]

K: stress correction factor
(dependent on the ratio D/d)

47.3 Bearings coefficient

The `Support` you select defines the value of the support coefficient ν, as shown in Figure 46.2.

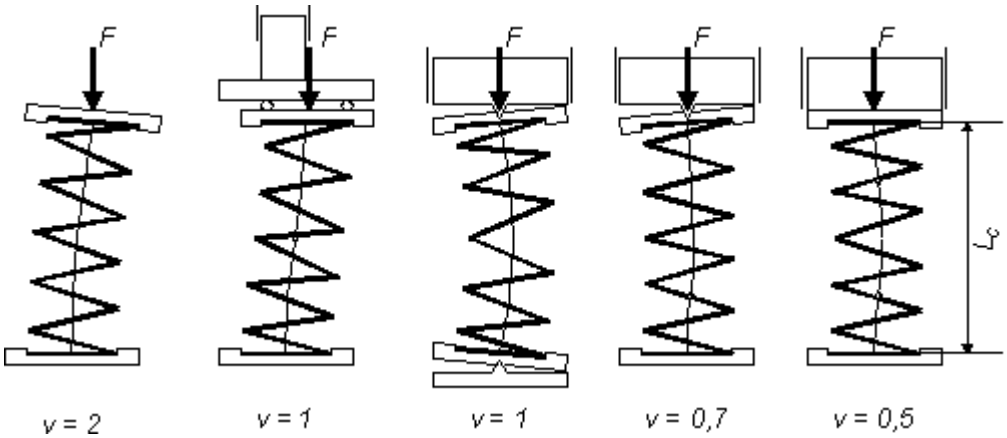


Figure 46.2: Support with associated support coefficients for axially stressed compression springs

The support coefficient ν is used for calculating the buckling spring travel sk. If the buckling safety factor is not reached then the spring must be guided, otherwise it will buckle.

47.4 Materials

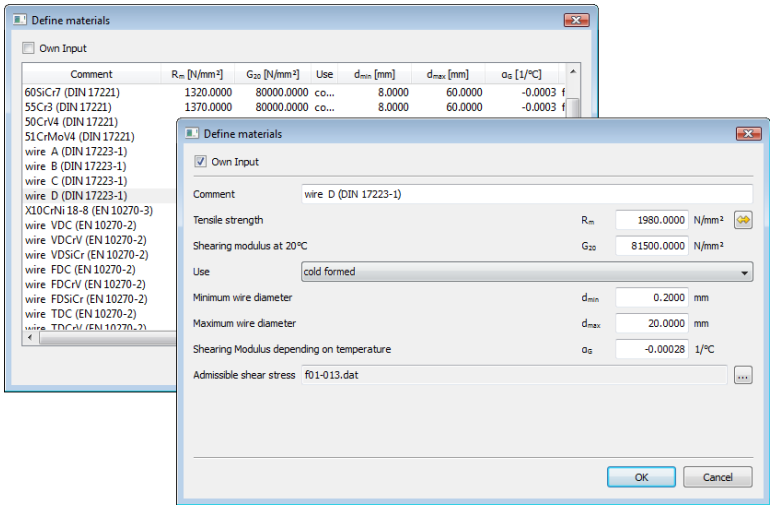


Figure 46.3: Compression springs

The selection list includes materials from the DIN 17221, DIN 17223-1, DIN 10270-1, DIN 10270-1 and DIN 10270-3 standards.
If you have set the `Own Input` flag, a new dialog appears here. This displays the material data used in the calculation which you can specify to suit your own purposes. You can also define your own materials directly in the database (see page I-114) so that these can also be used in future calculations.

47.5 Tolerances

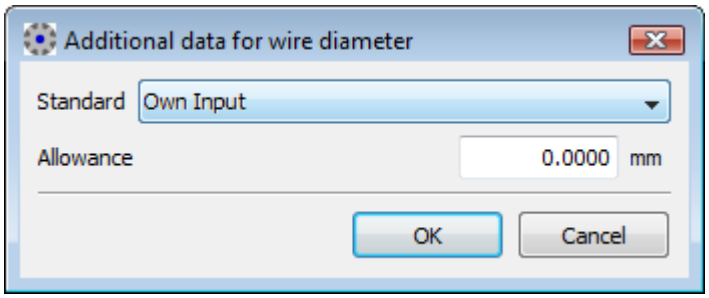


Figure 46.4: Additional wire diameter data for compression springs

When you select a spring from the table (in accordance with DIN 2098 part 1), the tolerance of the diameter used here is specified in DIN 2076 C. To change the diameter tolerance, toggle to the `Own input` list to open the input fields. Here click the button next to the wire diameter field to open another screen. (see Figure)

In the Tolerances screen you can select wire diameters in accordance with DIN 2076 (1984), DIN 2077 (1979), EN 10270-1 (2001), EN 10270-2 (2001), EN 10270-3 (2001), or enter your `Own input` value.
If you select a wire diameter tolerance in accordance with the standard, the tolerance will be inserted directly in the screen.
If you select `Own input`, you can define the value yourself.

Other tolerances are listed in accordance with the quality standard. In the `Tolerances` list in the basic data you can choose one of the quality standards in accordance with DIN 15800 (2009)[14] or DIN 2096 Part 1 (1981)[15].

47.6 Relaxation

The existing spring force can be located after a specific period of time by calculating the relaxation. The compression spring settles to a particular value. Relaxation is also known as creep. The relaxation values are listed in the DIN EN 13906-1 standard, and shown in diagrams. The diagrams show curves at specific diameters and temperatures, which are then recorded in a relaxation-stress diagram. By noting the data from 2 different wire diameters temperatures, you can then infer or extrapolate the relaxation value for a specified level of stress at operating temperature and for a specific wire diameter.

In KISSsoft, the relaxation diagram for 48h can be displayed in relation to diameter, temperature and stress. Other graphics are also available that show the progress of relaxation over time and the spring force. The results for the specified conditions are then displayed in the relaxation report for 48h. The value of the spring force is also calculated after 48h.

To extend the data for the materials relaxation curves, or add new data, add this new information to the *.dat file for the appropriate spring material.

The relaxation curves in this file can be defined with 2 or 3 given measurement points. The curves are then calculated from these points.

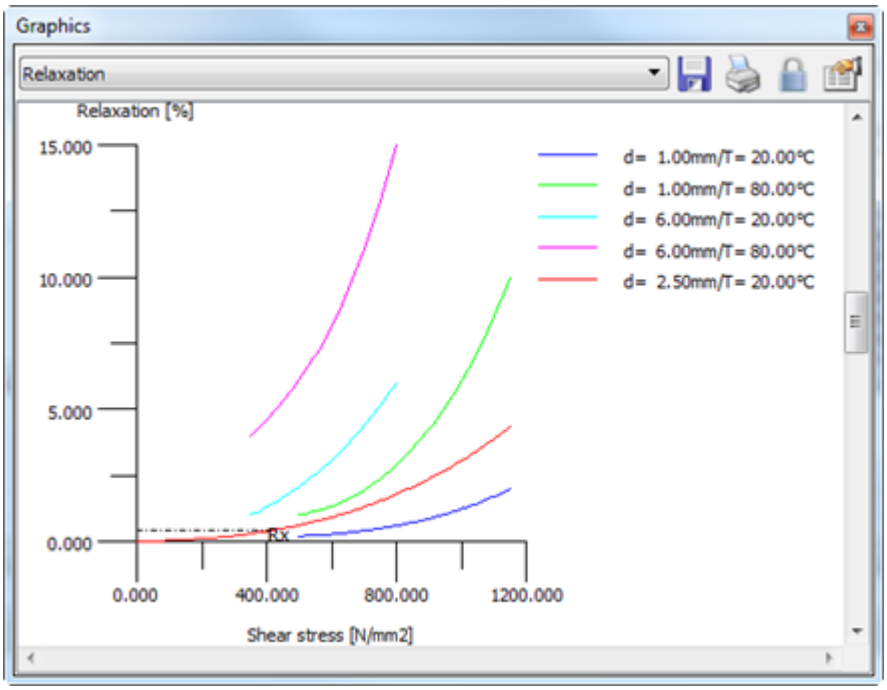


Figure 46.5: Relaxation for compression springs

47.7 Drawing data

To access the spring data required to create a drawing, click `Drawing data`. Use the `F10SPRING?.RPT` file (for compression springs), or the `F20SPRING?.RPT` file (for tension springs), etc. (? = d/e/f/i/s for the required language) to modify the template to your own requirements.

47.8 Sizings

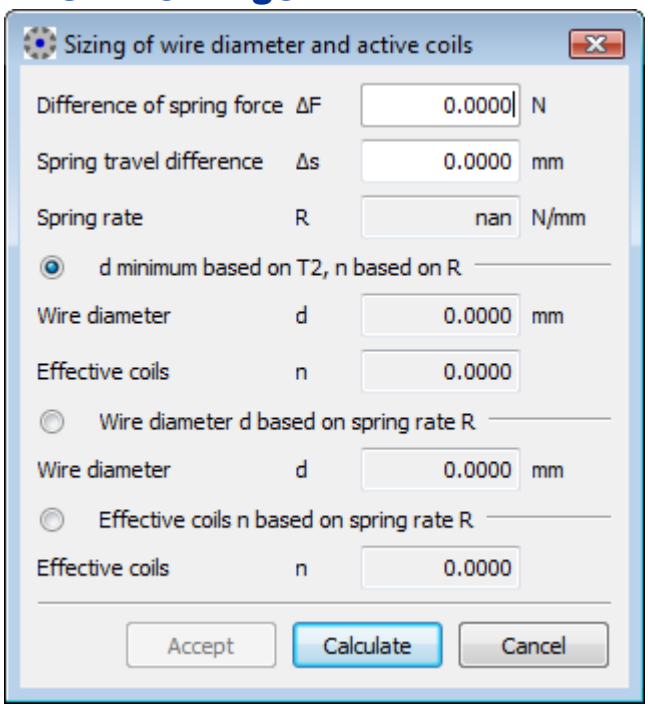


Figure 46.5: Sizing screen: Compression springs

If you selected `Own input` in the list under Geometry, you now see input fields here instead of a table showing the values defined in the standard. Next to the `Wire diameter` and the `Effective coils`, you can click the button to size the following values.

Using the predefined spring rate R = ΔF/Δs, the number of turns n can also be calculated if the wire diameter has been predefined. The number of turns is defined by this value, but the strength and the geometric constraints are not checked. The program also suggests a value for the minimum wire diameter and the associated number of turns. The minimum wire diameter here is defined by the strength of the material.

Chapter 47
Tension springs

The tension spring calculation is described in the DIN EN 13906-2 (2013)[31] standard.

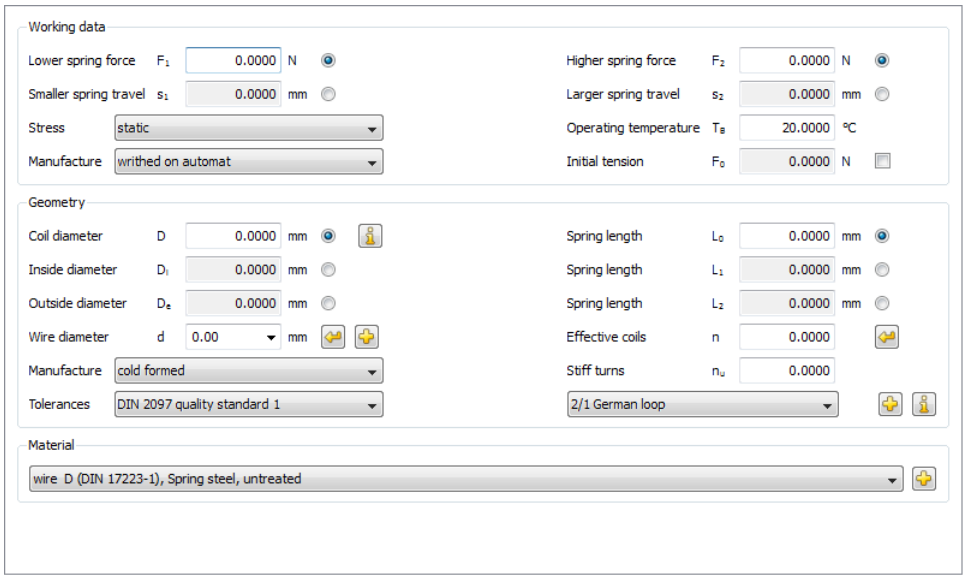


Figure 47.1: Basic data: Tension springs

Operating data

When you specify a load, you can use your own value as the spring force or travel. This force is defined as the `initial preload force` F0, which is required to open the coils which lie one on top of the other. This force is only present if the spring is pretensioned.
If the flag for `Inner preload` is not set, you can influence the number of effective coils.
You can also specify whether the spring is to be subject to static, quasistatic, or dynamic, stress.

Geometry

You can specify the spring length and the spring diameter directly in the main screen.
Instead of using the spring length in its non-stressed state L_0 you can also use a spring length in its stressed state L_1 or select L_2.

For the wire diameter, you can either select the diameter values as defined in DIN 2098 supplement 1 from the list or enter your own value directly in the list.

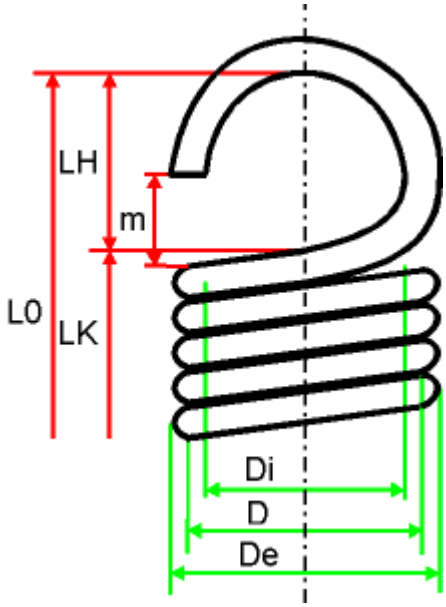


Figure 47.2: Definitions used for tension springs

48.1 Strength values

Permissible shear stress is calculated from the tensile strength of cold formed tension springs. The tensile strength values are determined by diameter values stored in various files. The shear stress is calculated using the formula $\tau_{zul} = 0.45 \cdot R_m$. Thermo-formed tension springs should not exceed the permissible shear stress of $\tau_{zul} = 600 N/mm^2$. These values apply to static or quasistatic cases. Tension springs as defined in DIN 2097 should not be subjected to dynamic stress if at all possible. Shear stress is distributed very unevenly over the cross section of the wire or pin of a tension spring. You can use an intensity factor k to approximate the highest arithmetical stress. Additional stresses are present at the transitions to the eyes. As they may be well above the permissible shear stress, no generally applicable fatigue strength values can be given.

48.2 Shear stress values

The shear stress τ is calculated for the sizing of springs that are subject to static and quasistatic stress:

$$\tau = \frac{8 \cdot D \cdot F}{\pi \cdot d^3} \tag{47.1}$$

τ Shear stress [N/mm^2]
D: medium coil diameter [mm]
F: spring force [N]
d: wire diameter [mm]

Calculating shear stress for springs subjected to dynamic stress:

$$\tau_k = k \cdot \tau \tag{47.2}$$

τ_k: corrected shear stress [N/mm^2]
τ: Shear stress [N/mm^2]
k: stress correction factor
(dependent on the ratio D/d)

48.3 Manufacturing type

Thermo-formed tension springs cannot be produced with inner preload force because the heat treatment creates an air gap between the coils. Cold shaped tension springs can be manufactured in two ways, either by winding on a coiling bench or by winding on a spring winding machine. As defined in DIN EN 13906-2, a formula is specified for each manufacturing method which gives the permissible inner shear stress τ_0.

48.4 Eyes screen

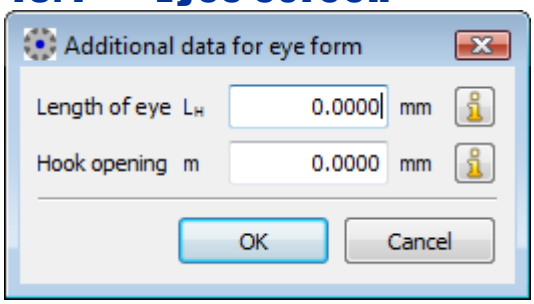
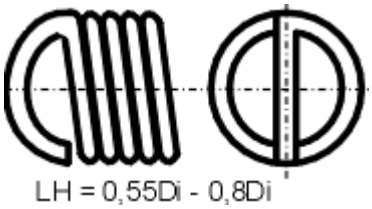


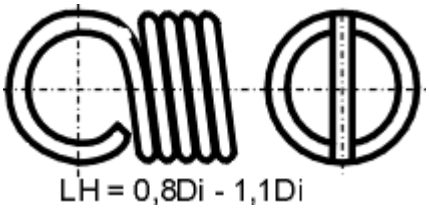
Figure 47.3: Definitions used for eyes

Using the definitions of the `Length of eye` L_H in each case, in this screen, you can then determine the total length of the spring. In contrast, the `Hook opening` m is a reported value that is not used in this calculation.

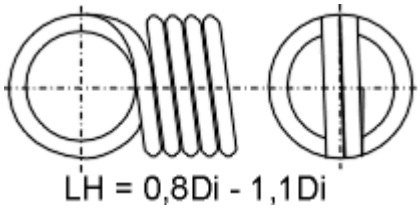
DIN 2097 defines 13 different eye shapes for tension springs. The program suggests different eye lengths depending on the shape of the eye. The position of both eyes is also handled separately in this DIN standard.



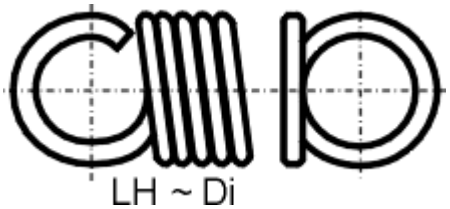
1/2 German loop



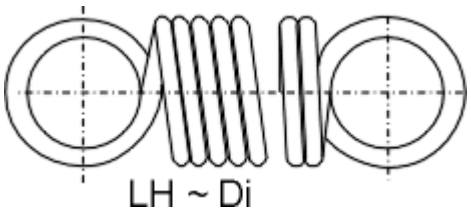
1/1 German loop



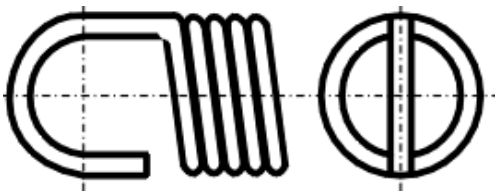
2/1 German loop

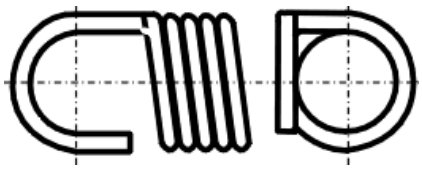


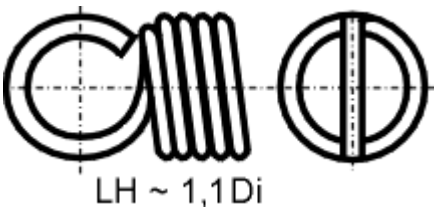
1/1 German loop at side



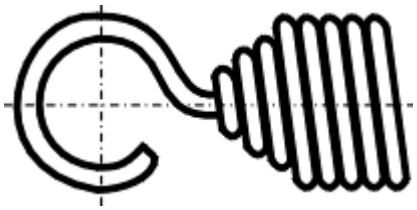
2/1 German loop at side

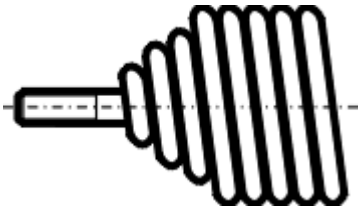

Hook

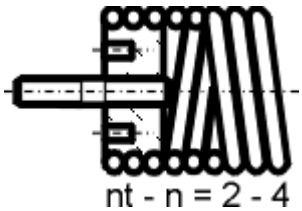

Extended side hook



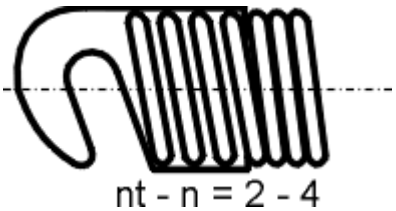
English loop


Coiled-in hook

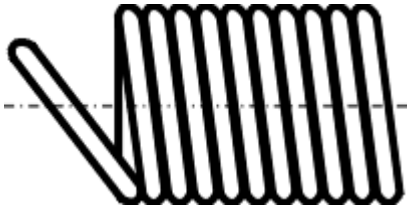

Screwed plug



Screwed-in screw cap



Screwed-in shackle



1/1 German loop inclined

48.5 Materials

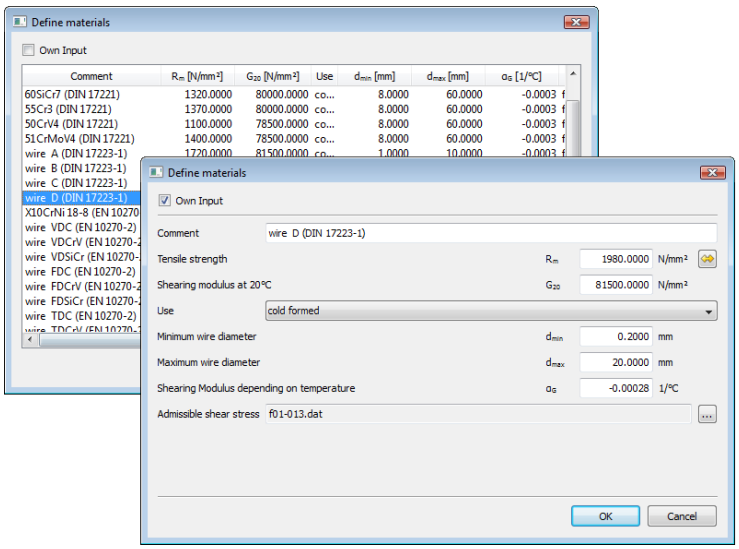


Figure 47.4: Materials screen: Tension springs

The selection list includes materials from the DIN 17221, DIN 17223-1, DIN 10270-1, DIN 10270-1 and DIN 10270-3 standards.
If you have set the `Own Input` flag, a new dialog appears here. This displays the material data used in the calculation which you can specify to suit your own purposes. You can also define your own materials directly in the database (see page I-114) so that these can also be used in future calculations.

48.6 Settings

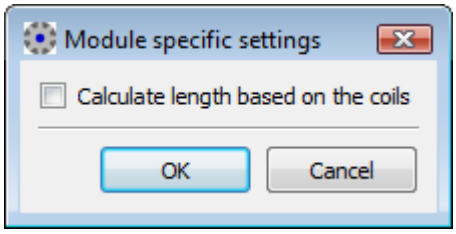


Figure 47.5: Settings: Tension springs

If the `Calculate length using coils` flag is set, and the spring is prestressed (`Initial tension force` flag set), the length of the spring is calculated from the number of coils. You can no longer input the spring length in the dialog.

48.7 Tolerances

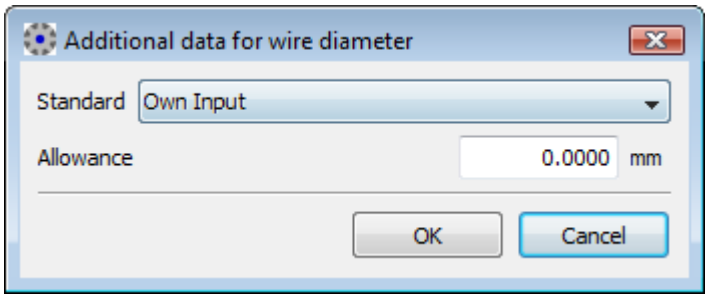


Figure 47.6: Additional wire diameter data for tension springs

Click the button next to the Wire diameter field to open the Tolerances screen. In this screen you can select a wire diameter as defined in DIN 2076 (1984), DIN 2077 (1979), EN 10270-1 (2001), EN 10270-2 (2001), EN 10270-3 (2001) or input your `Own input` to enter your own value.
If you select a wire diameter tolerance in accordance with the standard, the tolerance will be inserted directly in the screen.
If you select `Own input`, you can define the value yourself.

Other tolerances are listed in accordance with the quality standard. In the `Tolerances` list in the basic data you can choose one of the quality standards in accordance with DIN 2097[16] or DIN 2096 Part 1 (1981)[15].

48.8 Relaxation

The existing spring force can be located after a specific period of time by calculating the relaxation. The compression spring settles to a particular value. Relaxation is also known as creep. The relaxation values are listed in the DIN EN 13906-1 standard (the standard for compression springs), and shown in diagrams. It was assumed that the same relaxation values can be used for both for tension springs and compression springs. The diagrams show curves at specific diameters and temperatures, which are then recorded in a relaxation-stress diagram. By noting the data from 2 different wire diameters temperatures, you can then infer or extrapolate the relaxation value for a specified level of stress at operating temperature and for a specific wire diameter.

In KISSsoft, the relaxation diagram for 48h can be displayed in relation to diameter, temperature and stress. Other graphics are also available that show the progress of relaxation over time and the spring force. The results for the specified conditions are then displayed in the relaxation report for 48h. The value of the spring force is also calculated after 48h.

To extend the data for the materials relaxation curves, or add new data, add this new information to the *.dat file for the appropriate spring material.

The relaxation curves in this file can be defined with 2 or 3 given measurement points. The curves are then calculated from these points.

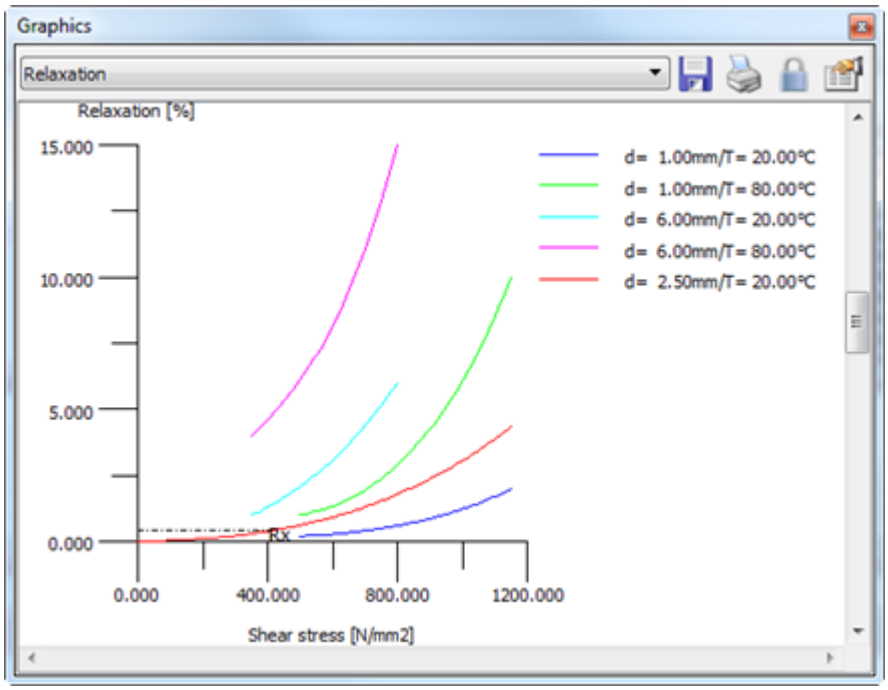


Figure 47.5: Relaxation for tension springs

48.9 Drawing data

To access the spring data required to create a drawing, click `Drawing data`. Use the `F10SPRING?.RPT` file (for compression springs), or the `F20SPRING?.RPT` file (for tension springs), etc. (? = d/e/f/i/s for the required language) to modify the template to your own requirements.

48.10 Sizings

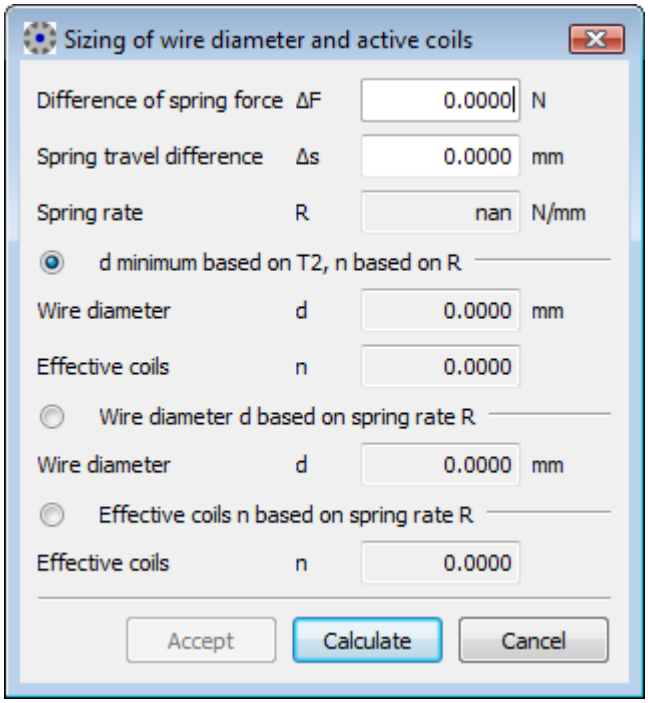


Figure 47.7: Sizing screen: Tension springs

Click the buttons next to the `Wire diameter` and `Effective coils` fields to use the spring rate $R = \Delta F/\Delta s$ to calculate the number of turns n for the predefined wire diameter. The program also suggests a value for the minimum wire diameter and the associated number of turns. The minimum wire diameter here is defined by the strength of the material.

Chapter 48
Leg springs

The calculation used for leg springs is defined in DIN EN 13906-3 (2002) [32].

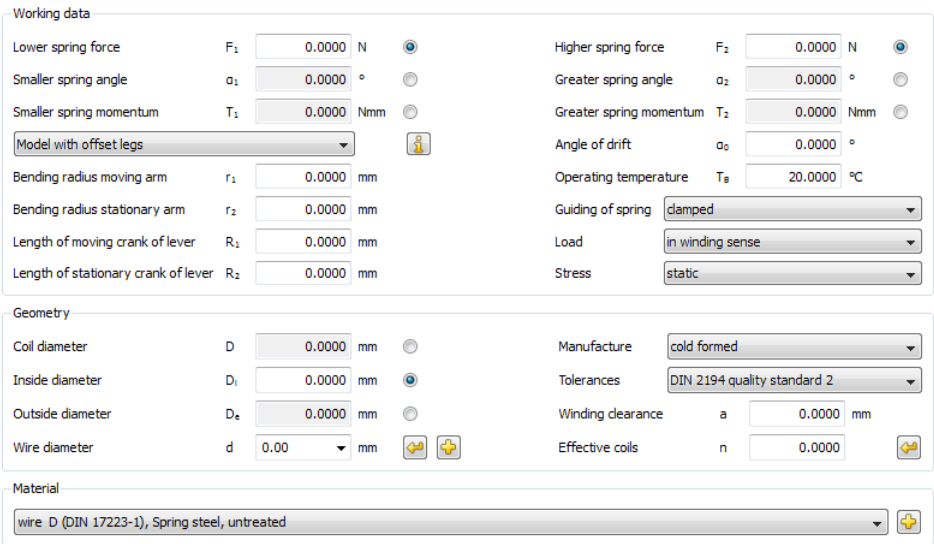


Figure 48.1: Basic data: Leg springs

Operating data

When you define a load you can either enter a value for the spring force, spring angle, or spring torque.
To do this, you must first specify the torsion arm (R_1,R_2) on which the force is applied to the spring.
The value α0 is used to identify the starting angle. This is used together with the direction of load (sense of winding) to calculate the maximum angle of the spring.
Depending on which value you select in the `Guiding of spring` list, the report will also include a reference value for the diameter of the working mandrel or the working bush.
You can also specify whether the spring is to be subject to static, quasistatic, or dynamic, stress.

Geometry

You can select the geometry data according to DIN 2098 Part 1 directly from this table.
If you select `Own input`, you can either take selected values from the list or enter your own values. If you select `Own input` you can select a value for the spring diameter and enter it directly.
The winding clearance is the distance between the coils.

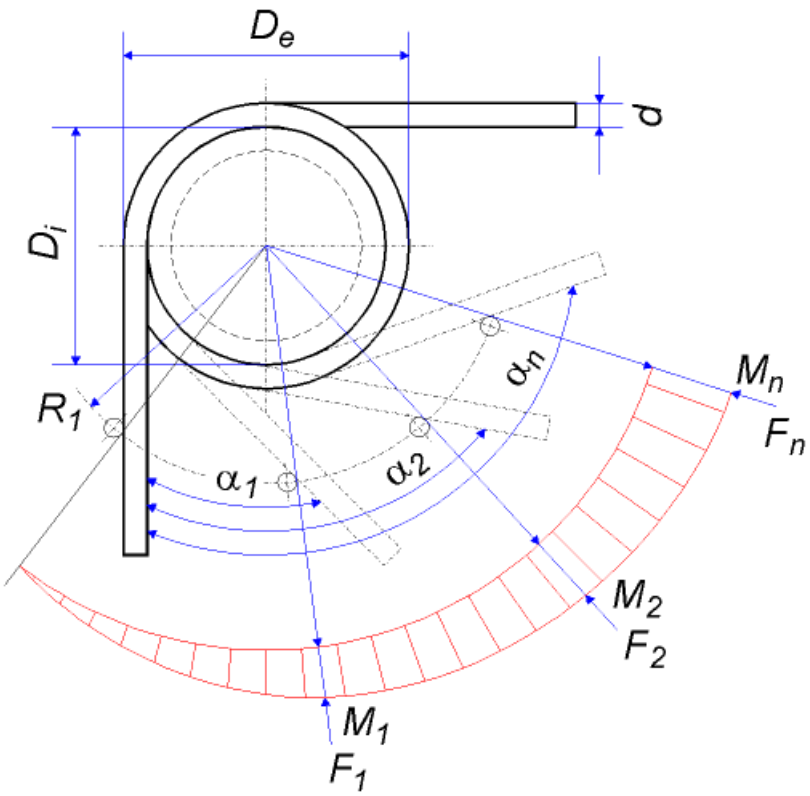


Figure 48.2: Definitions used for leg springs

49.1 Strength values

The permissible bending stress for cold formed leg springs is calculated from the tensile strength. The tensile strength values are determined by diameter values stored in various files. The bending stress is calculated using the formula $\sigma_{zul} = 0.7 \cdot R_m$. These values apply to static or quasistatic cases. The bending of the wire or pin axis due to the load causes an asymmetrical distribution of the spring stresses. In order to approximate the arithmetical stress (dynamic case), you can use the stress coefficient q in the calculation.

49.2 Bending stress values

The bending stress σ is calculated for the sizing of springs that are subject to static and quasistatic stress:

$$\sigma = \frac{32 \cdot T}{\pi \cdot d^3} \tag{48.1}$$

σ shear stress [N/mm^2]
T: spring torque [Nm]
d: wire diameter [mm]

Calculating the bending shear stress for springs subject to dynamic stress:

$$\sigma_q = q \cdot \sigma \tag{48.2}$$

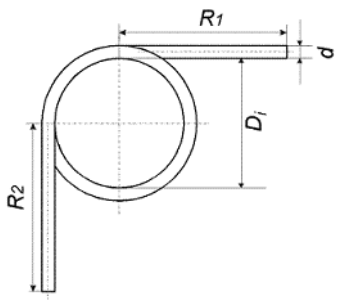
σ_q: corrected bending shear stress [N/mm^2]
σ: bending shear stress [N/mm^2]
q: stress correction factor
(dependent on the ratio D/d)

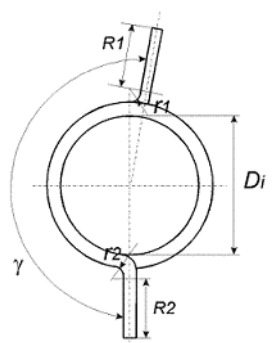
49.3 Spring design

In order to prevent friction, the coils either do not touch each other or under only slight stress. For the biggest achievable winding clearance the following applies:

$$a_{\max} = (0.24 \cdot w - 0.64) \cdot d^{0.83}$$

Generally, leg springs are wound. There are two options for the leg design: they can be either bent with offset (the radius must be specified) or tangential.





with tangential legs

with offset legs

49.4 Assumptions made for the calculation

The calculations apply only to leg springs with fixed or circular guided spring ends. If the leg is not fixed, the spring must be guided by means of a pin or sleeve.

49.5 Materials

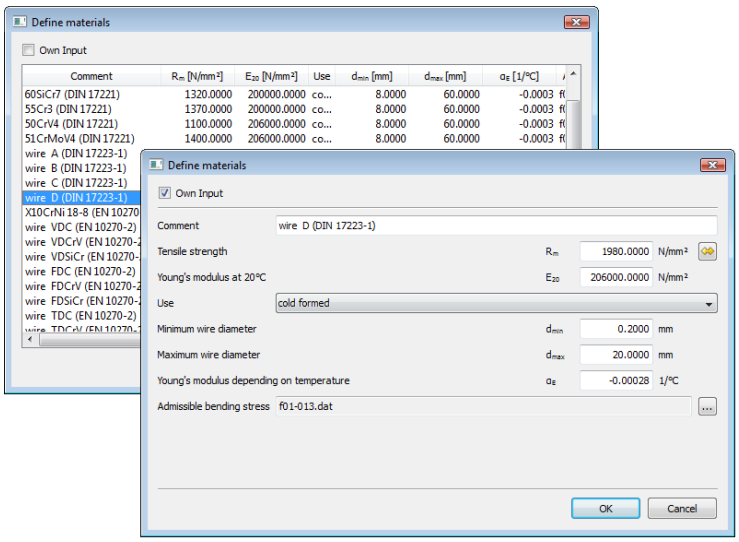


Figure 48.3: Materials screen: Leg springs

The selection list includes materials from the DIN 17221, DIN 17223-1, DIN 10270-1, DIN 10270-1 and DIN 10270-3 standards.
If you have set the `Own Input` flag, a new dialog appears here. This displays the material data used in the calculation which you can specify to suit your own purposes. You can also define your own materials directly in the database (see page I-114) so that these can also be used in future calculations.

49.6 Tolerances

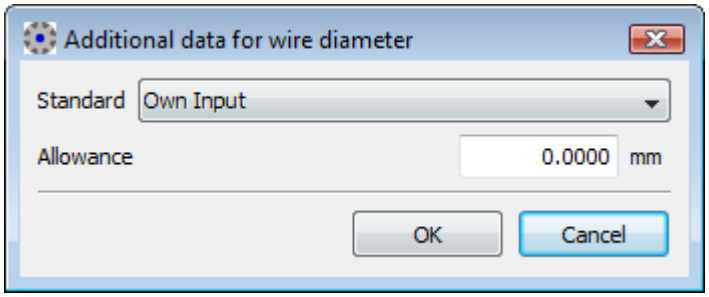


Figure 48.4: Additional wire diameter data for leg springs

Click the button next to the `Wire diameter` field to open the Tolerances screen. In this screen you can select a wire diameter as defined in DIN 2076 (1984), DIN 2077 (1979), EN 10270-1 (2001), EN 10270-2 (2001), EN 10270-3 (2001) or input your `Own input` to enter your own value.
If you select a wire diameter tolerance in accordance with the standard, the tolerance will be inserted directly in the screen.
If you select `Own input`, you can define the value yourself.

In the `Tolerances` list in the basic data you can choose one of the quality standards in accordance with DIN 2194 (2002)[17].

49.7 Drawing data

To access the spring data required to create a drawing, click `Drawing data`. Use the `F10SPRING?.RPT` file (for compression springs), or the `F20SPRING?.RPT` file (for tension springs), etc. (? = d/e/f/i/s for the required language) to modify the template to your own requirements.

49.8 Sizings

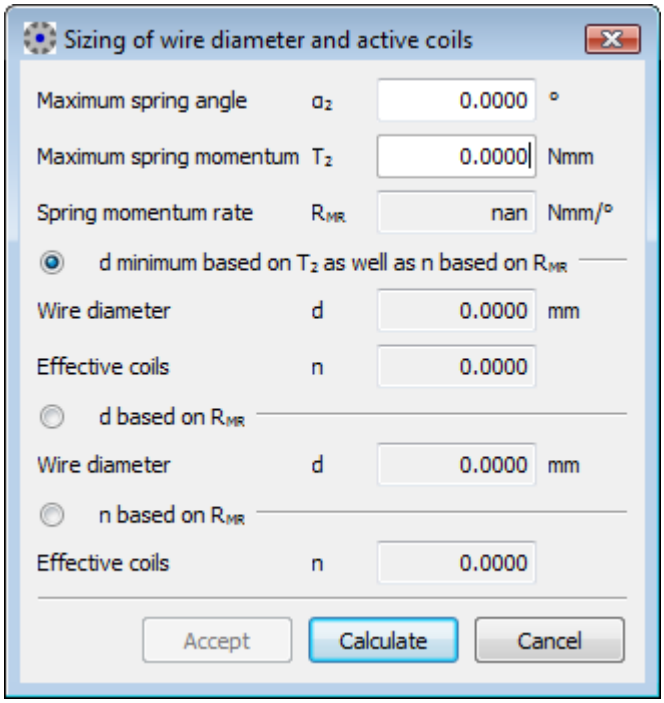


Figure 48.5: Sizing: Leg springs

Click the buttons next to the `Wire diameter` and `Effective coils` fields to use the spring momentum rate $RMR = \Delta M/\Delta\alpha$ to calculate the number of turns n for the predefined wire diameter. The program also suggests a value for the minimum wire diameter and the associated number of turns. The minimum wire diameter here is defined by the strength of the material.

Chapter 49
Disc springs

The calculation for disc springs is described in DIN 2092 (2006)[12]. The mass and quality requirements are handled in accordance with DIN 2093 (2006)[13].

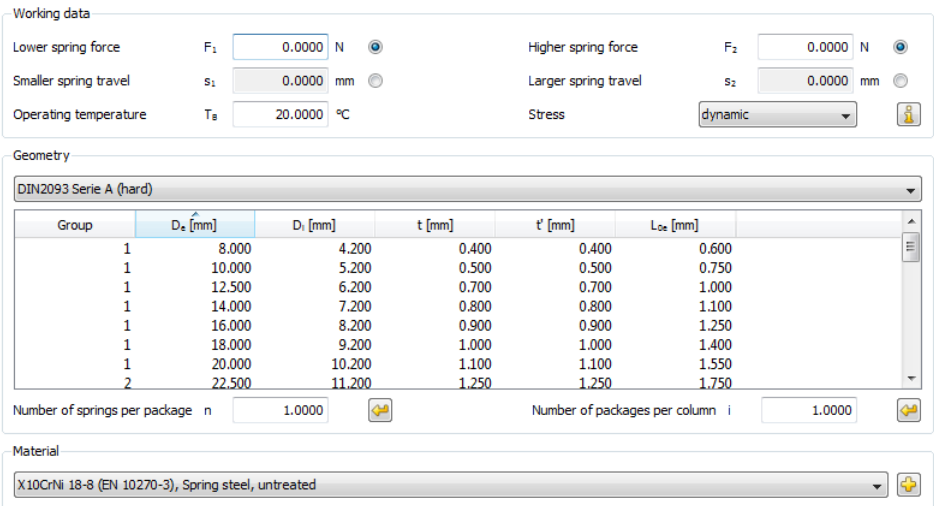


Figure 49.1: Basic data: Disc springs

Operating data

When you specify a load, you can use your own value as the spring force or travel. You can also specify whether the spring is to be subject to static, quasistatic, or dynamic, stress.

The calculations specified in DIN 2092 are for disc springs with or without bearing areas for the ratios 16 < De/t < 40 and 1.8 < De/di <2.5 and materials specified in DIN 2093.

Geometry

As specified in DIN 2093, disc springs are divided into 3 groups and 3 sequences. Groups 1 and 2 contain the springs without a bearing area, whereas group 3 has the springs with a bearing area. The disc thickness for group 1 is less than 1.25 mm, in the group 2 it is between 1.25 and 6 mm, and in group 3 it lies between 6 and 14 mm. The sequences differ according to spring hardness. Series A includes hard springs, i.e. they can withstand larger forces in a smaller travel of spring. This is followed by series B and series C which can withstand the least force in a larger

travel of spring. If you select `Own input`, the input fields for geometry data become active and you can therefore enter your own values here. This type of calculation only applies to springs without a bearing area, because the ratio of the thicknesses t'/t are not known, but it is still required for the calculation.

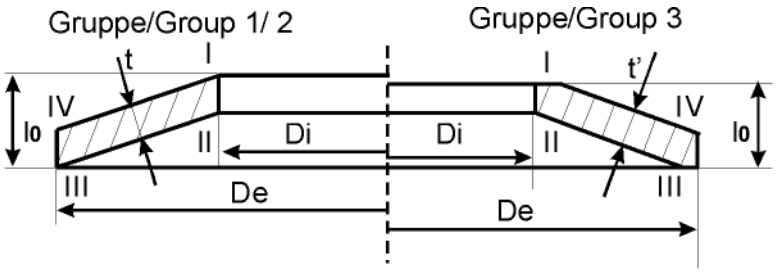


Figure 49.2: Dimensions of the disc springs

50.1 Strength values

In the case of springs that are subject to static or quasistatic load ($N <= 10^4$), the maximum force on the spring is calculated. The formula is predefined in DIN 2092. This force then is compared to the effective force Fn of the spring (at s = $0.75 \cdot h_0$) and the utilization of the spring force is calculated. If the required force is greater than the spring's effective force, Fn, the deviation in the calculation is too large. The DIN formula for calculating force only applies where the travel of the spring is $s = 0.8 \cdot h_0$. The springs can be used in packages or columns to handle larger forces. The calculation used to ascertain the overall force of the system is then represented in a force travel diagram. The calculation for disc springs is performed without taking friction into account.
Furthermore, the stresses are calculated at edge points I to IV. Points I and IV are subjected to compression, and points II and III are put under tension. Under dynamic load, the stress range is calculated using the maximum stress (either at point II or III) with the relevant lower stress level. The permissible permanent stress range is defined using a Goodman diagram. These values are then compared to give the number of load cycles the spring should be able to withstand under load. DIN 2093 includes Goodman diagrams which are only valid for materials specified in DIN 17221 and 17222. If any other materials are involved, you must contact the spring manufacturer.

50.2 Stress values

Stresses are calculated for edge points I to IV.

Point I:

$$\sigma_I = \frac{4 \cdot E}{1-\mu^2} \cdot \frac{t^2}{K_1 \cdot D_e^2} \cdot K_4 \cdot \frac{s}{t} \cdot \left[K_4 \cdot K_2 \cdot \left(\frac{h_0}{t} - \frac{s}{2 \cdot t} \right) + K_3 \right] \tag{49.1}$$

Point II:

$$\sigma_{II} = \frac{4 \cdot E}{1-\mu^2} \cdot \frac{t^2}{K_1 \cdot D_e^2} \cdot K_4 \cdot \frac{s}{t} \cdot \left[K_4 \cdot K_2 \cdot \left(\frac{h_0}{t} - \frac{s}{2 \cdot t} \right) - K_3 \right] \tag{49.2}$$

Point III:

$$\sigma_{III} = \frac{4 \cdot E}{1-\mu^2} \cdot \frac{t^2}{K_1 \cdot D_e^2} \cdot K_4 \cdot \frac{1}{\delta} \cdot \frac{s}{t} \cdot \left[K_4 \cdot (K_2 - 2 \cdot K_3) \cdot \left(\frac{h_0}{t} - \frac{s}{2 \cdot t} \right) - K_3 \right] \tag{49.3}$$

Point IV:

$$\sigma_{IV} = \frac{4 \cdot E}{1-\mu^2} \cdot \frac{t^2}{K_1 \cdot D_e^2} \cdot K_4 \cdot \frac{1}{\delta} \cdot \frac{s}{t} \cdot \left[K_4 \cdot (K_2 - 2 \cdot K_3) \cdot \left(\frac{h_0}{t} - \frac{s}{2 \cdot t} \right) + K_3 \right] \tag{49.4}$$

σ_I - σ_{IV} : Stress at points I-IV [N/mm^2]
E: Young's modulus [N/mm^2]
μ: Poisson's ratio [-]
D_e: External diameter [mm]
s: Spring travel of an individual disc [mm]
t: Thickness of an individual disc [mm]
h_0: Travel of spring until flat [mm]
δ: Diameter ratio (D_e/D_i)
K_1 - K_4: Variables calculated from formulae (DIN 2092)

50.3 Materials

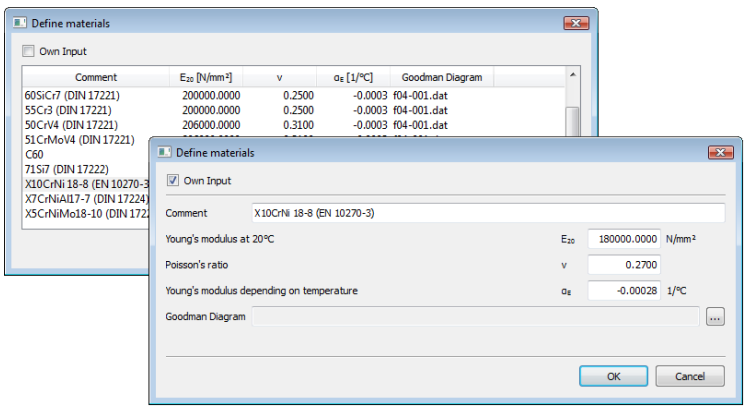


Figure 49.3: Materials screen: Disc springs

The selection list includes materials from the DIN 17221, DIN 17222, DIN 17224 and DIN 10270-3 standards.

If you have set the `Own Input` flag, a new dialog appears here. This displays the material data used in the calculation which you can specify to suit your own purposes. You can also define your own materials directly in the database (see page I-114) so that these can also be used in future calculations.

50.4 Calculate number

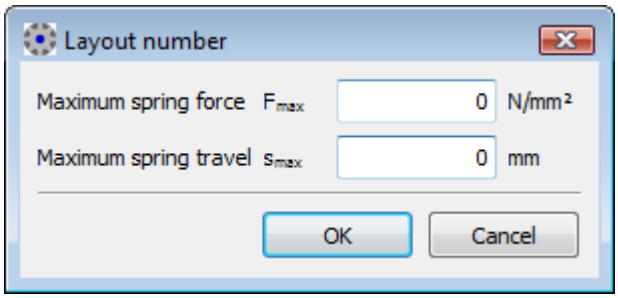


Figure 49.4: Sizing the number of packages/ columns

To estimate the number of discs or columns required, click the button next to the fields for number of springs per package or number of packages per column. In this screen, you can define the maximum force and the maximum travel can be defined. These values are then used to calculate and display the number of springs per package or the number of packages per column.

50.5 Limit dimensions

For all disc springs, the external diameter D_e must lie in a tolerance field h12 and the inside diameter D_i must lie in a tolerance field H12.

Chapter 50
Torsion-bar springs

The calculation of torsion-bar springs is described in DIN 2091 (1981)[11].

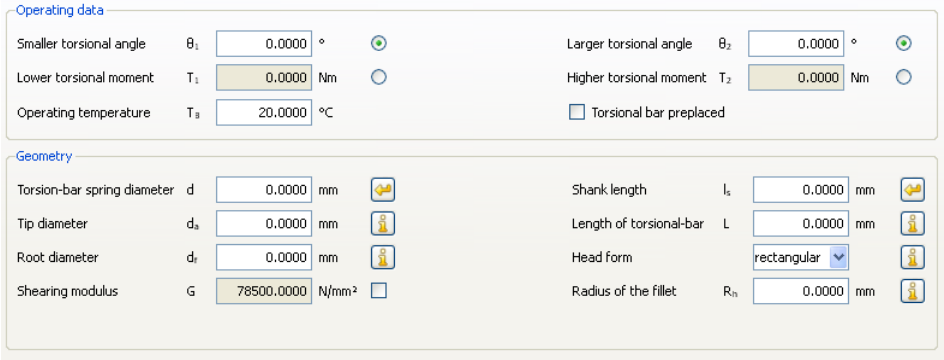


Figure 50.1: Main screen: Torsion-bar calculation

Operating data

When you specify the load, you can enter a value for either an angle of rotation or a torsional moment.

If a torsion-bar is set as the default (`Torsional bar preplaced` flag) the permitted shear stress of the torsion-bar, τ_{zul}, is increased.

Geometry

Enter specific parameters to define the geometry of the spring.
For the `toothed` head form, you must also specify the number of teeth, although this is purely for documentation and is not used for calculation. The standard assumes shearing modulus G as a constant. However, the calculation is still permitted even if this value is slightly different.

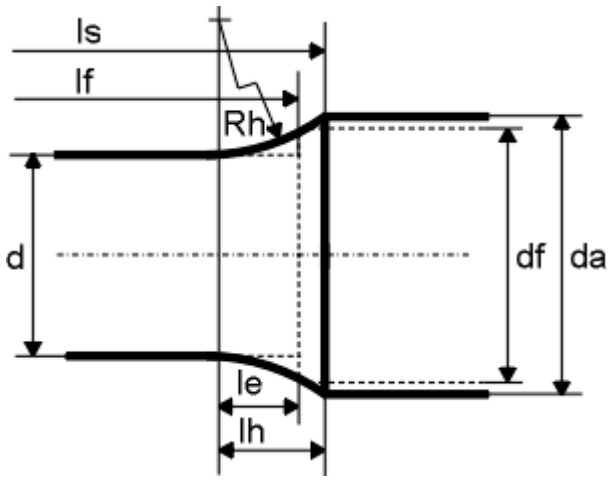


Figure 50.2: Defining a torsion-bar

51.1 Head forms

Torsion-bar springs as defined in DIN 2091 can have one of three different head forms: rectangular, hexagonal and toothed. Toothed torsion-bar heads are usually produced in accordance with DIN 5481 Part 1 or SAE J 498 b. However, they can also be manufactured with special toothings. The body forms of the heads apply only to bars that are loaded in the sense of rotation. Alternating loads require special design measures.

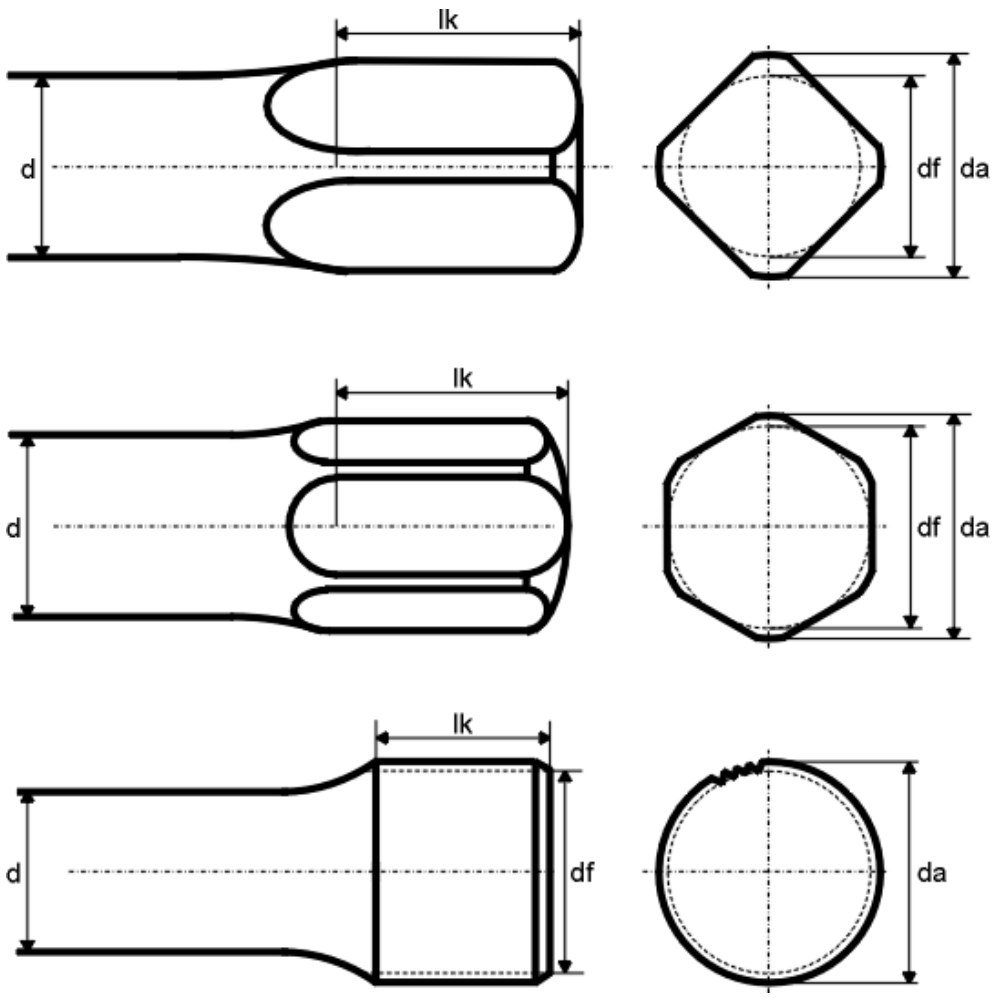


Figure 50.3: Forms of torsion-bar heads (rectangular, hexagonal, toothed)

51.2 Strength values

Definitions in DIN 2091:

DIN 2091 only applies to the materials defined in DIN 17221.
The permitted shear stresses:
For non-preloaded bars: $\tau zul = 700\ N/mm^2$
For preloaded bars: $\tau zul = 1020\ N/mm^2$
The heat treatment strength for these values is: $Rm = 1600 - 1800\ N/mm^2$
For the shearing modulus, $G = 78500\ N/mm^2$ is used as the default. Due to a preload (above the yield point, deformed in the direction of working stress) after the torsion-bar springs have been heat treated, there will be a better distribution of the operating stresses, and a relief in the boundary zone will be achieved.

51.3 Shear stress

Calculating shear stress τ:

$$\tau = \frac{16 \cdot T}{\pi \cdot d^3} \tag{50.1}$$

τ Shear stress [N/mm^2]
T: Torsional moment [Nm]
d: Wire diameter [mm]

51.4 Limiting values

The following limit ratios for torsion-bar heads apply to torsion-bar springs:
rectangular, toothed: $df/d > 1.3$; hexagonal: $df/d > 1.25$
The strength values from the DIN standard apply to bar diameters 10 to 60 mm.
The guide value is a head length between 0.5 and $1.5 \cdot d$.
The Rh/d ratio should be between 1 and 50.

51.5 Sizings

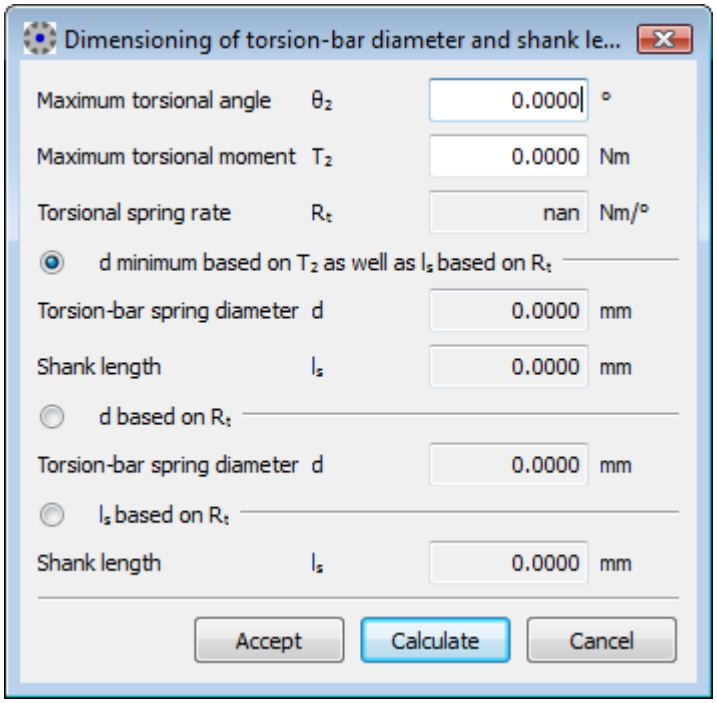


Figure 50.4: Sizings screen: Torsion-bar springs

Click the buttons next to the `Wire diameter` and the `Shaft length` fields to open the screen described above. You can enter the torsional moment and the angle of rotation in the sizing screen. These values are used to calculate the torsional spring rate which is then used to size bar diameter d or shank length ls. When sizing d and ls, you must first calculate d with the permitted shear stress value. You can then calculate the shank length ls from the bar diameter d. Various values are assumed so that you can size the dimensions. (rectangular, toothed: df = 1.35·d; hexagonal: df = 1.3·d; da = df + df/7; Rh = (da-d)·1.2)
These values are not transferred to the main screen.

Part VI

Belts and chain drives

Chapter 51
V-belt

Preamble:
Follow the manufacturer’s instructions when sizing and verifying V-belt drives. Most catalogs detail the entire calculation method. As the capacity of V-belts to transfer power increases, due to improvements in their materials and flank shapes, manufacturers' data provides the only really reliable values.

Fully automated calculation including standard V-belt lengths and standard effective diameters. Determining transmittable power per belt taking into account the speed, effective diameter, transmission speed ratio and belt length. All the data is taken from manufacturers tables (for example, ContiTech). This also includes a belt stress calculation module that uses data from belt-bending tests. This calculates the end of rope force and axis load at standstill and in operation for the optimum setting as well as for a setting in accordance with data in the catalogs.

As a variant, the calculation can also be performed with a third roller (tensioning pulley). You define the X and Y coordinates of the tensioning pulley in the **V-belt** tab. If you open the **Configuration** tab you can use the mouse to move the tensioning pulley. In this case, the particular X and Y value is displayed in the status row. This roller can be positioned outside or inside as required.

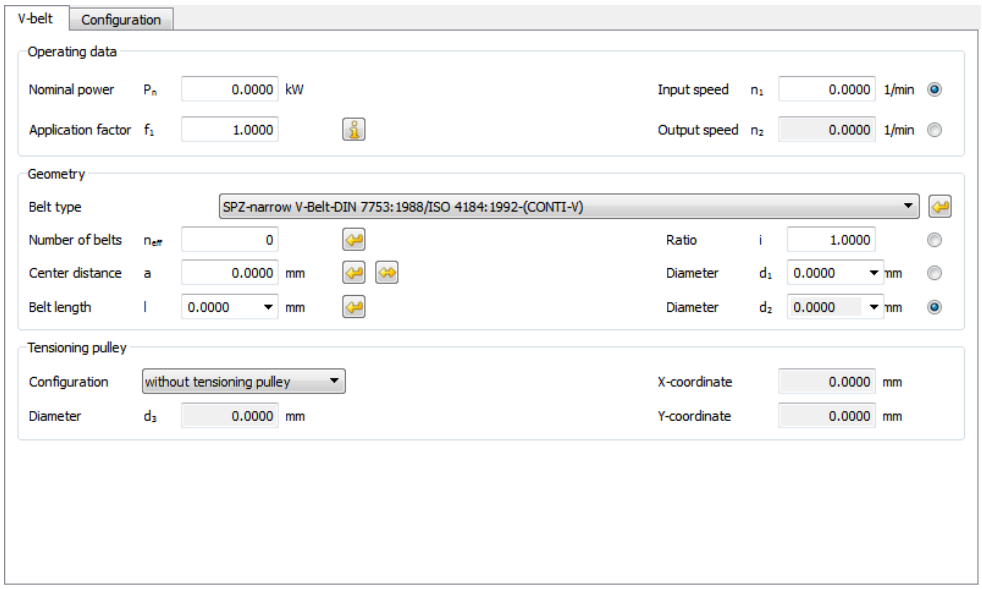


Figure 51.1: Basic data: V-belt calculation

52.1 V-belts data

KISSsoft stores the tabular values (catalog data) in files which you can then edit. You will find these file names in the KISSsoft database tool for the corresponding belt types (for example, Z090-015.dat for XPA narrow V-belts).

52.2 V-belt standards

The following belt types are available:

- XPA High-performance V-belts DIN7753/ISO4184 (CONTI-FO-Z)
- XPB High-performance V-belts DIN7753/ISO4184 (CONTI-FO-Z)
- XPC High-performance V-belts DIN7753/ISO4184 (CONTI-FO-Z)
- XPZ High-performance V-belts DIN7753/ISO4184 (CONTI-FO-Z)
- 5/- High-performance V-belts DIN7753/ISO4184 (CONTI-FO-Z)
- 6/Y- High-performance V-belts DIN7753/ISO4184 (CONTI-FO-Z)
- 8/- High-performance V-belts DIN7753/ISO4184 (CONTI-FO-Z)
- SPZ Narrow V-belt DIN7753/ISO4184 (CONTI-V)
- SPA Narrow V-belt DIN7753/ISO4184 (CONTI-V)
- SPB Narrow V-belt DIN7753/ISO4184 (CONTI-V)
- SPC Narrow V-belt DIN7753/ISO4184 (CONTI-V)
- 8/- Multiflex V-belt DIN7753/ISO4184 (CONTI-V STANDARD)
- 10/Z Multiflex V-belt DIN7753/ISO4184 (CONTI-V STANDARD)
- 13/A Multiflex V-belt DIN7753/ISO4184 (CONTI-V STANDARD)
- 17/B Multiflex V-belt DIN7753/ISO4184 (CONTI-V STANDARD)
- 20/- Multiflex V-belt DIN7753/ISO4184 (CONTI-V STANDARD)
- 22/C Multiflex V-belt DIN7753/ISO4184 (CONTI-V STANDARD)
- 25/- Multiflex V-belt DIN7753/ISO4184 (CONTI-V STANDARD)
- 32/D Multiflex V-belt DIN7753/ISO4184 (CONTI-V STANDARD)
- 40/E Multiflex V-belt DIN7753/ISO4184 (CONTI-V STANDARD)
- 3V-9J Force-belts
- 5V-15J Force-belts
- 8V-25J Force-belts

- 3V-9N Narrow-V-belt USA standard
- 5V-15N Narrow-V-belt USA standard
- 8V-25N Narrow-V-belt USA standard

52.3 Configuring Tensioning Pulleys

Here you can select one of the following:

- non-tensioning pulley
- inside tensioning pulley
- outside tensioning pulley

If you selected an inside/outside tensioning pulley, you can input the conical spring washer diameter and the position (x/y) of the tensioning pulley. In the `Configuration` tab you can position the tensioning pulley interactively using the mouse (the x and y values are displayed in the status row).

52.4 Application factor f1

You can enter this factor in the basic data screen. If you selected a configuration with a tensioning pulley, you should increase factor f1 by 0.1. The table shown below is used to define the f1-factor (refer to the catalogs for more information):

	Drive motor with normal starting torque, combustine engines and turbines with n over 600 min^{-1} for daily operation hours:			Drive motor with high starting torque, combustine engines and turbines with n to 600 min^{-1} for daily operation hours:		
	to 10 h	over 10 h to 16 h	over 16 h	to 10 h	over 10 h to 16 h	over 16 h
Slightly drives	1	1,1	1,2	1,1	1,2	1,3
Medium-heavy drives	1,1	1,2	1,3	1,2	1,3	1,4
Heavy drives	1,2	1,3	1,4	1,4	1,5	1,6
Very heavy drives	1,3	1,4	1,5	1,5	1,6	1,8

Figure 51.2: Application factor V-belts

52.5 Center distance

The minimum center distance is calculated from the two belt sheave diameters. You cannot enter a smaller value here. The sheaves must not touch each other during operation.

The conversion processes uses a suggestion for new designs in accordance with the ContiTech catalog. Here:

```
0.7*(d1+d2) <= a <= 2.0*(d1+d2)
```

In KISSsoft, the average coefficient for the suggested value is used in the conversion.

```
a = 1.35*(d1+d2)
```

52.6 Belt length

You need to know the belt length before you can calculate a V-belt. If you have not specified a length, or if you change to a configuration that involves a tensioning pulley, you must ensure that the program recalculates the belt length.

52.7 Effective number of V-belts

The effective number of V-belts is calculated from the theoretical number by rounding this value up to the next highest whole number.

52.8 Tensioning pulley diameter

The tensioning pulley diameter should be at least as big as the smallest belt sheave. If at all possible, you should not use tensioning pulleys, in particular outside tensioning pulleys. However, if you have to use a tensioning pulley, its diameter should be at least 1.33·d if it is an outside pulley or 1.0·d if it is an inside pulley (d: diameter of the smaller sheave).

Every manufacturer provides slightly different information about tensioning pulleys.

52.9 Position of tensioning pulley (x/y)

When you configure the tensioning pulley, you can enter the position of the pulley (in X/Y coordinates). Here, the axis of the small sheave is the origin of the coordinates system. If you use the mouse to change the position in the `Configuration` tab you can only move the tensioning pulley within the valid area.

52.10 Inspecting V-belts

(belt bending test)

The actual axial stress of V-belt drives is calculated from data provided by the belt bending test. Enthusiastic mechanics have a tendency to over-tension belt drives, and therefore subject them to loads that are too high for their capabilities.

Chapter 52
Toothed belts

Use this method to calculate and size all aspects of toothed belt drives, including the tooth number and belt length, whilst taking into account standard numbers of teeth. When you enter the required nominal ratio and/or the nominal distance of axes, the program calculates the best possible positions. You can also calculate the required belt width, taking into account the correction factors, the minimum tooth numbers, and the number of meshing teeth. You can also print out assembly details (belt bending test). The data for each type of belt is saved to self-describing text files which can be edited as required.
You can also perform calculations for special stress-resistant toothed belts with integrated steel ropes (e.g. AT5mm).

As a variant, the calculation can also be performed with a third roller (tensioning pulley). You define the X and Y coordinates of the tensioning pulley in the **Toothed belts** tab. If you open the **Configuration** tab you can use the mouse to move the tensioning pulley. In this case, the particular X and Y value is displayed in the status row. This roller can be positioned outside or inside as required.

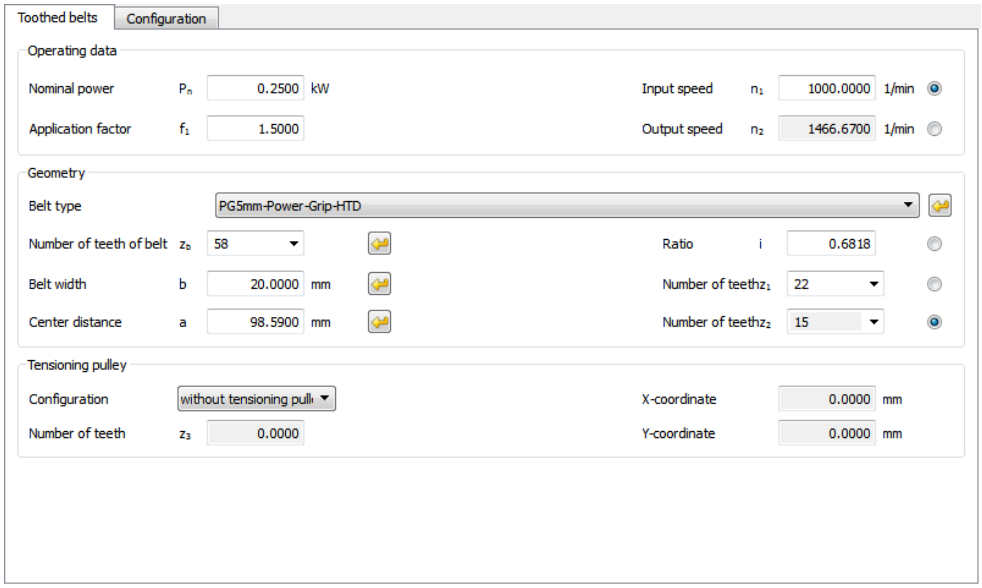


Figure 52.1: Basic data: Toothed belt calculation

53.1 Technical notes (toothed belts)

Preamble:
Follow the manufacturer’s instructions to achieve the best results when sizing and verifying toothed belt drives. Most catalogs detail the entire calculation method. As the capacity of V-belts to transfer power increases, due to improvements in their materials and flank shapes, manufacturers' data provides the only really reliable values.

Elasticity:
As the manufacturers catalogs provide very little data on this subject, you must treat the belt elasticity constraint values with caution. The elasticity (in N) is the force required to lengthen the belt by 100%.

Weight:
As the details provided in manufacturers catalogs about this subject are not complete, you must treat these values with caution.

Pretensioning the belt:
As the manufacturers catalogs provide very little data on this subject, you must treat the constraint values with caution. The calculation method and the factors it uses are stored in the Z091-0??.DAT files where they can be changed if required. You can use one of the following procedures to calculate the required pretensioning values for various types of belts. The data here is taken from the catalogs:

Belt type:	**Pretension:**	
Breco AT5, AT10, AT20	0.5	* Circumferential force
Synchroflex AT3, AT3 GIII, AT5 GIII, AT10 GIII	0.5	* Circumferential force
Isoran XL, L, H, 8, 14	0.625	* Circumferential force
HTD 3, 5, 8, 14	0.25	* Max. permitted circumferential force
8MGT, 14MGT Poly Chain GT2	0.5	* Circumferential force
RPP-HPR 8, 14	0.5	* Circumferential force

Table 52.1: Pretension

Forces in no load/load are calculated in accordance with [66], equation 27/23.

$$F_V = \frac{F_1 + F_2}{2} \tag{52.1}$$

$$F_1 = F_V + \frac{F_t}{2} \tag{52.2}$$

$$F_2 = F_V - \frac{F_t}{2} \tag{52.3}$$

53.2 Toothed belt standard

You can select one of these standards:

- XL-ISORAN RPP (FENNER)
- L-ISORAN RPP (FENNER)
- H-ISORAN RPP (FENNER)
- 8mm ISORAN RPP (FENNER)
- 14mm ISORAN RPP (FENNER)
- 8mm-ISORAN-RPP-GOLD (Megadyne)
- 14mm-ISORAN-RPP-GOLD (Megadyne)
- 8mm-ISORAN-RPP-SILVER (Megadyne)
- 14mm-ISORAN-RPP-SILVER (Megadyne)
- RP8mm-Pirelli RPP-HPR
- RP14mm-Pirelli RPP-HPR
- PG3mm-Power Grip-HTD
- PG5mm-Power Grip-HTD
- PG8mm-Power Grip-HTD
- PG14mm-Power Grip-HTD
- 8mm-DAYCO-RPP-(Panther)
- 8MGT-Poly-Chain-GT-Carbon (Gates)
- 14MGT-Poly-Chain-GT-Carbon (Gates)
- 8mm MGT-Poly Chain-GT2

- 14mm MGT-Poly Chain-GT2
- AT3mm-SYNCHROFLEX
- AT3mm GEN III-SYNCHROFLEX
- AT5mm GEN III-SYNCHROFLEX
- AT10mm GEN III-SYNCHROFLEX
- AT5mm-BRECOflex
- AT10mm-BRECOflex

AT20mm-BRECOflex*Additional standards on request.*

53.3 Possible Sizings/Suggestions

The following sizings are possible if you select the different buttons:

Variable	Influencing/necessary variables
Belt profile	Power Speed (small disc) Operating factor
Number of teeth on belt	Center distance Number of teeth on sheave
Center distance	Number of teeth on belt Number of teeth on sheave (all)
Ratio	Center distance Nominal ratio Speed (small disc)
Number of teeth on tensioning pulley	Number of teeth (small disc)

Table 52.2: Possible sizings

53.4 Configuring Tensioning pulleys

Here you can select one of the following:

- non-tensioning pulley
- inside tensioning pulley
- outside tensioning pulley

If you selected an inside/outside tensioning pulley , you can input the conical spring washer diameter and the position (x/y) of the tensioning pulley. In the `Configuration` tab you can position the tensioning pulley interactively using the mouse (the x and y values are displayed in the status row).

53.5 Application factor and summand for operational behavior

You can either enter the application factor manually in the load factor interface, or have the program define it from the operating parameters. If you selected a configuration with a tensioning pulley, you must increase the operating factor by 0.1. Use the data in this table to define the factor (refer to the catalogs for more information):

Driven machine	**Operating hours per day**		
	0-10	**10-16**	**16-24**
Light drive	1.2	1.3	1.4
Medium-light drive	1.4	1.5	1.6
Medium-heavy drive	1.5	1.6	1.7
Heavy drive	1.7	1.8	1.9
Heavyweight drive	1.8	1.9	2.0

Table 52.3: Application factors

Summand for operational behavior
(This summand is added to the operating factor in the calculation)

Operational behavior:	**Summand**
continuous, 0-10 hr/day	0
continuous, 10-16 hr/day	+0.1
continuous, 16-24 hr/day	+0.2
intermittent or with alternating load	-0.1

Table 52.4: Summand

53.6 Center distance

The minimum center distance is calculated from the two belt sheave diameters. You cannot enter a smaller value here. The sheaves must not touch each other during operation.

53.7 Belt length and number of teeth on belt

In toothed belt drives the number of teeth on the belt is used to define the belt length. You need this value when you perform the calculation for the belt. If you did not specify the number of teeth on the belt or, if you switched to configuration with a tensioning pulley, you must ensure that the program recalculates the value for the number of teeth on the belt.

53.8 Effective belt width

The theoretical belt width (minimum width required to transmit the torque) can be calculated from the data in the manufacturer catalogs. The effective belt width is then taken as the next largest standard belt width.

As a general rule, the belt width should not be larger than 5*pitch. A warning message appears if you select a belt that is either too wide or too narrow. Although the calculation continues, you use the data it provides at your own risk.

Defining the effective belt width/factor for the belt width:

To define the belt width, you will need the belt width factor (f_b). Use this formula to calculate this factor:

$$f_b = \frac{Betriebsleistung\ /\ operating\ power}{Nennleistung\ nach\ Katalog\ /\ Nominal\ power\ according\ to\ catalog\ e} \tag{52.4}$$

The nominal power as specified in the catalog is a tabular value taken from the manufacturers' catalogs and is dependent on the speed and number of teeth on the smaller belt sheave.

With the calculated factor *f_b* you can then define the effective belt width from a catalog table. However, if the value of *f_b* does not match a standard belt width, the next biggest width will be used.

Remarks:
The theoretical belt width in the KISSsoft calculation reports corresponds to an interpolated value, according to calculated factor *f_b*.

KISSsoft stores the tabular values (catalog data) in files which you can then edit. Use the KISSsoft database tool to find the exact file name for a specific belt type (e.g. Z091-001.DAT for XL-Isoran).

53.9 Tensioning pulley tooth number

The value you use for the number of teeth on a tensioning pulley should be at least as large as the value given for the diameter of the smallest belt sheave.

Where possible, tensioning pulleys should be inside toothed lock washers. However, if necessary, they can also be used as smooth sheaves from outside. The diameter of the tensioning pulley should be at least 1.2 *d if positioned outside, or 1.0 *d if positioned inside (d: diameter of the smaller sheave). The data provided by each belt manufacturer about tensioning pulleys varies greatly.

For Poly Chain GT:

An outside tensioning pulley reduces service life and should be avoided if possible.

For AT belts:

AT5mm	with tensioning pulley inside:	25 mm ($z > 5$)
	with tensioning pulley outside:	50 mm ($z > 10$)
AT10mm	with tensioning pulley inside:	50 mm ($z > 5$)
	with tensioning pulley outside:	120 mm ($z > 12$)
AT20mm	with tensioning pulley inside:	120 mm ($z > 6$)
	with tensioning pulley outside:	180 mm ($z > 9$)

53.10 Position of the tensioning pulley x/y

You must enter this value when you configure a tensioning pulley. Here, the axis of the small sheave is the origin of the coordinates system. If you use the mouse to change the position in the `Configuration` tab you can only move the tensioning pulley within the valid area.

Chapter 53
Chain drives

Calculating chain drives with roller chains as defined in ISO 606 (with standardized roller chain values taken from a database). The chain geometry (center distance, number of chain elements) for simple and multiple chains and the transmissible power, axial forces, and variation in speed, are calculated by the polygon effect, etc. Basis: DIN ISO 10823, [38] and [64].
During this calculation the program checks the highest permitted speed and displays a suggested value for the required lubrication.

As a variant, the calculation can also be performed with a third roller (tensioning pulley). The X and Y coordinates of the tensioning pulley can be defined in the **Chain drives** tab. If you open the **Configuration** tab you can use the mouse to move the tensioning pulley. In this case, the particular X and Y value is displayed in the status row. This roller can be positioned outside or inside as required.

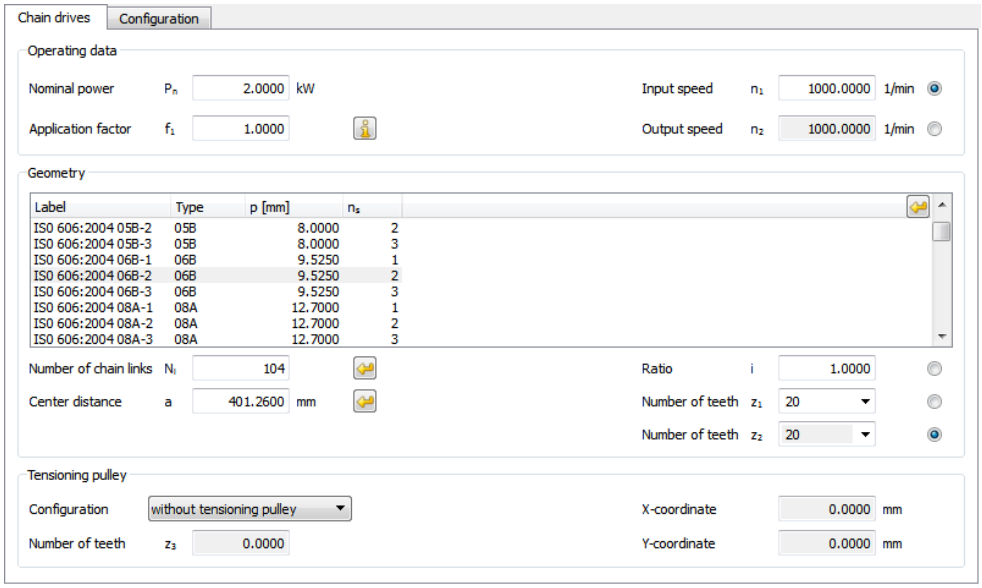


Figure 53.1: Basic data: Chain calculation

54.1 Sizings

Using the drive data as a starting point, the program displays a list of suggested values for suitable chain drives.

- Calculating the center distance from the chain length.
- Calculating the chain length from the center distance.

54.2 Tensioning pulleys

You require tensioning pulleys if you need to limit the chain deflection or keep to a minimum loop angle. You must arrange the tensioning pulleys under no load. They must have at least three teeth.

54.3 Standard

Chain profile standard:

- Roller chain ISO 606

The roller chain standard, ISO 606, includes chains as defined in the DIN 8154, 8187 and DIN 8188 standards. Roller chains are the most frequently used type of chain because lubricated rollers reduce noise and wear. The chains defined in DIN 8187 correspond to the European type, and those defined in DIN 8188 correspond to the American type. You should only install bush chains as defined in DIN 8154 in closed gear cases with sufficient lubrication.

54.4 Chain type

The data shown below depends on the type of chain:

- Chain pitch
- Maximum permitted speed of the small gear.
- Nominal power at maximum permitted speed.

Tables in ISO 606 pages 8 to 10.

54.5 Number of strands

You can achieve high levels of power by using multiple chains. Chains are often arranged in two or three strands (duplex, triplex). The values for duplex and triplex chains are also given in the same standard.

54.6 Application factor

Guide values in accordance with DIN ISO 10823, Table 2:

Procedure of the driven machine	Procedure of the driving machine		
	continuous, without impacts	light impacts	moderate impacts
continuous, without impacts	1,0	1,1	1,3
moderate impacts	1,4	1,5	1,7
heavy impacts	1,8	1,9	2,1

Figure 53.2: Application factor for chain calculation

54.7 Speed/number of teeth/transmission ratio

Range of ratio:

favorable i = 1. . . 5,

good i = 1. . . 7,

unfavorable i = > 10.

Number of teeth:
Due to the polygon effect, we recommend a minimum number of teeth of between 17 and . . 25. Tooth numbers of less than 17 should only be used to produce low levels of power. The preferred numbers of teeth for use in chain gears, as stated in ISO 606, are: 17, 19, 21, 23, 25, 38, 57, 76, 95, 114.

You should use at least three teeth for tensioning pulleys.

54.8 Configuration

You can select one of these configurations:

- without tensioning pulley
- with tensioning pulley inside
- with tensioning pulley outside

In a configuration involving tensioning pulleys, you must specify the number of teeth and the position of the tensioning pulley (x/y). In the `Configuration` tab

you can position the tensioning pulley interactively using the mouse (the x and y values are displayed in the status row).

54.9 Center distance

Recommended center distance:	$a = 30 \cdot p \ldots 50 \cdot p$ (p: pitch)
You should avoid:	$a < 20 \cdot p$ and $a > 80 \cdot p$

Click the button to calculate the center distance from the number of chain links.

54.10 Polygon effect

When calculating chains, you must take the polygon effect into account both for the reference circle and the center distance.

- Formula for the reference circle:

$$d = \frac{Teilung}{\sin\left(\frac{\pi}{z}\right)} \tag{53.1}$$

- (see also [66], equations 26/46)
- Formula for the center distance:
  The length of the loop on the chain wheel differs as follows from the formula used for V-belts/toothed belts:

$$l_{UK} = l_{UR} \cdot \frac{\sin\left(\frac{\pi}{z}\right)}{\frac{\pi}{z}} \tag{53.2}$$

- l_{UK}: Length of loop for chains
  l_{UR}: Length of loop for V-belts

54.11 Number of links

The number of links should usually be an even number.

Click the button to calculate the number of links from the center distance.

54.12 Geometry of chain sprockets

In KISSsoft, you can display and print out the geometry of chain sprockets as defined in ISO 606 as a graphic. The graphics are created with a mean deviation.

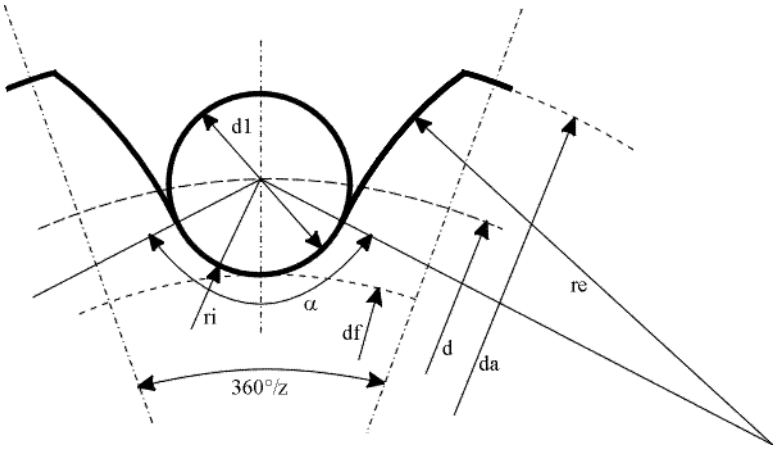


Figure 53.3: Geometry of chain sprocket

You can also output other values for a sprocket wheel in a report. The figures in this section show how specific information is represented in this report.

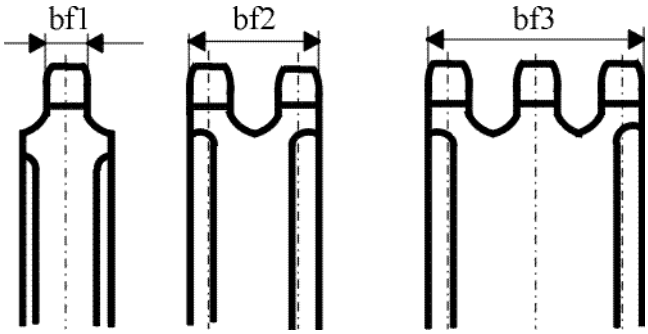


Figure 53.4: Chain sprocket width

Part VII

Automotive

Kapitel 54

Synchronization

Use this module to calculate the gear synchronization time and total time, based on the specified geometry, forces and application data. Some additional calculations for heat development, frictional power, and wear resistance, are also performed. Calculations can be performed for common types of synchronizations for a given number of cones (single, double or triple cone).

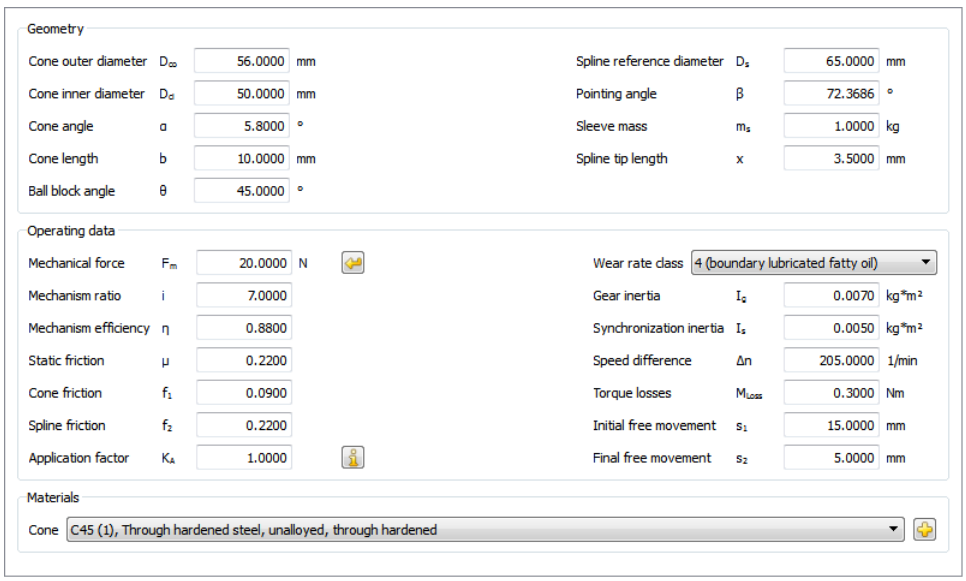


Figure54.1 Synchronization module tab

55.1 Geometry

Geometry data is needed for the synchronization ring, also called the cone. Additional data is needed for the spline shaft tip definition (the indexing) and ball block angle. This is the external ball angle which holds the synchronizer at its position (engaged or disengaged). Specific limit values have been defined for the angle input to ensure synchronization can be guaranteed.

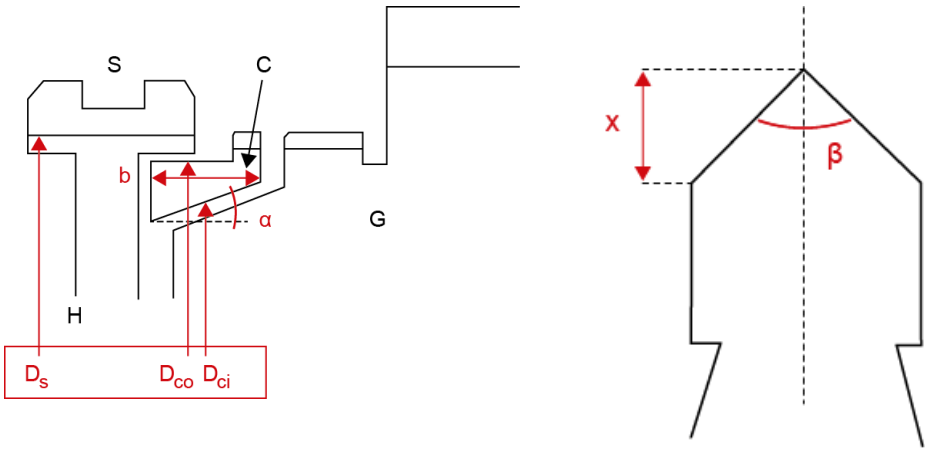


Figure 54.2: (a) Description of main geometry: S = Sleeve, C = Ring/Conus, H = Hub, G = Gear, (b) Spline shaft tip geometry

55.2 Operating data

The mechanical force is the force applied to the shifting handle. This force is multiplied by the mechanical ratio and applied to the sleeve. The friction coefficient at the beginning of the synchronization can be defined, for the conus and the sleeve.

The gear inertia and the speed difference are required entries. Torque losses due to mechanical friction, oil splashing, and other sources, can be defined. The defined losses during the shifting will either help or hinder the process depending on the shifting direction. If the synchronizer sleeve is subject to free movement before and after the actual synchronization, the distances can be entered here to enable the total time to be calculated correctly (from another gear to the final end position of the sleeve).

Kapitel 55
Friction clutches

This module is used to calculate friction clutches and brakes in accordance with VDI 2241 [90]. The results of this calculation can then be used to select a suitable clutch or brake. The clutches are operated either mechanically, electromagnetically, or by pressure (e.g. hydraulically), thereby either generating or removing pressing force. The clutches can be designed to run either dry or with lubrication. This has a significant effect on the coefficient of sliding friction and the coefficient of static friction.

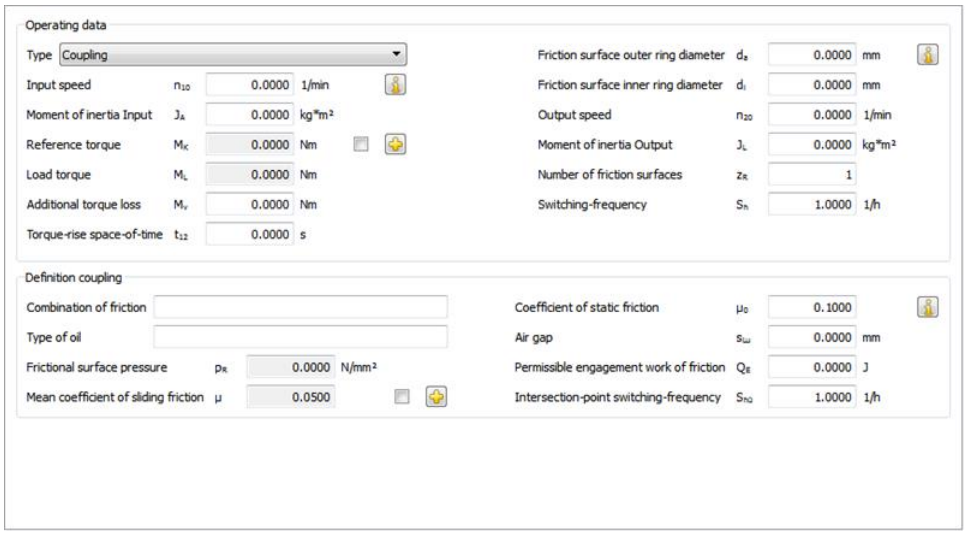


Figure 55.1: Basic data: Friction clutches

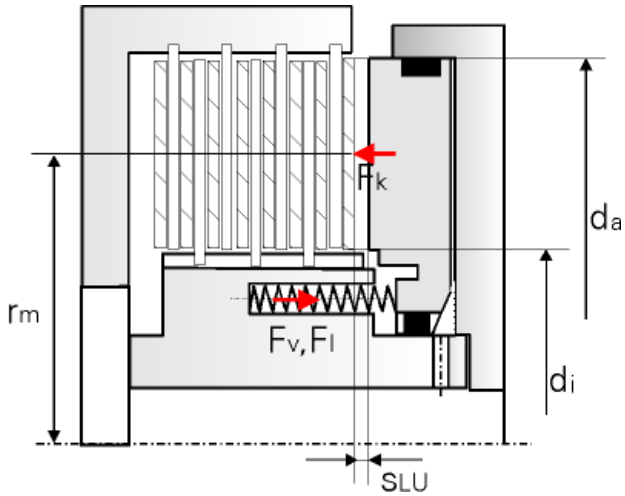


Figure 55.2: Display of a clutch

Force is stored in a spring. When the spring is released, the force returns the clutch to its open state. Compression or disc springs are usually used here. Both types of spring are pretensioned in their open state. In this example the compression is created hydraulically, and therefore affects the piston. This additional definition of storing force is not included in the VDI guideline. The guideline assumes that frictional surface pressure is applied directly to the plate. As the dynamic characteristics of the springs can also be non-linear, the force generated by the contact with the first plate is used in the calculation.

In KISSsoft, you can either define the spring forces or input the reference torque MK and the load torque ML directly. As specified in VDI 2241, the work of friction and the switching capacity are defined using an average sliding velocity and an average coefficient of sliding friction. You can also specify the coefficient of sliding friction as a dependency of 5 sliding velocities, because this coefficient can vary greatly depending on which sliding velocity is present. However, this does not take into account the aging of the oil, which would reduce the coefficient of sliding friction.

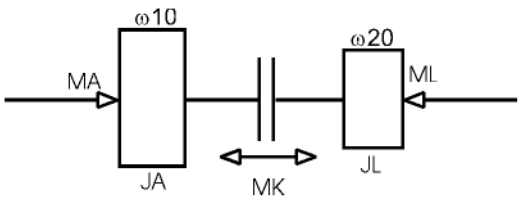


Figure 55.3: Schematic display of a clutch

The dynamic moment of inertia JL can also be made up of a number of different parameters. If there is a mass m at the distance r from the rotational axis, its moment of inertia can be calculated with $JL2 = r^2*m$.

This can then be added to the existing moment of inertia JL1.

JL =JL1 + JL2.

Ratios can then reduce the moment of inertia on the clutch shaft $J2red = J2*(n2/n1)^2$. This reduced moment of inertia can then be added to the clutch shaft's moment of inertia.

JL = JL1 + J2red

56.1 Calculation

Inputting the spring forces/defining the reference torque

If you decide to input the spring forces (Reference torque flag is set), the reference torque is calculated as follows:

$$F_{stat} = Fl$$

$$F_{dyn} = Fk - Fv$$

Fk (N): Piston force (FK = pK/AK)

pK (N/mm^2): Compression on the piston

AK (mm^2): Piston surface area

Fl (N): Spring force to plates contact

Fv (N): Pretension for spring force

F (N): Resulting force on the first plate (N)

The accelerating torque or the holding torque is then determined from this. Using the different coefficients of sliding friction, if these have been defined, otherwise using the mean coefficient of sliding friction:

$$M_A = F_{dyn} \cdot \mu \cdot r_m \cdot z_R$$

$$M_L = F_{stat} \cdot \mu 0 \cdot r_m \cdot z_R$$

MA (Nm): Accelerating torque

Fstat, Fdyn (N): Resulting force on the first plate

μ (-): Coefficient of sliding friction

rm (mm): average friction radius

zR (-): Number of friction surfaces (plates)

ML (Nm): Holding or load torque

μ0 (-): Coefficient of static friction

The reference torque is then defined from MA+ML. You can also define a torque loss, which has a negative prefix for a clutch and a positive prefix for a brake.

However, if you define the reference torque directly, you cannot also define a torque loss. This must then be taken into account with the reference torque.

The formulae specified in VDA 2241 [90] are used to define the sliding time t_3.

For a clutch:

MK=MA, with a specified MK, with the influence of t12 (in this sequence):

$$t_3 = J_L \cdot (\omega_{10} - \omega_{20})/(M_K - M_L)$$

$$t_3 = (J_A \cdot J_L)/(J_A \cdot (M_K - M_L) + J_L \cdot (M_K - M_A)) \cdot (\omega_{10} - \omega_{20})$$

$$t_3 = J_L \cdot (\omega_{10} - \omega_{20})/(M_K - M_L) + t_{12}/2 \cdot (M_L/M_K + 1)$$

For a brake:

with a specified MK, with the influence of t12 (in this sequence):

$$t_3 = J_L \cdot (\omega_{10} - \omega_{20})/(M_K - M_L)$$

$$t_3 = J_L \cdot (\omega_{10} - \omega_{20})/(M_K - M_L) + t_{12}/2 \cdot (M_L/M_K + 1)$$

The engagement work of friction Q is then calculated with, or without, taking the torque-rise time t12 into account, depending on whether or not this value has been defined. The switching capacity on the total friction surface and the maximum switching capacity are also calculated.

If you input curve points for the coefficient of sliding friction, the area below the calculated curve in the torque diagram is calculated as the engagement work of friction. The switching capacity is then derived from the time-based conclusion of this calculation.

Each of these values must be input as specific values for the friction surface because these are provided by the manufacturers in the relevant catalogs.

Furthermore, when you input the switching frequencies and the permitted engagement work of friction (one-time switching) the program calculates a utilization to show whether the selected coupling will be adequate.

$$Q_{zul} = Q_E \cdot (1 - e^{-Sh\ddot{u}/Sh})$$

Qzul (kJ): permissible engagement work of friction

QE (kJ): permissible engagement work of friction (one-time switching)

Shü (1/h): Intersection-point switching-frequency

Sh (1/h): Switching frequency per time unit

The utilization AQ is then determined from this permitted value and the calculated engagement work of friction:

$$A_Q = Q_{zul} / Q$$

When you select a clutch, you must take into account the reference torque, and most importantly, the permissible engagement work of friction QE (one-time

switching) and the calculated permissible engagement work of friction (for higher switching frequencies).

56.2 Definition of spring forces

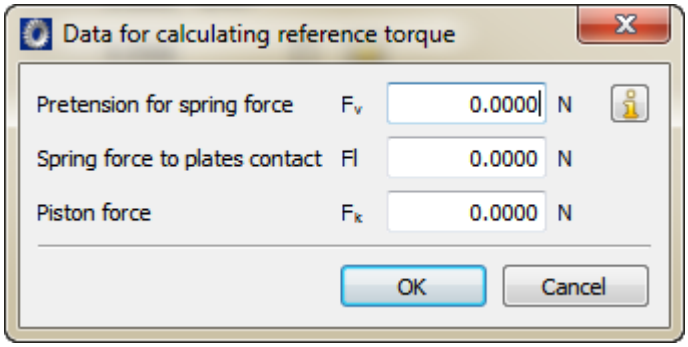


Figure 55.4: Definition of spring forces

These additional inputs, `Pretension for spring force Fv` and `Piston force Fk`, are used to determine the characteristic values required to calculate the resulting spring force. The coefficient of sliding friction and the average radius rm and the number of plates are then applied to determine the accelerating torque.

The coefficient of static friction from the `Spring force to plates contact Fl` is then used to define the holding torque.

56.3 Definition coefficients of sliding friction and velocities

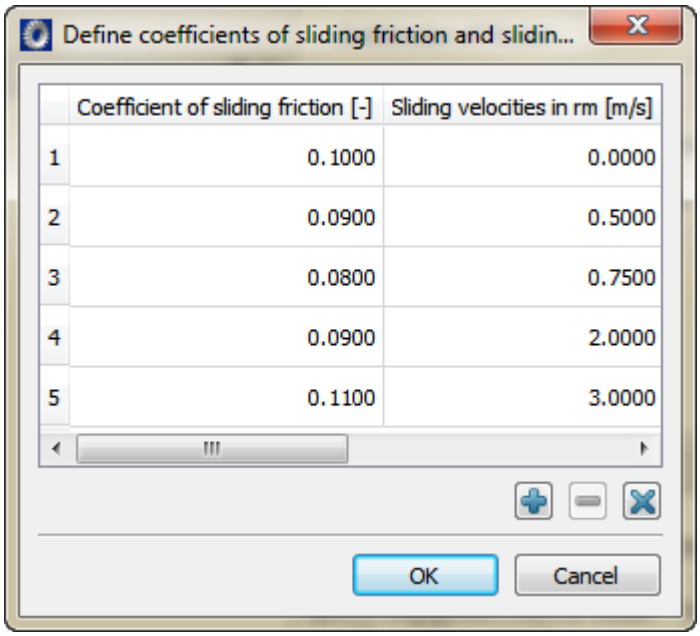


Figure 55.5: Definition coefficients of sliding friction and velocities

The coefficients of sliding friction are specified by the manufacturers in accordance with the sliding velocities. The VDI 2241 standard assumes that a constant value is used. However, this may result in a large deviation in results. By inputting a maximum of 5 points you can create a poly line that connects these points. From this line the program can then derive 10 values for the coefficients of friction in the sliding velocity areas at the start and at the end. The 10 different accelerating torques derived from this can then be used later on in the calculation.

56.4 Graphics

The graphics show the speed curve over sliding time t3, the torque diagram over sliding time t3, and the coefficient of sliding friction curve for the sliding velocity, of which a maximum of 5 points have been entered (if defined by the user).

56.5 Settings

If the `Use radius to plates gravity center for the calculation` flag is set, the radius at the center of gravity for the plates is used in the calculations instead of the mean radius of the plates rm. This radius is calculated with:

$$rs = \frac{2}{3} \cdot \frac{ra^3 - ri^3}{ra^2 - ri^3}$$

Part VIII

Various

Chapter 56
Tolerance calculation

In this module you enter the nominal lengths and their corresponding allowances for various elements. These values are then used to calculate an overall tolerance. This calculation uses a constant distribution (arithmetical sum) and the square root of the tolerance squares (standard distribution) to define the maximum and minimum size of the measurement chains. You can also use the appropriate dimensions to calculate the nominal length/expected value of the measurement chain. The `tolerance field specified in ISO` is defined in accordance with ISO 286 in which the tolerances are defined up to a size of <= 3150 mm. In KISSsoft, for tolerance classes H, h. JS and js, the values used in the standard are extrapolated up to a value of 10000 mm.

Basic data

General tolerance: DIN ISO 2768-f (fine)

	+/-	Nominal length [mm]	Tolerance type	Tolerance field	Upper allowance [mm]	Lower allowance [mm]	Comment
1	+	80.0000	Tolerance field according ...	H7	0.0300	0.0000	
2	-	16.0000	General tolerance		0.1000	-0.1000	
3	-	15.0000	General tolerance		0.1000	-0.1000	
4	-	14.0000	General tolerance		0.1000	-0.1000	
5	-	16.0000	Tolerance field according ...	H7	0.0180	0.0000	

Figure 56.1 Basic data

Chapter 57
Strength verification with local stresses

58.1 General

You can start this calculation in the `Various` section of the modules-tree.

58.1.1 Functionality of the software

The calculation program supplies a complete, written proof of integrity for static and fatigue strength at the proof point W.

The strength verification is supplied in accordance with the local stress concept as described in the FKM "Rechnerischer Festigkeitsnachweis für Maschinenbauteile" guideline. The idea behind the local stress concept is to estimate the service life on the basis of the elastic-plastic, local stress at the critical point compared to the Woehler line strain derived from an unnotched test beam. The local concept is implemented as a stress-based variant within the framework of the FKM guideline. Therefore, before it can be used, the material must be in an elastic state. In this context, the concept used is not really a local concept like the elastic-plastic notch root strain concept, but a concept close to the nominal stress concept, except that the diameter quotient stands on the other side of the equation. It is a useful tool for calculating static and high cycle fatigue proof in the high cycle range ($N > 10^4$).

Input: You can enter stress amplitudes and stress ratio at a proof point W and at a neighboring point B. Alternatively, the stress ratio at the proof point and the support coefficient are estimated mathematically. Alternatively, the stress ratio at the proof point and the support effect are estimated mathematically. You will also need a number of parameters, such as surface roughness, part size etc. to calculate the design coefficients. Additional load data, such as number of cycles, spectrum, temperature etc. are also predefined.

Output: The calculation calculates the utilization factors for static cases and fatigue. It creates a complete set of documents for this.

58.1.2 Areas of application for the FKM guideline

The software is based on FKM Guideline 183 "Rechnerischer Festigkeitsnachweis für Maschinenbauteile", chapters 3 and 4. The guideline applies to mechanical engineering and its associated industrial sectors. In real life scenarios, contractual partners must agree how this guideline is to be implemented. For parts that are subject to mechanical stress, this guideline can be used to calculate the static and fatigue strength either be for a finite or infinite working life. However, this guideline does not cover other mathematical proofs such as brittle fracture stability, stability or deformation under load, or experimental strength verification. Before the guideline can be applied, it is assumed that the parts have been manufactured so that all aspects of their design, material and operation are technically free of error

and fit for purpose. The guideline is applicable for parts made of iron and aluminum alloys, even at elevated temperatures, either by machining or welding, and in particular for

- parts with geometric notches
- parts with welded joints
- static stress
- fatigue loads ranging from approximately ($N > 10^4$) cycles as an individual or collective load
- rolled and forged steel, including stainless, mix cast iron alloys as well as forged and cast aluminum alloys
- different temperatures
- a non-corrosive ambient media

Supplementary agreements must be drawn up if this guideline is to be used outside the specified area of application. The guideline does not apply if a strength verification is required using other standards, codes or guidelines, or if specific calculation data, such as VDI 2230 for bolted joints, is applicable.

For simple rod-shaped and planiform elements, we recommend you use a calculation method that involves nominal stresses. The calculation using local stresses is to be used for volumetric parts or, in general, where stress is to be calculated using the finite element method or the boundary element method, if no specifically defined cross sections or simple cross section forms are present or if the diameter quotients or notch effect values are unknown.

58.2 Background

58.2.1 The FKM Guideline: Rechnerischer Festigkeitsnachweis für Maschinenbauteile

The idea for this guideline was proposed at a meeting of the DVM in Berlin, Germany, in May 1990, when experts from the then Federal Republic of Germany met together with experts from the then German Democratic Republic. The objective was to combine the standards from what was then two separate standards (VDI in the Federal Republic of Germany and TGL in the German Democratic Republic) to create a single new strength assessment guideline. The new guideline was to be based, in particular, on the former TGL standards for strength calculation, VDI guideline 2226, DIN 18800, Eurocode 3 and the recommendations of the IIW. It was also to take into account the latest discoveries from research into the fatigue strength of metallic parts. The FKM guideline is designed for use in mechanical engineering and associated industrial sectors. The first edition of the FKM guideline, "Rechnerischer Festigkeitsnachweis für Maschinenbauteile" appeared in 1994, followed in 1998 by a third, completely reworked and extended edition (characterized by its much more practical updates and a more user-friendly structure). A fourth edition, which was even more comprehensive, was published in 2002. The main innovation of this edition was the inclusion of aluminum materials. An English translation of this guideline appeared as the fifth edition. The sixth edition of the guideline (2012) has, once again, been completely reworked and now includes the results of new research, such as the data supplied by the tests in "Margin of safety static resistance" and "Improved FKM Guideline". In the meantime, the FKM guideline has been widely accepted and is regarded as the best reflection of the current state of technology.

58.2.2 Usefulness of the service life calculation

It is a well known and proven fact that the service life calculation is not sufficiently accurate. In other words, factors in the range from 0.1 to 10, and in some cases even greater, may occur between the calculation and the test, relating to the tolerable number of alternating cycles. However, a basic, if somewhat simplified, statement about the difficulties in achieving a reliable service life calculation has been made: In this case the strength verification is based on a comparison of the stress values and the stress itself. In a static strength verification, the occurring force can be compared with the sustainable force. For a proof of service strength, the characteristic functions, i.e. the stress spectrum and the Woehler line, are compared. If the total damage, which is of central significance to the service life calculation, is then understood as a quotient of the characteristic functions for stress and sustainable stress, it is clear that this quotient is very sensitive to changes

in these critical values. This means errors in determining the characteristic functions will have a significant effect on the result. In addition, by influencing the critical values, for example, by implementing specific measures when selecting materials and at the production stage, the long-term sustainable service life can be increased.

Three different concepts can be used to calculate the service life of components that are subjected to cyclical stress. These are: the nominal stress concept, the local concept and the fracture mechanics concept. These concepts have specific application areas. For many decades, the technical set of rules was based solely on the nominal stress concept. However, nowadays the local concept and the fracture mechanics concept are being used more and more frequently in this set of rules. Whereas, in the nominal stress concept, the complex transfer function between stress and service life contained in the total stress-strain event in critical material volumes (notch bottom area) is given directly with the part Woehler line for nominal stresses, in the local concept this must be represented mathematically by a number of relatively complex modules. This may be the reason for results according to previous experience not being any more accurate than those achieved with the nominal stress concept.

Possible sources of errors in calculating the local concept:

Load assumptions

It must be emphasized that the load assumption must be as precise as possible to ensure an accurate calculation of component service life. Any errors in load assumption can have significant effects on the service life calculation results. The effect may even be greater than those due to insufficient accuracy of the different methods used for service life estimations. We recommend you check the load assumptions carefully and test them if necessary. In this way, any uncertainties in the load assumptions can be resolved by actual measurements performed at a later date. This is particularly because this type of measurement can be performed non-destructively and can usually provide important information for subsequent designs.

Local stress

Local stresses can be determined either mathematically or by measurement. It is essential that the part's geometry is entered exactly, in particular the splines and wall thicknesses. A convergence check must also be performed to ensure the effective stresses are not underestimated. However, a problem in productive operation still to be resolved is how to calculate the effective level of internal stresses in a part cross section or in a surface layer so that this can be evaluated when subjected to load stresses in a service life calculation.

Combined stress

In the case of combined stress, a strength calculation should fulfill the instance of the invariant (results independent of the selected coordinate system). However, as Woehler lines (with different inclinations) are used for normal and shearing stresses, the resulting calculated service life/damage is no longer separate from, and independent of, the selected coordinate system.

Material properties

Since it is usually not possible to ascertain material properties by simply measuring the finished part, we recommend you use standardized or, at least, well-documented values. It is acknowledged that these values may be dispersed and not always suitable. It is also not possible to determine reliable endurance limit values from tensile strength Rm alone. The fatigue limit can be estimated using the proof stress Rp02. The FKM guideline defines the values from Rm and also for the material type.

Cyclical deformation characteristic

A check to see whether cyclical compaction or loss of cohesion is present must be performed to see whether or not the sequence of load cycles plays a significant role. This effect is not considered in the calculation program.

Support effect

A number of different models can be used to determine the support effect. As many comparisons between calculated results and test results have shown, a mathematical estimate of the support effect is fraught with uncertainties.

Production processes

When a local concept is applied, it is assumed that the volume element displays cyclical material behavior. Influences encountered during the production process, in particular surface layer properties, surface roughness, material state, and internal stresses, must be taken into consideration. Currently used calculation methods also have their limitations here.

Damage parameters

A number of damage parameters have been proposed to help determine the influence of mean stress and the influence of multiple shafts. PSWT, the most well-known damage parameter, corresponds to a mean stress sensitivity of M=0.41, which is present in this order of magnitude for heat treatable steel, but assumes entirely different values for low strength steels or wrought aluminum alloys. The use of PSWT should be seen as a major source of errors. Also in question is the extent to which the influence of internal stresses can be determined. In the latter case, this is only known for exceptional cases in practice. Damage parameters are

still widely used by researchers to determine multi-shaft behavior, excluding proportional stress. The influence of multi-shaft stress states on service life depends greatly on which material is being used. This is because the material's resilience determines which different damage mechanisms are present.

Damage accumulation

In practice, damage accumulation occurs almost exclusively in accordance with the Palmgren-Miner linear hypothesis. Although the shortcomings of this hypothesis were recognized early on, no significant advances that would lead to tolerable errors in the service life calculation have been made in this area despite decades of intense international research. The only progress is that, by summarizing the amplitudes below the endurance limit, various modifications have been proposed which achieve much better results than the original Palmgren-Miner rule, and in which no damage is caused to amplitudes below the endurance limit.

Even if the service life calculation methods for evaluating variants and analyzing weak points are implemented correctly, it is not certain that the current level of knowledge can achieve a reliable service life calculation for new parts. This requires the use of strategies in which calculations are validated and calibrated by specific experimental analyses. At the current level of knowledge it is only possible to make relative forecasts about service life on a purely mathematical basis.

58.3 Implementation in KISSsoft

58.3.1 Main screen

58.3.1.1 Selection of the part form

Selection of the part form: you can choose between parts that are rod-shaped (1D), shell-shaped (2D) or block-shaped (3D). They each have different stress components or stress types, and different indexing. If the local concept is applied, block-shaped (3D) parts are usually present. The selected part form influences the data input for the stress components.

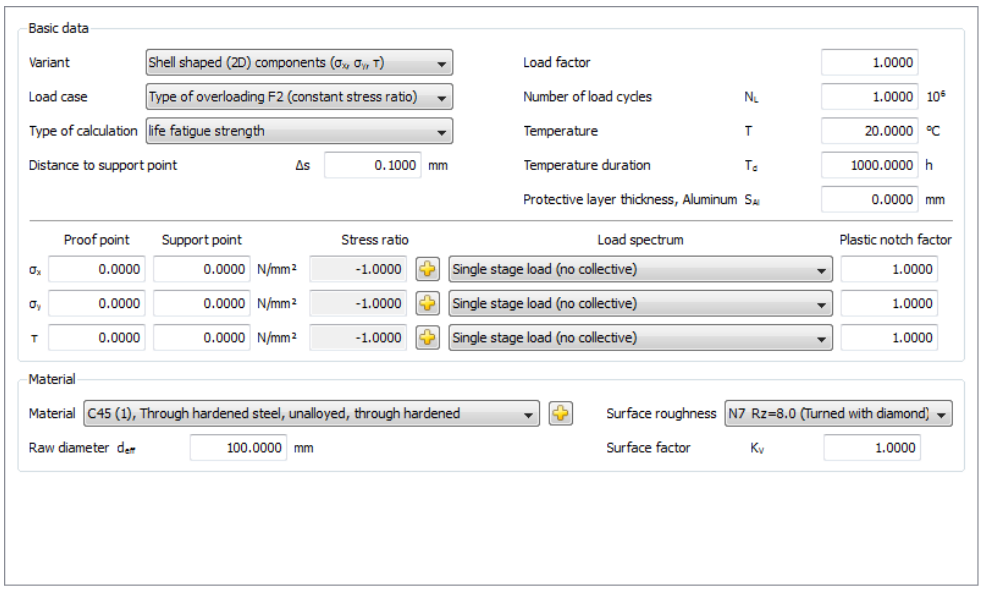


Figure 57.1: Main screen for the proof with local stresses

Rod-shaped parts: for rod-shaped parts - rod, beam, shaft - the following part-related coordinates system applies: The X-axis lies in the rod axis, and the Y- and Z-axes are the main axes of the cross section, and need to be specified in such a way that Iy > Iz applies for the moment of inertia.

For planiform (flat) parts - disc, plate, shell, - the following part-related coordinates system should apply in the proof point: the X- and Y-axes lie in the plane, and the Z-axis is vertical to it in the direction of thickness. The normal stress and the shear stresses in the direction of Z should be negligible.

Block-shaped parts: volume-related coordinates systems apply. The primary stresses S1, S2 and S3 need to be calculated. In the proof point W on the free surface of a 3D part, the primary stresses S1 and S2 should act in the direction of the surface and the primary stress S3 points into the interior of the part, vertically to them. Generally, there is one stress gradient that runs vertically to the surface, and two stress gradients in the direction of the surface, for all stresses. However, only the stress gradients for S1 and S2, running vertically to the surface, can be taken into account in the calculation, and not the stress gradients for S1 and S2 in both directions on the interface and none of the stress gradients for S3.

58.3.1.2 Inputting the stress values on the proof point and on the neighboring point

If the support factor is determined according to the stress state on the neighboring point, then the stresses on the proof point W and on the support point B, and also the distance from point B to point W, will be entered. (Enter compressive stresses as negative values):

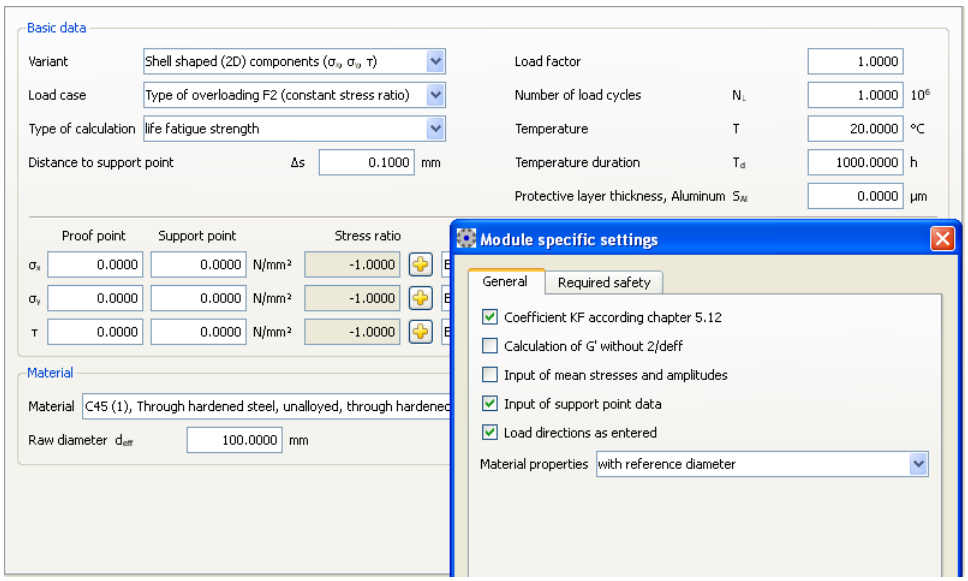


Figure 57.2: Inputting the stress values on the proof point and on the neighboring point. Inputting the neighboring point distance.

58.3.2 Load cases

In the endurance limit diagram, different assumptions are used to determine different levels for the maximum stress amplitude S_{AK}. Assumptions where sm=const. result in a larger S_{AK} than for R=const. This is because the limit lines in the Smith diagram rise by an angle < 45° (mean stress sensitivity). The most suitable assumption depends on the expected change in stresses in the part when it is subjected to permitted operational fatigue load. The overload case can therefore be a decisive factor in whether or not a part is overloaded [62].

Load case

- Type of overloading F1 (constant mean stress): at a constant mean stress the stress amplitude increases as the decisive operating force increases
- Type of overloading F2 (constant stress ratio): When the operating force increases, the ratio between the maximum stress and minimum stress remains the same. This overload case usually returns conservative results (compared to other overload cases) and should therefore be used in cases of doubt.
- Type of overloading F3 (constant minimum stress): when the operating force increases, the minimum load remains the same.
- Type of overloading F4 (constant maximum stress): when the operating force increases, the maximum load remains the same.

58.3.3 Woehler line

Miner elementary, Section 4.4.3.5.2 of the FKM guideline

If a stress collective is present instead of individual stress, the calculation should usually be performed using the Miner elementary procedure.

Miner consistent, Section 4.4.3.5.2 of the FKM guideline

The Miner consistent procedure (derived from Haibach, see [94]) takes into consideration the fact that the part endurance limit will reduce as damage increases. The reduction applies from $K_{D,}\sigma$=1*10e6.

58.3.4 Number of load cycles

Number of load cycles. If calculation in accordance with elementary Miner Rule is selected, then inputs greater than ND result in constant use.

58.3.5 Temperature

Inputting the temperature in degrees Celsius. The area of application of the FKM Guideline is limited according to material, see section 1.2.1.7. The temperature factor $K_{T,D}$ is defined on the basis of the temperature and the material type.

58.3.6 Temperature duration

Time period during which the part is subjected to the temperature.

58.3.7 Protective layer thickness, aluminum, chapter 4.3.4, Figure 4.3.4

Protective layer factor K_S (which is defined via the protective layer thickness) takes into account the influence of a protective layer on the fatigue strength of a part made of aluminum.

58.3.8 Stress ratios

The mean stress is recorded in the R-value. In comparison to the mean stress-free case (cyclic loading, R=-1), the Woehler line is moved to higher sustainable stress amplitudes in the case of trials with mean compression stresses, and in the case of trials with mean tensile stresses the Woehler line is moved to lower sustainable stress amplitudes. The sustainable stress amplitude's dependency on the mean stress is material-specific, and is called the influence of the mean stress. This usually increases along with the tensile strength of the material.

Here R is defined from -1 up to +1

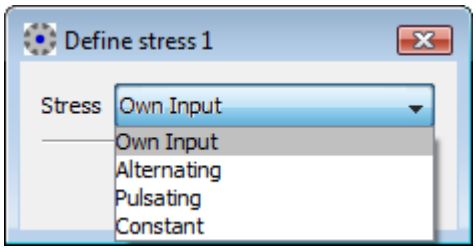


Figure 57.3: Inputting the specific R-value

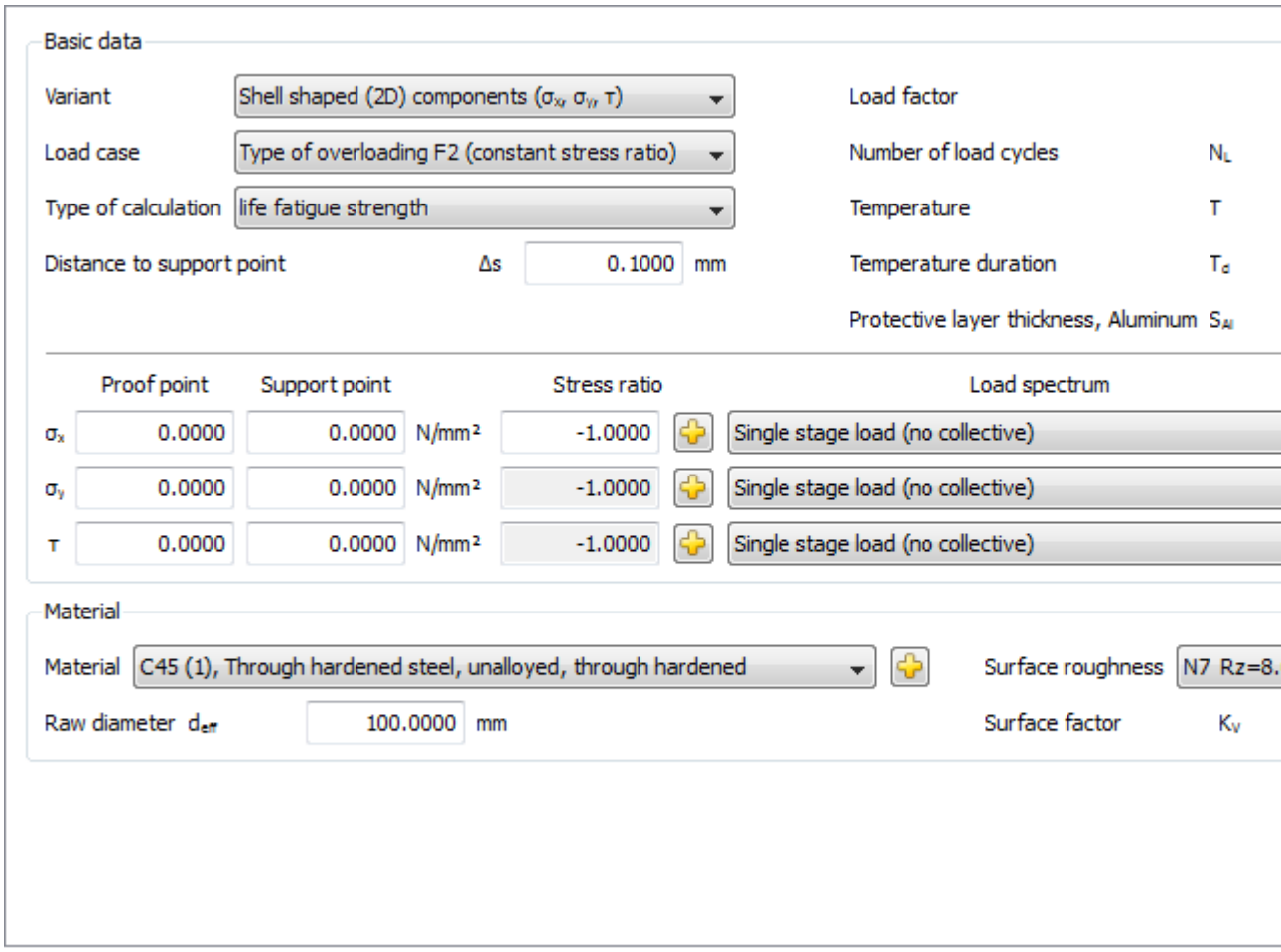


Figure 57.4: Inputting your own R-value.

As the surface roughness increases, the Woehler line moves to lower stress amplitudes, but the surface roughness alone is not the cause for this. The strength is much more affected by the detailed properties of the surface. In addition, despite similar surface properties and the same surface roughness, different processing procedures can cause different material internal stress states, resulting in Woehler lines differing from each other greatly.

58.3.9 Spectra

You can select existing load spectra directly.

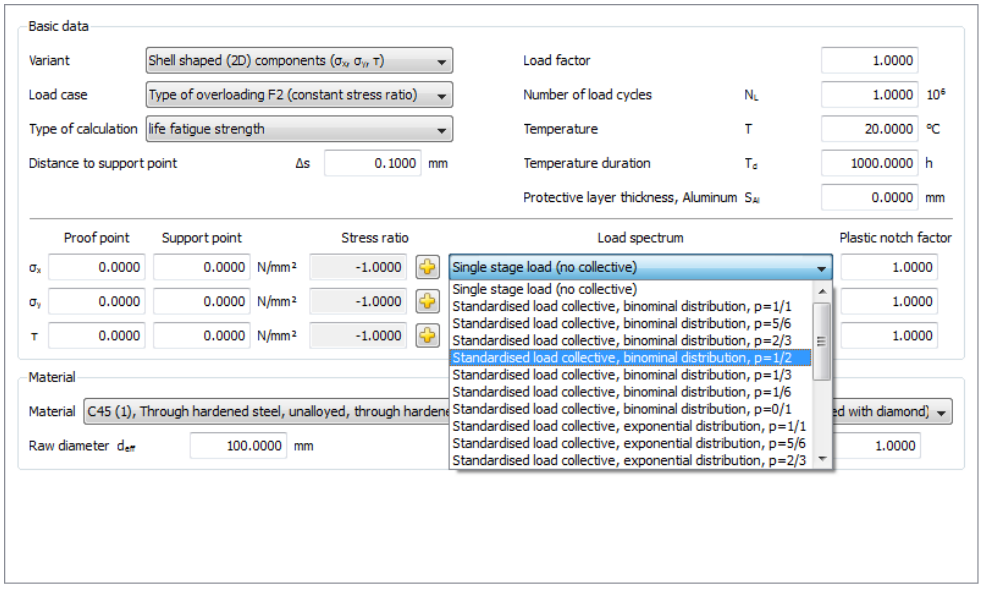


Figure 57.5: Selecting spectra

You can create a new load spectrum in the database tool (see section "Define load spectrum" on page II-287).

58.3.10 Surface factor KV, section 4.3.3, Table 4.3.7

Case factor K_V takes into account the influence of edge layer strengthening on the fatigue strength.

58.4 Materials

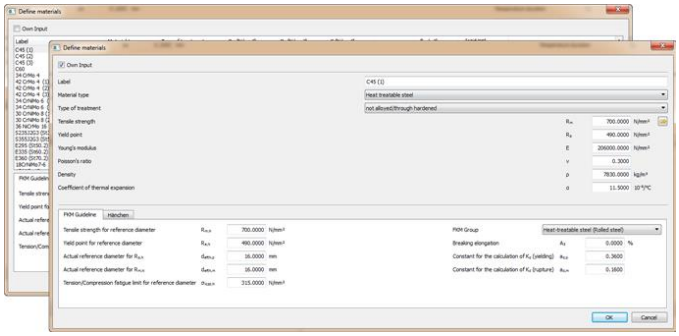


Figure 57.6: Materials screen: strength verification using local stresses

The selection list contains materials from the FKM Guideline.
If you have set the `Own Input` flag, a new dialog appears here. This displays the material data used in the calculation which you can specify to suit your own purposes. You can also define your own materials directly in the database (see page I-114) so that these can also be used in future calculations.

58.4.1 Surface roughness

The roughness factor takes into account the influence of the surface roughness on the part's fatigue strength. Experiments are performed to derive it from the endurance limits of unnotched test rods with and without surface roughness, and shown in dependency of the material's total height Rz and tensile strength Rm. For polished surfaces it has the value 1.0. For rolled, forged and gray cast scale, the average roughness Rz=200µm applies. The average roughness can also be defined as your Own input.

58.4.2 Settings

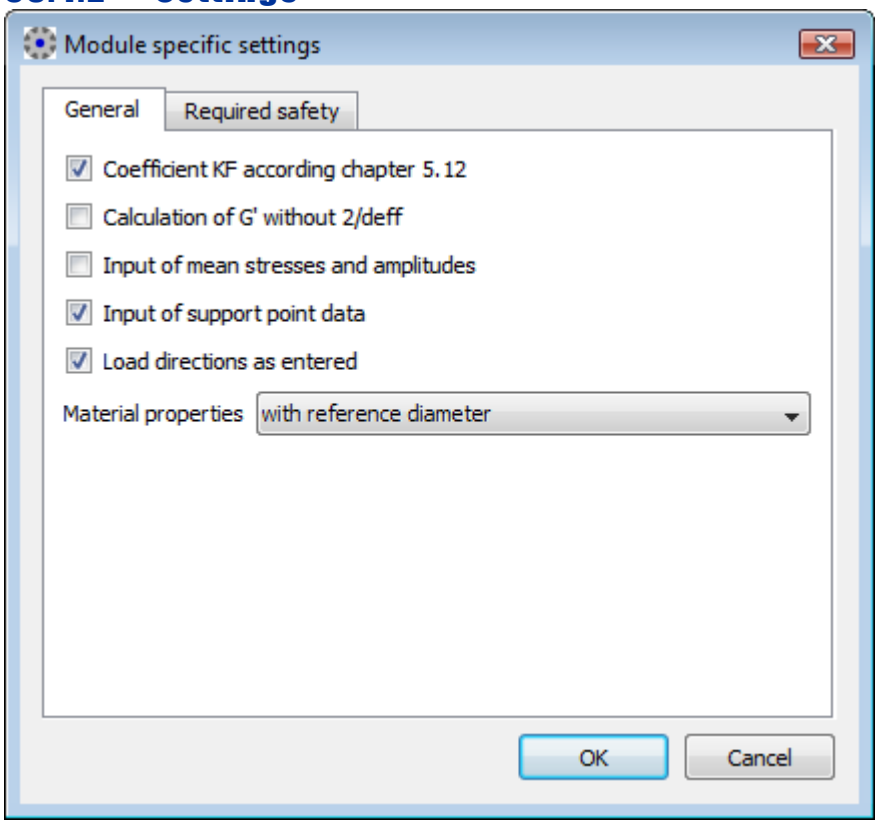


Figure 57.7: Settings

58.4.2.1 General settings

The references are to sections in the FKM guideline.

Coefficient KF according to equation 4.3.2 and 4.3.3, section 4.3.1.2

Notch effect coefficient as an estimated value to enable the effect of the roughness coefficient to be determined, according to the nominal stress concept, when the local stress concept is in use.

Flag set: Kf is defined according to formulae 4.3.2 and 4.3.3, described in section 4.3.1.2.

Flag not set: The KF coefficient is set as shown in Table 4.3.1.

Calculating G without 2/deff, section 4.3.1.3.3

If the flag is not set in General data, Neighboring point data entry, then an approximation of the related stress gradient is calculated, using the calculation based on the equations in 4.3.18. This contains terms for tension/compression, torsion, and bending. If no bending is present, it is questionable whether the second term (2/d) in the formulae makes any sense. The option programmed here is not provided in the FKM Guideline!

Flag set: the stress gradient is defined without applying the second term in formula 4.3.18.

Flag not set: the stress gradient is defined while also applying the second term in formula 4.3.18.

Input of mean stresses and amplitudes

If the flag is set, then the stresses are input in the main screen via the medium and amplitude stress.

Input of mean stresses and amplitudes

- **Material values at reference diameter:**
  values are taken from database (at reference diameter) and multiplied by K1
- **Rm, Rp depending on value from database, sigW at reference diameter:**
  Rm, Rp are read from the database according to size (excluding K1), and the fatigue strength is determined for the reference diameter entered in the database, and then it is multiplied by K1
- **Rm, Rp depending on value from database, sigW constant:**
  Fatigue strength not multiplied by K1, correct value must be in database
- **Rm, Rp depending on value from database, sigW calculated from Rm:**
  Fatigue strength is calculated from Rm. Rm is in database according to size, conversion according to FKM

Neighboring point data entry, section 4.3.1.3.3, Formula 4.3.17

Flag set: Support coefficient-related stress slope is defined in the neighboring point via the stress state. To do this, the stress values and the distance between the proof point and neighboring point must be entered in the main screen.

Flag not set: Support value-related stress gradient is not determined from the values at a neighboring point. The related stress gradient at the point of maximum stress is estimated using formula 4.3.18. To do this, two radii (Radius 1 and Radius 2) must

be defined (for the two directions on the surface), and also a typical part dimension d. See also: module specific settings, Calculation of G without 2/deff, above.

Direction of load as specified, sections 4.1.0, 4.6.2

Flag set: the calculation is carried out for synchronous stresses.

Flag not set: the calculation is performed for asynchronous stresses (4.6.2.2). It can safely be assumed that this method of approach is a cautious one.

Use the mechanical material support coefficient, sections 4.3.1, 4.3.1.3.2

If this flag is selected, the mechanical material support coefficient is used for the calculation, otherwise the Stieler support coefficient is used. If sharp notches are present, the mechanical material support coefficient takes into account the strength reserves and contains the static size coefficient. The mechanical material support number (nwm) is made up of three parts: the static support number (nst), the mechanical deformation support number (nvm) and fracture mechanical support number (nbm). Assumption: nst = 1 is applied to the "Smooth shaft" and "Own Input" notch types.

Selecting materials data, section 3.2.1

The part standard values Rm and Rp must be calculated from the semi-finished product or test piece standard values $R_{m,N}$ and $R_{p,N}$ or from the part drawing value $R_{m,Z}$. In exceptional situations, the part actual values $R_{m,I}$ and $R_{p,I}$ can be applied. Refer to section 3.2.1.2.

58.4.2.2 Required safeties

The FKM Guideline is one of the few calculation guidelines that lists the required safeties according to the consequences of failure etc. In combination with safe load assumptions and an average probability of survival of the strength variables Pü=97.5%, they apply for both welded and non welded parts. Safety factor are defined on the basis of the selected material and the defined consequences of failure, probability of occurrence of the load, and also inspection and test. It differentiates between steel, cast iron (ductile or non ductile), and also aluminum (ductile or non ductile), i.e. five different classes. Alternatively you can also set the safety factors manually.

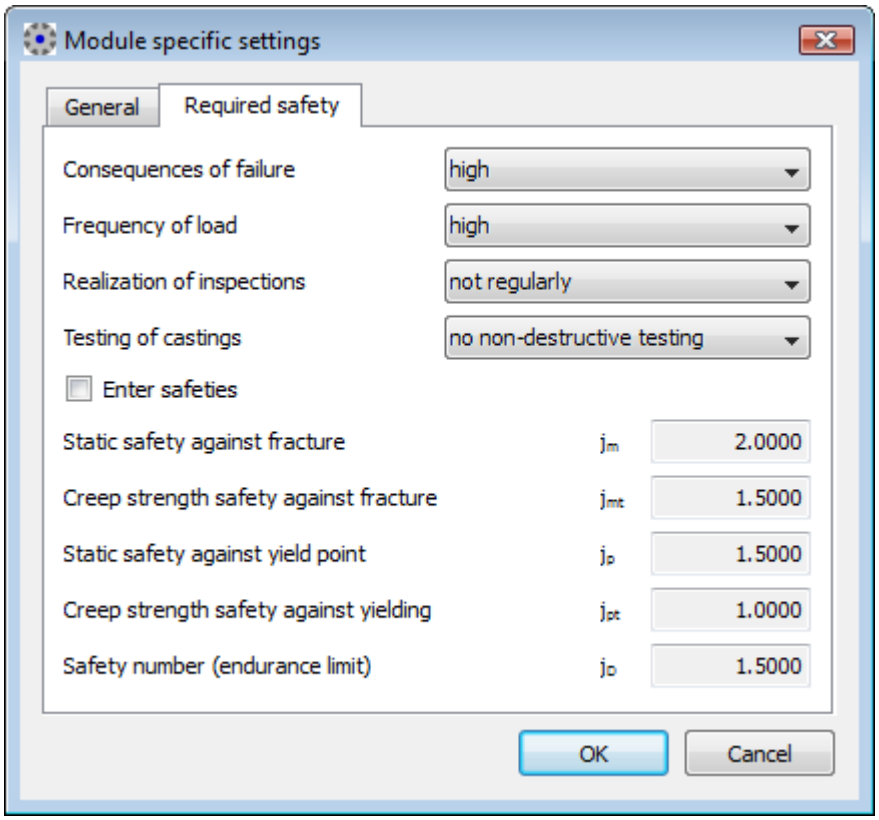


Figure 57.8: Selecting the safeties according to material and load properties

jmt	Safety margin against creep strength depending on time
jp	Safety margin against yield point
jpt	Safety margin against creep limit
jF*j	Safety against the endurance limit

G

Chapter 58
Hertzian pressure

In this module, the Hertzian pressure of two bodies is calculated. In the case of a load on a rolling pair that is applied vertically to the contact surface, elliptical flattening occurs for point contact, and rectangular flattening occurs in the case of linear contact. The Hertzian equations are used to help calculate the maximum pressure (Hertzian pressure) and also the proximity of the two bodies (ball, cylinder, ellipsoid, plane; convex or concave). The calculation formulae have been taken from "Advanced Mechanics of Materials, 6th Edition [78]. The underlying principle for calculation for point contact is that the diameter of the bodies is defined on two principal planes, from which an equivalent diameter is then defined. In the case of linear contact, the calculation is performed in one main plane, so there is only one equivalent diameter. In addition the location and value of the maximum primary shearing stress in the interior of the body are determined.

An approximation of the cylinder/cylinder configuration has been calculated using the dissertation from Weber/Banaschek [69]. The formula (55) from Norden's book [89] is used to calculate the approximation of the cylinder area.

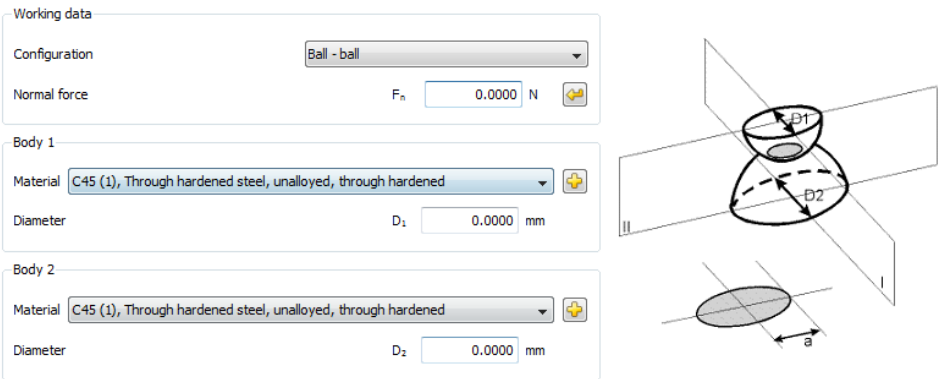


Figure 58.1: Main window for Hertzian pressure

The main window for Hertzian pressure (see Figure 58.1) is where you define the normal force, the configuration, and the diameter (in addition to the supporting

length for linear contact), and the materials used in the body.
You can select one of these configurations:

- Ball - ball
- Ball - cylinder
- Ball - ellipsoid
- Ball - plane
- Ellipsoid - ellipsoid
- Ellipsoid - cylinder
- Ellipsoid - plane
- Cylinder - cylinder
- Cylinder - plane

On the right, in the main screen, an image of the current configuration is displayed to help you input the values more easily.
For normal force, there is also a sizing option. If you click the sizing buttons next to the normal force, you can enter the required Hertzian pressure, and the system will then calculate the normal force from it.
If the bearing area has a concave bend then you must enter the diameter as a negative value. Negative diameters are only possible in the case of Body 2.

59.1 Settings

Use the `Depth display factor` to define the depth display in the graphic. The depth of the point τmax is multiplied by this coefficient. The resulting depth is then displayed in the graphic. The default setting of this coefficient is 6.

Chapter 59
Hardness conversion

You access the Hardness conversion module in the `Extras > Hardness conversion` menu. In addition, the hardness conversion is contained in the materials screens as a sizing function, where, for example, the tensile strength can be defined by means of a hardness value.

This module contains the hardness conversion calculation as specified in DIN EN ISO 18265, 02/2004 Edition. The conversion applies to non-alloyed and low-alloy steels and steel castings. According to each case, the stored tables can be used to convert the value of the tensile strength into Vickers, Brinell, or Rockwell hardness, and vice versa. Due to possible variations, the received values should only be used if the default testing process cannot be applied. The interim values of the value conversion table will be interpolated from the neighboring values.

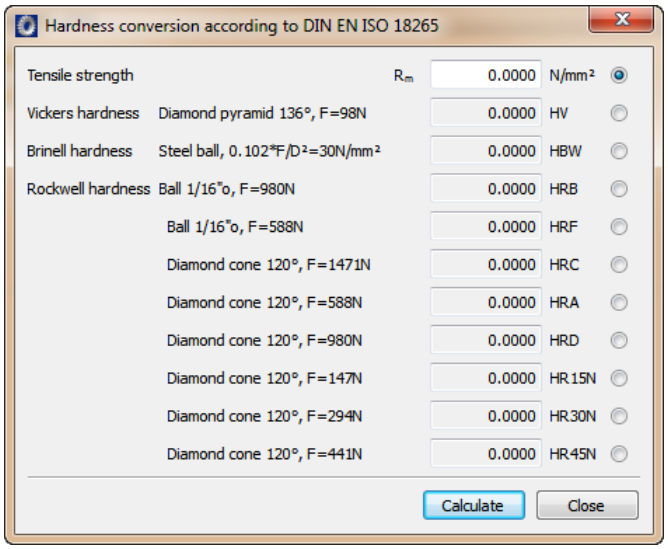


Figure 59.1: Hardness conversion input screen

The validity area of the different processes will be restricted as follows:

- Tensile strength Rm: 255 to 2180 N/mm2
- Vickers hardness HV: 80 to 940 HV
- Brinell hardness HB: 76 to 618 HB

- Rockwell hardness HRB: 41 to 105 HRB
- Rockwell hardness HRF: 82.6 to 115.1 HRF
- Rockwell hardness HRC: 20.3 to 68 HRC
- Rockwell hardness HRA: 60.7 to 85.6 HRA
- Rockwell hardness HRD: 40.3 to 76.9 HRD
- Rockwell hardness HR 15N: 69.6 to 93.2 HR 15N
- Rockwell hardness HR 30N: 41.7 to 84.4 HR 30N

Rockwell hardness HR 45N: 19.9 to 75.4 HR 45N

Kapitel 60
Linear drive train

Use this calculation module to calculate drive screws. Drive screws are used to convert rotational movement into longitudinal movement or to generate great forces.

Although trapezoidal screws are almost exclusively used as drive screws, some rough operations also use buttress threads.

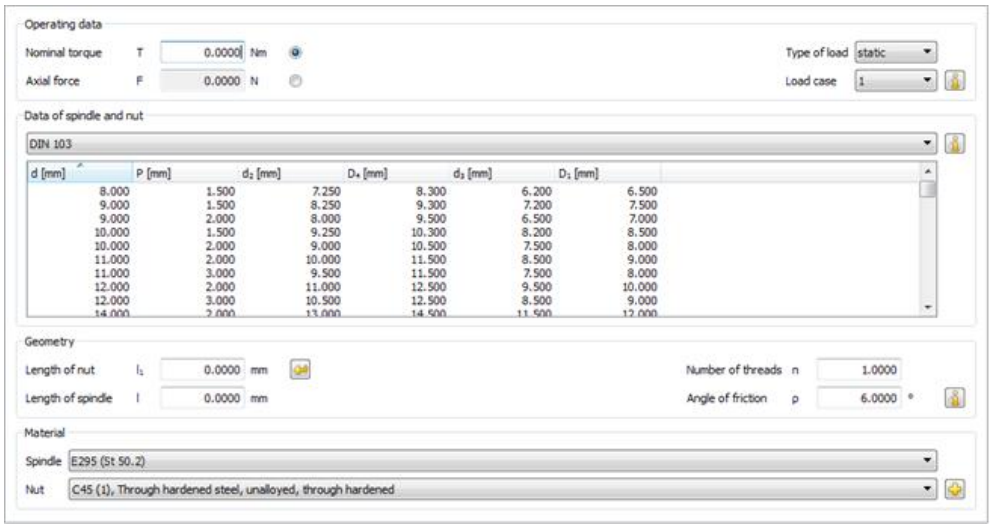


Figure 60.1: Basic data: Linear drive train

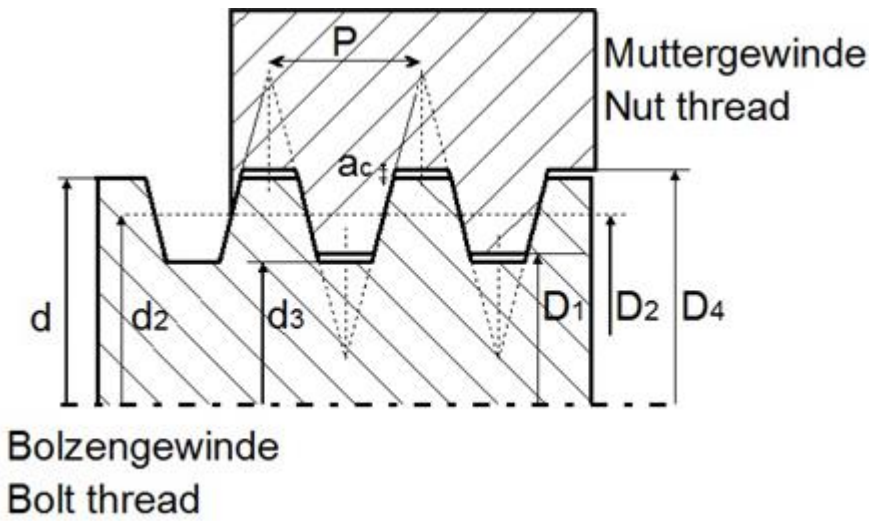


Figure 60.2: Dimensions of trapezoidal screws

There are two different configurations of linear drive train that can be calculated:

- `Load case 1`
  Stress on the spindle in a spindle press

Load case 2

Stress on the spindle in a gate valve

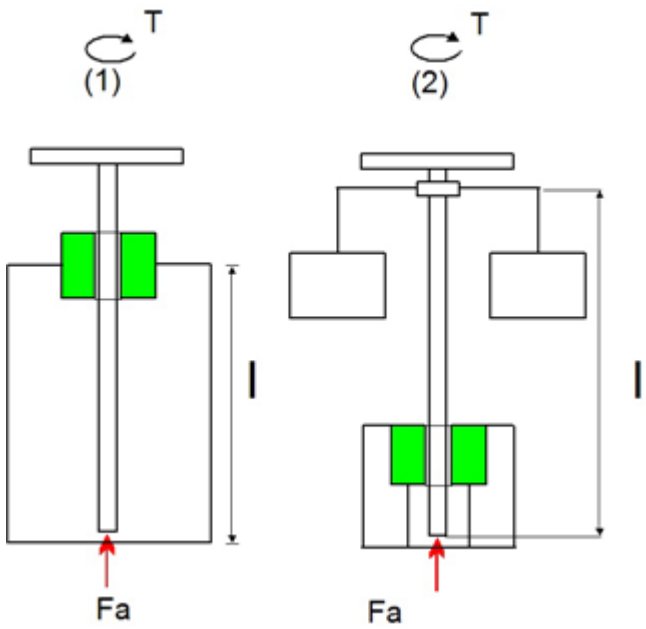


Figure 60.2: Load cases Linear drive train

The information provided in Roloff Matek [62] is used to calculate linear drive trains (drive screws).

61.1 Calculation

Short and long linear drive trains subjected to pressure are handled separately in the calculation process.

Short pressure stressed drive screws

Short pressure stressed drive screws are not at risk of buckling and therefore are not tested for this.

The required cross section of the thread can therefore be defined using the formula:

$$A_3 \geq \frac{F}{\sigma_{d(z)zul}}$$

σd(z)zul: under static load: Rp/1.5; under pulsating load σ zdSch/2.0; under alternating load: σ zdW/2.0;

Long pressure stressed drive screws

The formula for calculating the necessary core diameter of the thread is taken from the Euler equation:

$$d_3 = \sqrt[4]{\frac{64 \cdot F \cdot S \cdot lk^2}{\pi^3 \cdot E}}$$

S: Safety (S≈6 to 8)

lk: mathematical buckling length, lk ≈ 0.7*l (Euler buckling case 3 used for general, guided spindles)

Calculation of the strength:

Load case 1:

The upper part of this configuration is subject to torsion and the lower part is subject to compression and therefore buckling.

Torsional stress:

$$\tau_t = \frac{T}{Wp} \leq \tau_{tzul}$$

Wp: polar moment of resistance Wp≈0.2*d3^3

ττzul: permissible torsional stress; static load ττF/1.5; pulsating load ττsch/2.0; alternating load ττW/2.0;

Compressive (tensile) stress:

$$\sigma_{d(z)} = \frac{F}{A_3} \leq \sigma_{d(z)zul}$$

A3: Thread minor diameter cross section

σd(z)zul: permissible compressive (tensile) stress:

Load case 2:

The upper part of this configuration is subject to torsion and the lower part is subject to compression, infrequent tension and torque.

Formula for the part to be checked:

$$\sigma_v = \sqrt{\sigma_{d(z)}^{\ 2} + 3 \cdot \left(1 \cdot \tau_t\right)^2} \leq \sigma_{d(z)zul}$$

The required torque corresponds to the thread moment, if not subject to any moments of friction.

$$T = F \cdot d_2 / 2 \cdot \tan(\varphi \pm \rho')$$

d2: Flank diameter of the thread

φ Lead angle of the thread (for single thread trapezoidal screws φ≈3° to 5.5°)

ρ': Thread friction angle

Werkstoff Material		Schmierung lubricant	Reibwinkel Friction angle
Spindel/screw	Mutter/nut		ρ' [°]
Stahl/steel	Gusseisen Cast iron	Trocken Dry	12
Stahl/steel	CuZn - CuSn Legierungen CuZn - CuSn Alloys	Trocken Dry	10
Stahl/steel	Gusseisen Cast iron	Geschmiert Lubricated	6
Stahl/steel	CuZn - CuSn Legierungen CuZn - CuSn Alloys	Geschmiert Lubricated	6
Stahl/steel	Spezial Kunststoff Special plastic	Trocken Dry	6
Stahl/steel	Spezial Kunststoff Special plastic	Geschmiert Lubricated	2.5

Figure 60.3: Values for the friction angle

The + in the formula stands for "tightening the spindle", and - stands for "loosening the spindle". The KISSsoft procedure calculates both situations and outputs the results in a report.

Calculation for buckling (only for long spindles):

First of all, calculate the slenderness ratio.

$$\lambda = \frac{lk}{i} = \frac{lk}{\sqrt{I / A_3}} = \frac{lk}{\sqrt{\frac{\pi \cdot d_3^{\,4} \cdot 4}{64 \cdot d_3^{\,2} \cdot \pi}}} = \frac{lk \cdot 4}{d_3}$$

λ Slenderness ratio of the spindle

lk: mathematical buckling length

i: Gyration radius

Only 3 different materials can be used for the spindle so that the slenderness ratio can be defined correctly.

Elastic buckling is present if λ>=λ0 = 105 for S235; λ>=89 for E295 and E335.

$$\sigma_K = \frac{E \cdot \pi^2}{\lambda^2}$$

The non-elastic area as defined by Tetmajer and λ<105 for S235.

$$\sigma_K = 310 - 1.14 \cdot \lambda$$

For λ<89 and for E295 and E335:

$$\sigma_K = 335 - 0.62 \cdot \lambda$$

For a non-elastic case, the Johnson parabola equation can also be used for the calculation. (also for other materials)

$$\sigma_K = \sigma_{dS} - (\sigma_{dS} - \sigma_{dP}) \cdot \left(\frac{\lambda}{\lambda 0}\right)^2$$

The safety can then be calculated as follows:

$$S = \frac{\sigma_K}{\sigma_{vorh}} \geq S_{erf}$$

The required safety for elastic buckling is Serf≈3 to 6. For non-elastic buckling it is Serf≈4 to 2.

Buckling no longer needs to be calculated for a slenderness ratio < 20.

Analysis of the nut:

The surface pressure of the nut is calculated from the nut length:

$$p = \frac{F \cdot P}{l_1 \cdot d_2 \cdot \pi \cdot H_1} \leq p_{zul}$$

P: Pitch of thread

l1: Length of the nut thread

d2: Flank diameter of the thread

H1: Flank engagement of the thread

pzul: permissible surface pressure

Due to the uneven distribution of surface pressure, the nut length should be no greater than 2.5*d. During sizing, the length is limited to 2.5* d even if a longer one is input.

Efficiency and self-locking:

The efficiency of the conversion from rotational movement into longitudinal movement:

$$\eta \approx \frac{\tan \varphi}{\tan(\varphi + \rho')}$$

The conversion of movement is only possible for non self-locking threads, because the limiting value in this case is, if$\varphi = \rho'$, the efficiency is 0.5.

If $\varphi > \rho'$ the thread is no longer self-locking.

Each of the permissible values are listed in the Roloff Matek tables.

61.2 Sizings

This calculation module can calculate the core diameter d3 of a long spindle that is subject to pressure, when you select "Own Input".

In addition, it can also define the nut length on the basis of permissible surface pressure and the required safety.

61.3 Settings

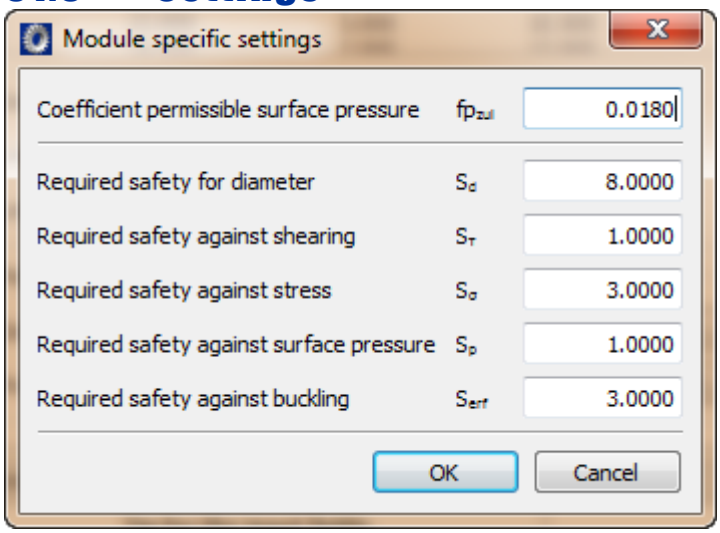


Figure 60.4: Settings input window

- Coefficient of permissible surface pressure; this factor is used to define the ratio to Rm, in other words pzul = fpzul*Rm
- Required safeties for diameter, shearing, stress, surface pressure and buckling: for the calculation and the sizings

61.4 Materials

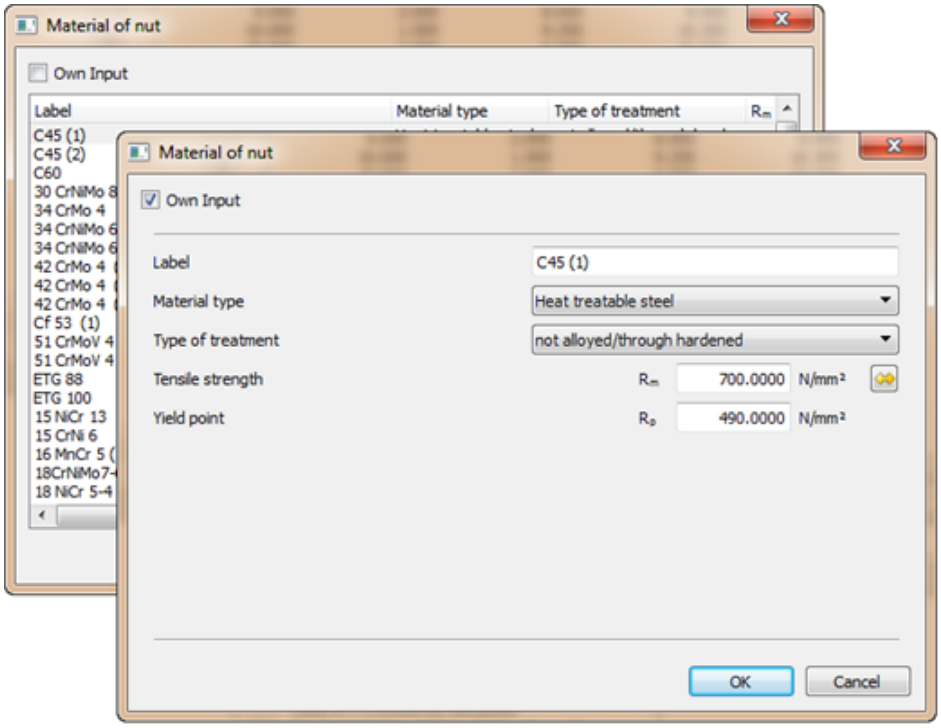


Figure 60.5: Nut materials input window.

In the selection list, you can select materials in accordance with the standard. If you have set the `Own Input` flag, a new dialog appears here. This displays the material data used in the calculation which you can define to suit your own purposes. You can also define your own materials directly in the database (see page I-114) so that these can also be used in future calculations.

You can only select these different materials for nuts. For the spindle material you can choose E295 (St 50.2), E335 (St 60.2) and S235 (37.3) materials, because the calculation of buckling is only designed for use with these materials.

The strength values for the 3 materials have been fixed:

- E295 (St 50.2): Rp02 = 295 N/mm^2; σzdSch = 295 N/mm^2; σzdW = 195 N/mm^2; λ0 = 89; ττSch = 205 N/mm^2; ττW = 145 N/mm^2
- E335 (St 60.2): Rp02 = 335 N/mm^2; σzdSch = 335 N/mm^2; σzdW = 235 N/mm^2; λ0 = 89; ττSch = 230 N/mm^2; ττW = 180 N/mm^2
- S235 (St 37.3): Rp02 = 235 N/mm^2; σzdSch = 225 N/mm^2; σzdW = 140 N/mm^2; λ0 = 105; ττSch = 160 N/mm^2; ττW = 105 N/mm^2

Part IX

KISSsys

Chapter 61
KISSsys: Calculation Systems

62.1 General

KISSsys is an extension to the KISSsoft calculation program. With KISSsoft, you can arrange, optimize, and recalculate, individual shafts, gears, or shaft-hub connections. In contrast, KISSsys is suitable for administering machine element systems.

Some special links between different calculations are already present in KISSsoft. For example, bearing forces can be transferred from the shaft calculation and gears can be placed onto a shaft. However, in the case of larger systems, such as a multi-level gearbox with several shafts and gears, separate performance data and speed must be entered for each individual stage. load has to be updated berechnet werden, must be jeder Calculation the Rating has to be updated.

In contrast with KISSsoft, where the individual calculation takes center stage, KISSsys provides a way to observation a system as a whole. However, KISSsys has not been designed to replace KISSsoft. Instead it is an extension that uses the existing, tried and tested calculation modules. You could say that KISSsys administers the relationships between individual elements but leaves the calculation of the individual elements to KISSsoft.

62.1.1 Structure of KISSsys

KISSsys is based on an object management system called Classcad. Classcad manages the administration of KISSsys elements, evaluates the expressions for variables and provides an interpreter with which the user can also generate functions for special purposes.

This forms the basis for a user interface and a link with KISSsoft. The functionality of the user interface is different for administrators, who generate new systems or change the systems' structure, and for normal users who, while using the same structure, merely want to change data, recalculate, and observe results. It takes more effort, and a better understanding of the program structure, to generate new systems than to use an existing system, which is easy to do.

62.1.2 Ways in which KISSsys can be used

At the most basic level, KISSsys provides a way for grouping calculations. All calculations belonging to a system can be called up from one interface. In addition, you can get an overview of the most important results of all calculations. This makes it immediately obvious which particular gear pair or shaft is critical.

Even just this view of all the calculations that are of interest makes work considerably easier.

KISSsys then offers a way for you to specify relationships between variables. For example, you can calculate the speeds in a gearbox from the initial speeds and the transmission ratios. Moreover, KISSsys can also describe the power flow. Consequently, in KISSsys you only need to enter the load for the calculations in a few places. This enables you to quickly recalculate a complex system for varying load cases.

KISSsys enables you to store tables for loading cases or even variants. In this way, you do not have to constantly reenter the load data. KISSsys can also store the data for variants of a construction. With one click of a button you can then perform all the calculations for a selected load or variant.

For example, imagine a shaft with a radial force of unknown direction (e.g. via a belt drive/ belt force, whose direction is only determined when the equipment is installed). If it is necessary to define the worst case scenario, you could use KISSsys to rotate this force in steps of up to 360°.

KISSsys is not only of great benefit during construction, it is also useful in the sales environment. With KISSsys you can for example store a standard gearbox in your computer. If the client later requests different loads on a gearbox of this kind, instead of the ones originally used for its construction, KISSsys lets you quickly check whether the gearbox will meet the new load requirements.

Different example applications are illustrated on the KISSsoft CD or website.

62.2 The user interface

The user interface provides several views of the administered data. There are table views, which are primarily designed to provide you with a good overview of the calculations. Another view, which has a tree structure, represents the hierarchy of an assembly structure, while the two-dimensional power flow diagram is primarily designed to display the kinematic coupling of the system. In addition, you can produce a three-dimensional display of the entire system or of subsystems.

This section details the options for using the KISSsys system without administrator rights.

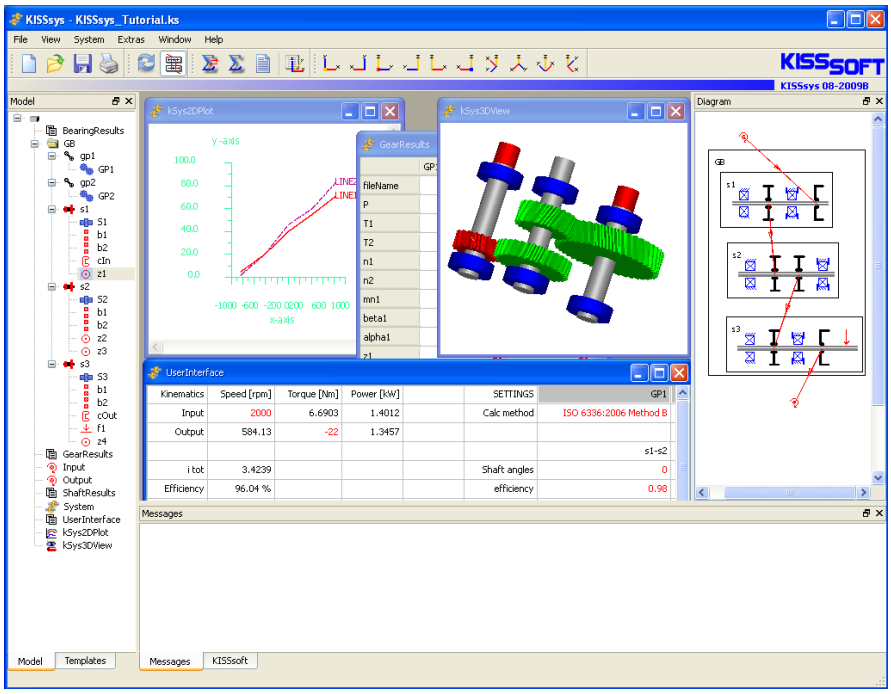


Figure 61.1: The KISSsys user interface with tree view, diagram view, 3D view, tables and 2D diagram

62.2.1 Tree view

The tree view (on the left in → Figure on page IX-1068) lists all elements present in the system, hierarchically. This provides a way to display an assembly structure. Beside the name of the element there is a bitmap that identifies the type of the element. Bitmaps in blue represent KISSsoft calculations, and bitmaps in red represent KISSsys elements. You can identify variables such as numbers, functions

or character strings. With a click with the right-hand mouse button you can open a separate context menu which provides functions for an element.

Each element has a `Properties` dialog which you can display here. The `Properties` dialog contains an overview of the available data elements or variables. However, these can only be changed by the administrator.

In the case of the KISSsoft calculations you can select `kSoftInterface` in the context menu to start the appropriate KISSsoft module. The calculation data can then be changed or evaluated in KISSsoft. Select `kSoftReport` to display the calculation report and select `Calculate` to perform the calculation in the background without a user interface. Data is only exchanged with KISSsoft via the KISSsys calculation elements.

62.2.2 Diagram view

Diagram view (on the right in → Figure on page IX-1068) shows the kinematic coupling of the elements. To start with, the element structure has nothing to do with the calculations. The calculations only use the data that relates to the shafts, gears and connections, and they can be added or deleted as you wish.

The structure consists of shafts and their sub-elements: gears, forces, couplings and bearings. The kinematic coupling and the power flow between the shafts is achieved via `connections`. The connection has the calculation standard to transfer the speed to the next element (usually simply the transmission ratio) and it transfers a torque, also with loss of efficiency.

The externally supplied torque and a speed are defined with speed/torque elements. In each case you can specify whether the speed or the torque are known or whether they should be calculated by KISSsys. The number of predefined values must correspond with the number of degrees of freedom.

The elements in the diagram view can be moved with the left-hand mouse button. Click with the right-hand mouse button to display a context menu like the one in tree view. You can change the zoom factor by pressing the '+' or '-' keys, or in the context menu which you access by right-clicking.

62.2.3 Table view

To display the tables, select `Show` in the context menu in tree or diagram view. The contents of the tables are defined during system set-up. The values displayed in black cannot be changed, but the red numbers or strings can be edited. A special table for user interfaces contains fields with a gray background. These are functions and you can start them by double-clicking on them with the left-hand mouse button.

You can print the contents of the table, or press `Ctrl-C` to copy it and, for example, paste it into a spreadsheet.

62.2.4 3D view

To display the windows for the 3D view, select `Show` in the context menu in tree view. You can rotate the view with the left-hand mouse button, enlarge or reduce it with the right-hand one, and move it with the center mouse button. One of the main views can be selected via the menu or the Tool bars.

In 3D view you can export the 3D geometry into the CAD system (via the context menu). If you want to display graphical elements (see `System.kSys3DElements`), you can export the elements from that file if you have the appropriate license. If a 3D kernel is present and you want to generate solid elements, a CAD file will be generated directly.

62.2.5 Message output

In the lower part of the program window (see → Figure on page IX-1068) there is an output window for messages. Error messages and warnings from KISSsoft calculations will be displayed under `Messages`. Calls by KISSsoft are reported under `KISSsoft`, so this view is usually not required.

In the lower program window, as in KISSsoft, you will see an information tab. If information about a particular function is present, it is displayed in this tab.

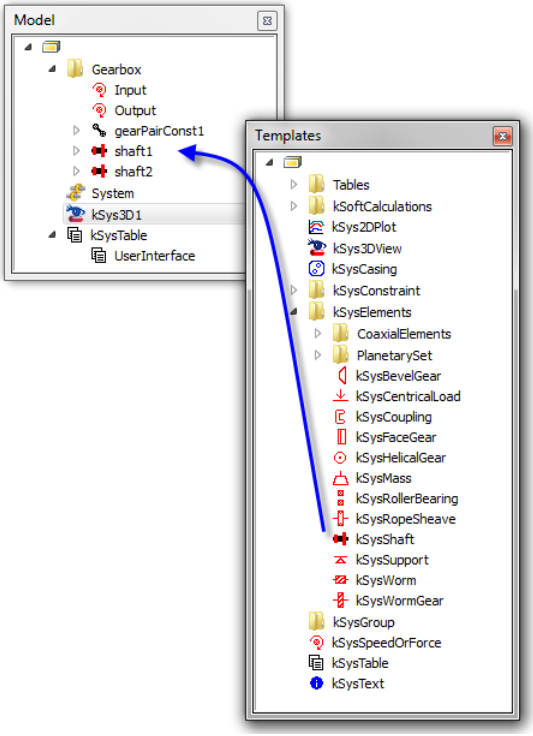
62.3 Creating Models in KISSsys

This chapter is intended for KISSsys users.

There are four ways for you to create new models in KISSsys. They are described in the next four sections.

62.3.1 Classic method

If you enable the Administrator in `KISSsys Extras -> Administrator`, the system displays, in the Template tab, all elements that are required for creating a model. You create a model by copying the particular elements from the template and inserting them in the navigation tree. It is also possible to create the elements by moving them from the template to the navigation tree. All possible models can be created with the classic method.



62.3.2 Element Assistant

The `Insert > Elements` wizard is based on the classic method. If you use this wizard, you no longer have to drag and drop or cut and paste data. To insert an element, click on it, and the system automatically inserts it in the current structure in the navigation tree. Using the Element Assistant you can create all possible kinds of variants.

62.3.3 System Assistant

Under `Insert > System of Shafts` KISSsys provides an Assistant with which you can create cylindrical gear stages. This Assistant leads you through the model step by step when you are creating it. You use it in the same way as the Assistant. Use this function to create a cylindrical gearbox.

Use the KISSsys Assistants to create parallel shafts with the following combinations of gears:

Cylindrical gear

Bevel gear

Worm wheel

Face gear

Use the Planetary gear Assistant to create a single stage planetary gear unit.

This Assistant leads you through the model step by step when you are creating it. You use it in the same way as the Assistant.

62.3.4 Creating and modifying tables

To create your own tables in KISSsys Model you can use a predefined table called "UserInterface", which is stored in the template in "Tables". You use this "UserInterface" table to add all the parameters from the elements and your own texts. You can select the table's name to suit your needs.

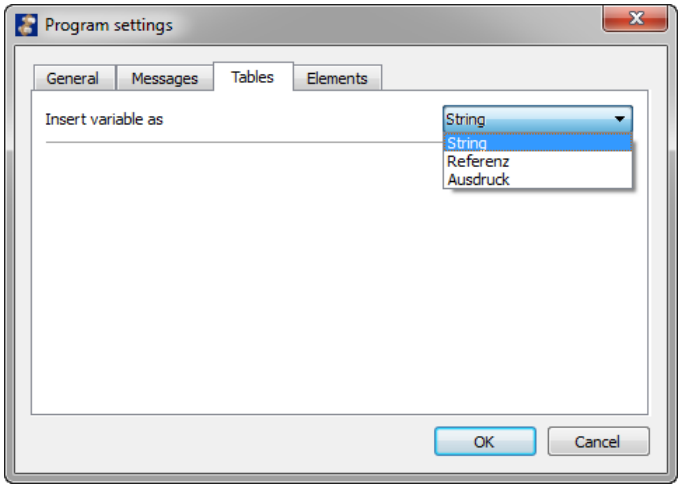
For each cell, no matter what it contains, you can right-click with the mouse to select Format in the context menu. There you can set the font, color, background color, and position, of the text for that cell.

62.3.5 Adding variables in tables

One way of inserting variables in the "UserInterface" table is to select a variable from the "Properties" dialog and then click on the corresponding icon in the "Tool bar" menu. You can then insert the variables either as text (Name), as a reference, or as an expression.

Text

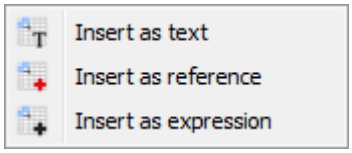
You can also input texts directly in the cell. Alternatively, you can use the Text icon to help you. Select the parameter you want to insert from an element's property and preselect a cell in the table. Then click on the Text icon. This transfers the required value to the cell. You can also hold down the left-hand mouse button and drag and drop the required parameter to insert it directly. The default setting is for text to be inserted. Click on `Extras > Settings > Tables tab` to define the default setting as a text, reference or expression



Reference

Referenced data is displayed in red. These values are referenced with a parameter in the element property. You can modify the value both in the table and in the property.

You can add referenced values to cells in the same way as you add texts. Select the parameter you want to insert from an element's property and preselect a cell in the table. Then click on the Reference icon. This transfers the required value to the cell. Alternatively, you can hold down the right-hand mouse button and insert the parameter in the appropriate cell. A selection window appears in which you can select a text, reference or expression, as required.



Expression

The expression is merely shown as a value and cannot be modified in the table. You can insert an expression into a cell in the same way as a reference.

62.3.6 Individual names for elements

Individual names can be used for all KISSsys elements. The individual name is assigned to the element automatically when it is assigned to the model. This behavior can be set individually for each element (`Extras->Settings->Elements`).

Use the <autoInc> and <localInc> tags to add an index to the individual name at the insertion position. The first of these tags increments the index globally. This means that no other element in the model can have the same name. The second of these tags increments the index locally (in the same folder). Use the <parentName> tag to add the name of the hierarchically superior item.

The "Automatically/Ask" option is set to suppress or display the dialog that prompts the user to define the name for the new element for the model. Click on "Reset" to select the KISSsys default name. You can use the "All questions", "All automatic" and "Reset all" functions to modify all the elements at the click of a button.

62.3.7 Menus, context menus and the Tool Bar

In the File menu you can open, store and save models, open and close projects, and close KISSsys. You can also open or close KISSsys templates.

Each individual docking window in the user interface can be hidden or displayed in the View main menu. You can also refresh all views.

In the System main menu you can generate the KISSsoft report and perform kinematics and KISSsoft calculations. You can also select a load spectrum for the model from the KISSsoft database or define your own in KISSsys. Use these functions to call the characteristics of the element and variable overviews.

In the Insert menu, the Assistant will guide you through the steps required to create a simple gear unit or a single stage planetary gear unit. The Elements Box operates like the Assistant in that it lets you add elements, but without any structural limitations. Default templates are used to add pre-defined models or templates to an existing model.

In the Extras menu you will find the administrator settings, the license tool, the configuration tool and the language setting. Select Extras > Settings to change general program settings such as the names of individual elements or table settings.

As in Windows, at the end of the menu bar you will see the Help entry which you can use to navigate in the KISSsoft manual and in the KISSsys program. Click on `Help > Info` to find information on the program version and on the support provided by KISSsoft.

In the Window main menu you will find actions for organizing the opened sub-windows such as tables and 3D views. The printing action is only enabled if a table is open.

In addition to the main menu, KISSsoft uses context menus in many places. Use context menus to access actions for a particular area or model element. Context menus are normally called up by clicking the right-hand mouse button.

Select the Tool bar for rapid access to the menu actions that you need to use most often. Also note the tool tips: they display information about the actions in the Tool bar and also the more detailed explanations in the status bar.

62.4 Extended functionality for developers

In addition to the functionality already described, more functions are available for developers.

- To open a template file, click on File>Open Templates. Alternatively, click on Insert > Default templates to load the template file. It is displayed as a tree under Templates.
- To add new elements in tree view, you can "Copy" and "Paste" them. The new elements are added as copies from a template file.
- You can rename and delete elements via context menu functions.
- The data in the `Properties` dialog can be edited. New variables can be added and deleted
- Hidden variables will be displayed and all functions can be performed.
- Hide messages by selecting `Extras>Suppress messages`.

62.4.1 Properties dialog

In tree view, or in the diagram for an element, you can open the KISSsys `Properties` dialog via the context menu. In it you can add new variables or change existing ones. Only one `Properties` dialog is available. A second one will not be displayed.

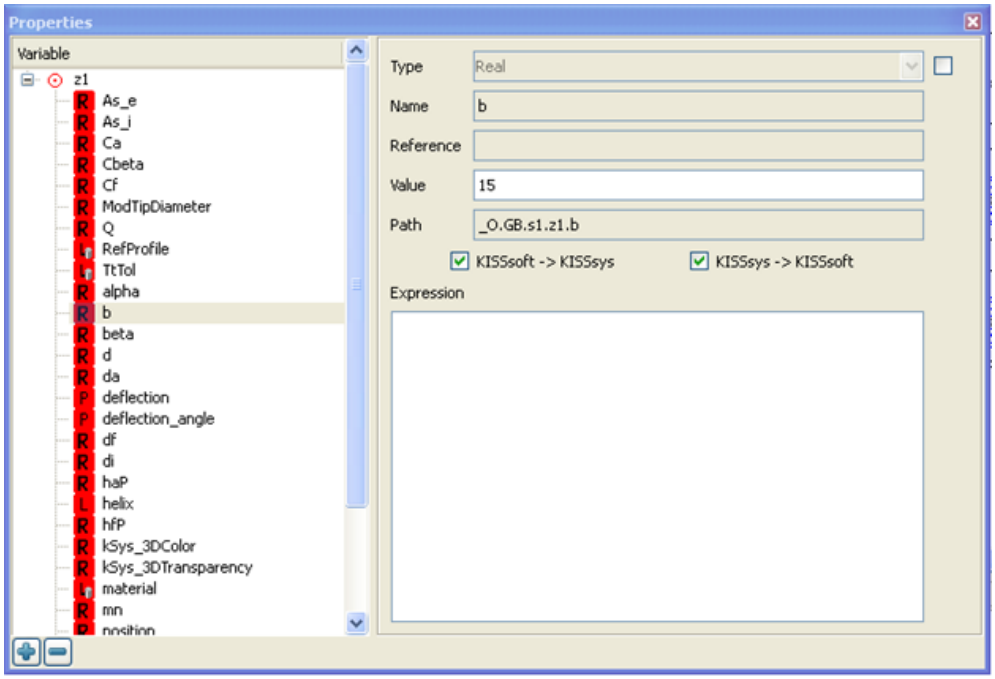


Figure 61.2: The KISSsys Properties dialog

<Figure 51>.2 shows the `Properties` dialog. On the left you see a tree view in which you can select data elements or variables, and on the right you see a dialog for the selected variable. The following fields are available for the variables:

- **Type:** Display the Variable type (see section "Variables" on page IX-1083).
- **Name:** The name of the variable. You can change the name here. However, if a variable has to be used in formulas or references, you must also change the name there, as otherwise the variable cannot be found.
- **Reference:** Enter the reference target here for reference elements. A name must be entered in quotation marks. An alternative would be the name of a string variable (see page IX-1084). In the case of Variants (see page IX-1084) the index must be entered here in an array. Here, an invalid reference will be marked in red.
- **Value:** The current value of the variable.
- **Expression:** An Expression used for calculating the variable (see page IX-1090). The value will be calculated on the basis of the expression, if an expression is present.
- **"KISSsoft** → KISSsys" flag. The variable can be transferred from KISSsoft to KISSsys.
- **Flag "KISSsys** → KISSsoft" flag. The variable can be transferred from KISSsys to KISSsoft.

Click the `Reference` and `Variant` buttons to convert the variable into a reference or variant variable and back.

62.4.2 Table view

The format of the tables is defined in the hidden `definition` variable. There are different types:

- **Table for calculations:** This table is best suited for displaying the data for several elements of the same type. The definition format is:

```
[[type,rows,columns],['variable1','variable2',..],
[element1,element2,..]]
```

In the case of type 1, you can edit each displayed value, in the case of type 2 you can edit all values that have no expression, and in the case of type 3 you can edit all values for which the `KISSsys` → KISSsoft flag has been set. The Number of Rows or Columns is not used.

- **Table for arrays or variants**: In this table, the arrays or variant variables are displayed in separate columns. The definition format is:

  ```
  [[type,rows,columns],['variable1','variable2',..]]
  ```

  In the case of type 21, you can edit each displayed value, in the case of type 22 you can edit all values that have no expression, and in the case of type 23 you can edit all values for which the `KISSsys` → KISSsoft flag has been set. The Number of Columns is not used.

- **Table for user interface:** You can configure this table to suit your needs. The definition is `[[type,rows,columns],[[A1,B1],[A2,B2]]]`. The contents can be inserted via a context menu in the table, and should not be changed in the definition. Since the definition is changed interactively, you must not set an expression here. The number of rows or columns should also only be changed via a dialog, as otherwise information on reference elements will be lost.

62.5 The existing elements

62.5.1 Variables

The following variables can be used:

- **Real**: A numerical value.
- **String**: A character string. Input in quotation marks e.g. "`Text`".
- **Point**: A coordinate or vector with 3 components. Input in {`1,2,3`} format.
- **Array**: A one-dimensional or multidimensional field. Input e.g. as `["Text",1.23,{1,2,3},[1,2]]`.
- **Function**: An executable function. Input best entered via the special input screen.
- **ElementID**: The ID of a Classcad object. Output as $31, input as name of the object with no quotation marks.
- **List**: Displayed as selection list and acts as a number in the Interpreter (index of the list beginning with 0). The selection list is defined as an array via the `Edit list` menu item, e.g. ["`one`","`two`","`three`"].
- **Database List**: The name from the KISSsoft database is displayed in a selection list. In the Interpreter, this type also acts as a number according to the database ID. The database assignment is defined as an array via the `Edit list` menu item: ["`database`","`table`"]

Each of the variables has a name, a value, an expression and different flags. If an expression is present, the value of the variables is defined via this expression. The expression is therefore particularly suited for the input of formulas. If, in contrast, a formula is entered in place of the value, this formula will be evaluated and the result will be assigned. The actual formula will be lost. The `KISSsoft->KISSsys` and `KISSsys->KISSsoft` flags determine how data is exchanged between the two programs. Only variables with the appropriate flag activated will be exchanged.

In the case of functions, the function is placed in the expression, and the value has no meaning.

For the `Real`, `String`, `Point`, `List` and `Database List` data types there are additional reference elements and variant elements.

62.5.1.1 References

A reference element behaves like any other variable, with the difference that another variable fetches the data. A valid variable name must be entered as the target for the reference element. The reference target must be entered as a character string. This will be either an actual name in quotation marks or an expression resulting in a character string, e.g. a concatenation of character strings (e.g. `gear1+'.z'` with the string variables `gear1` or `'gearwheel1.z'`). The system marks an invalid reference in red.

62.5.1.2 Variants

Internally, the variant elements administer a field of variables, whereas externally they behave like a normal variable. As additional data, the variant is assigned an index variable, which indexes the field. The index variable must be entered as an `array` of variables (e.g. `[system.index]`). With these data types you can store load spectra or system variants and the results can be displayed in tables.

62.5.2 Calculation elements

All the elements for KISSsoft calculations are derived from classes which begin with the name `kSoft`. In tree view they have a bitmap with a blue background.

The calculation elements have a series of functions:

- **Calculate**: performs a KISSsoft calculation in the background.
- **kSoftInterface**: starts KISSsoft interactively
- **kSoftReport**: performs the calculation and displays the report.
- **SetFlags**: Sets the flags for data exchange between KISSsoft and KISSsys to suit the required storage location.
  - **Save in KISSsys**: The data will be passed on in both directions.
  - **Save in KISSsoft**: Data with a stored expression will be transferred from KISSsys to KISSsoft, and all other data will only be transferred in the other direction.

  This function sets the flags only once when selected. It therefore has no effect on later changes.
- **kSoftModul**: This hidden function displays the KISSsoft module descriptor.
- **getTranslationTable**: This hidden function shows the translation table for variable names from KISSsys to KISSsoft. In the calculation element, the translation table can be extended via the `TranslationTable` array: For

example, an entry of `[['eps_a_min','ZP[0].Eps.aEffI'],['eps_a_max','ZP[0].Eps.aEffE']` adds a link between the variables `eps_a_min` and `eps_a_max` and the corresponding KISSsoft variables. Until now the names of the KISSsoft variables could only be taken from the report templates, `*.rpt`.

- **getUtilization**: This function returns the utilization, and the required safety/safety ratio.

In the `fileName` variable you can specify a KISSsoft calculation file which will automatically be loaded at the start of the calculation, before any other variables are transmitted. You can use the `savingMode` variable to specify whether this KISSsoft calculation file should be saved automatically:

- **Don't ask and don't save** When KISSsoft is shut down you will not be asked if the file should be saved after changes have been made to it.
- **Ask for saving** When KISSsoft is shut down you will be asked if the file should be saved. (KISSsoft default response)
- **Save automatically** When KISSsoft is shut down, the calculation file will be saved automatically without a user confirmation prompt.
- **Save file in KISSsys** No file name will be entered in `fileName`. Instead, the entire calculation file will be saved in the KISSsys element.

The shaft calculation contains the special method UpdateShaftElements. This must be called up if an element of force is to be added/deleted on a shaft. It evaluates the type and number of elements of forces on the shaft and transfers them into the 'forces' array in the shaft calculation. This array is a defining factor for the forces in the shaft calculation.

62.5.2.1 Relationship of calculations with elements

Templates are provided which automatically link the calculation with the shafts and gears. To do this, use the `Dialog` function. In the case of fundamental changes, i.e. when more elements of forces are added to the shaft, this dialog must be called up again to update the relationships.

62.5.2.2 Storage strategies for calculations

There are different options for saving the calculation data:

1. All data is to be saved in the KISSsys file and the calculations can only be accessed via KISSsys: Select `Save file in KISSsys` in `savingMode`. It is best to set the flags bidirectionally.
2. All data is to be saved in a KISSsoft file and the file can also be changed outside of KISSsys: Select `Ask for saving`, or `Save automatically` in `savingMode`. Use `SetFlags` to set the mode to `Save in KISSsoft`. Note here that the calculation data will only be loaded from the KISSsoft file when the calculation is called up for the first time. After the KISSsys file is opened, you should therefore call up `kSoftCalculate` occasionally.

62.5.2.3 Importing existing KISSsoft calculations

If there are already KISSsoft calculations present for elements of a new KISSsys system, you can simply load the files into the KISSsoft window. However, you should note a few points:

- The file name under `fileName` in the KISSsys calculation element will be changed. The name must either be deleted or modified.
- During the shaft calculation the elements of forces and the bearings are overwritten. For this reason, you need to call up the `dialog` or the `UpdateShaftElements` function after importing the calculation. The elements of forces and bearings cannot be imported, and neither can the positions. This data must be entered in KISSsys.
- In the case of gears you must ensure that the sequence of the gears matches up.

62.5.3 Elements for shafts

Different elements can be placed onto shafts. They will also be transferred into the KISSsoft shaft calculation. The position on the shaft is defined with the `position` variable.

- **kSysHelicalGear**: A cylindrical gear.
- **kSysBevelGear**: A bevel gear. The position of the peak is defined by the `direction` variable.
- **kSysWorm**: A worm.
- **kSysWormGear**: A worm wheel.
- **kSysCoupling**: A coupling. Diameter `d` and Width `b` can be entered for the 3D display.

- **kSysBearing**: A normal type of bearing. Losses can be recorded in `Tloss`. The direction of the loss torque should be defined with a `sign (speed)` in the expression.
- **kSysRollerBearing**: A rolling element bearing. The bearing geometry will be loaded from the KISSsoft bearings database during each refresh. Losses can be recorded in `Tloss`. The direction of the loss torque should be defined with a `sign (speed)` in the expression.
- **kSysCentricalLoad**: A centrical load. KISSsys will always prompt with a torque (Ty) but no power. This torque will also be included in the kinematics calculation.
- **kSysMass**: An additional mass on the shaft.
- **kSysRopeSheave**: A rope sheave. Unlike the torque, the belt force will not be calculated via the connection. It is up to the user to ensure that the belt force matches up in two belt pulleys.
- **kFaceGear:** A face gear.

62.5.4 Connection elements

- **kSysGearPairConstraint**: a connection between two cylindrical or bevel gears.
- **kSysPlanetaryGearPairConstraint**: a connection between a gear and a planet. You can select the type of pairing: sun-planet, planet-internal gear or planet-planet. Both gears must also be entered in this sequence. In addition, a planet carrier must be selected. The number of planets needs to be defined in the `NofPlanets` variable in the planet carrier coupling.
- **kSysPlanetaryBevelGearConstraint**: a connection between a bevel gear and a rotating bevel gear for bevel gear differentials. As in the case of the planetary connection, the sequence of the bevel gears and the number of planets must be defined. An efficiency cannot be specified here.
- **kSysWormGearConstraint**: a connection between a worm and wormwheel. Optionally, you can define two efficiencies (`eta1` and `eta2`) for the driving worm or driving gear.
- **kSysCouplingConstraint**: a connection with transmission ratio 1 between two couplings. The kinematic force of the coupling can be activated or deactivated. Additionally, it is possible to specify a slip, e.g. for flake graphite couplings or synchronizations. The torque in the connection will usually be calculated, but it can also be specified.

- **kSysBeltConstraint**: a connection between belt sheaves. The transmission ratio is calculated from the diameter ratio. A slip and an efficiency can be specified. When you are inputting the slip, take into account the transmission ratio and the sign. The calculation is performed in accordance with: n_1 - d_2/d_1 . n_2 = slip
- **kSysConnectionBearing:** A connection between two shafts
- **kSysConnectionRollerBearing:** A connection between two shafts
- **kSysSynchronizer:** A shaft synchronizer

  Using the `setConfig(slipConstraint_r/[slipConstraint_r, slip_r], torqueConstraint_r/[torqueConstraint_r, torque_r])` function you can activate or deactivate the connection:
  1. Closed, without slip: `setConfig([TRUE, 0], FALSE)`,
  2. Open, without torque: `setConfig(FALSE, FALSE)`,
  3. Open, with torque: `setConfig(FALSE, [TRUE, 20])`

- **kSysSpeedOrForce**: An element for specifying speed or torque. Both values can either be specified, otherwise they will be calculated. For the torque, you can also preset the power as an alternative.

  Using the `setConfig(speedConstraint_r, torqueConstraint_r/[torqueConstraint_r, type_r, torque_r])` function you can change the presets. If you specify a load type, the values below have these meanings: 0..torque with sign, 1..torque driving, 2..torque driven, 3..power driving, 4..power driven. Examples:
  1. Speed and torque specified: `setConfig(TRUE, TRUE)`,
  2. Speed and torque with value specified: `setConfig(TRUE, [TRUE, 0, 20])`,
  3. Only driving power specified: `setConfig(FALSE, [TRUE, 3, 20])`

62.5.5 Displaying elements in 3D graphics

Each element has an `OnRefresh3DView` function which generates the 3D display. If necessary, this function can be overwritten. You can set the color of an element in the range from 0 to 255, with the `kSys_3DColor` variable, and set the transparency with the `kSys_3DTransparency` variable. These two variables must be created if necessary.

62.5.6 System settings

You can make use of a series of setting options in the `System` element:

- **kSoftAcceptChanges**: The default setting `yes` means the changes will be transferred from KISSsoft. If the setting is `no`, nothing will be transferred. The setting `asked` means you are prompted to confirm whether the changes should be transferred when KISSsoft is shut down.
- **kSysKinematicFunc**: During the kinematics calculation you can call up the `OnCalcTorque` function. The standard implementation of this function calls up the calculation of the bearing actions for all shafts.
- **kSysKinematicMode**: The calculation of the kinematics can either be iterative or not. Iterations for the torque must be activated if the efficiency needs to be included. Iterations for speeds are only necessary if formulas for speeds have been entered.
- **kSys3DElements**: You can optionally display graphical elements or solid elements (3D kernel required). Graphical elements will be generated faster, although solid elements are more detailed, and it is for example possible to also display a loaded housing.
- **project_name**: The project name will be displayed in the KISSsoft calculation reports.
- **project_contract**: The commission number will be displayed in the KISSsoft calculation reports.

62.6 Programming in the Interpreter

There are programming options in the expressions used in variables and in functions.

62.6.1 Expressions in variables

The programming options in expressions are restricted. No local variables may be used.

Between the data types, the operators are defined in accordance with Table 61.1. A series of mathematical functions is also available. They are listed in table 61.2.

Data type	Operations	Description
`Real`	+,-	Addition and subtraction
	*,/	Multiplication and division
	<,>=,=,!=,>=,>	Relational operators
	!,AND,OR	Logical operators
`String`	+,LEN	Concatenation and length operators
	<,>=,=,!=,>=,>,!	Relational operators
`Point`	+,-	Addition and subtraction
	*,**	Scalar and vector multiplication
	:x,:y,:z	Access to components
	LEN	Vector length
`Array`	[],+,LEN	Indexing, concatenation and length operator

Table 61.1: Permitted operators for data types

`abs(x)`	Supplies the value of x
`sign(x)`	Supplies the sign of x (+1, -1 or 0 if x=0)
`min(a,b,...)`	Supplies the smallest value of the arguments
`max(a,b,...)`	System supplies the largest value of the arguments
`a_r(x)`	System converts from degrees to radian measure
`r_a(x)`	System converts from curve to degrees
`sin(x)`	System calculates sin of x in the radian measure
`sinh(x)`	System calculates sinh of x

`asin(x)`	System calculates arcsin of x
`cos(x)`	System calculates cos of x in the radian measure
`cosh(x)`	System calculates cosh of x
`acos(x)`	System calculates arccos of x
`tan(x)`	System calculates tan of x in the radian measure
`tanh(x)`	System calculates tanh of x
`atan(x)`	System calculates arctan of x
`atan(y,x)`	System calculates arctan of y/x
`exp(x)`	System calculates e to the power of x
`ln(x)`	System calculates the natural logarithm of x
`log(x)`	System calculates the decadic logarithm of x calculates
`sqrt(x)`	System calculates square root of x
`pow(x,y)`	System calculates x to the power of y
`fmod(x,y)`	System calculates x modulo y

Table 61.2: Predefined mathematical functions

A variable's expression can contain the specified operations and any function calls. If limited expressions are to be used, the expression must begin with `#` and the result has to be returned with `RETURN`:

```
#
IF a>b THEN
   RETURN a;
ELSE
   RETURN b;
ENDIF
```

62.6.2 Functions

The different options for programming in functions are best described with the help of examples. A function's header looks like this:

```
// Transferred variables from calling program
PAR Parameter1, Parameter2;

// Declaration of constants
CONST PI=3.1415926, E=2.71828;
```

```
// Declaration of local variables
VAR a,b,c,d;
```

Here, the lines that begin with `//` are comments. Each of these three lines may only occur once, and the declared variables must be separated with a comma. A non-initialized parameter or variable is `VOID`. This can be checked with `ISVOID(variable)`.

Limited statements have two variants: `IF` or `SWITCH` statements:

```
// IF statement with optional ELSIF and ELSE Block
IF Parameter1 > 5 THEN
    a = sin(PI*Parameter1);
ELSIF Parameter1 < 0 THEN
    a = Parameter1;
ELSE
    a = 0;
ENDIF

// SWITCH Statement with selection of figures or texts
SWITCH Parameter2
CASE 'Zero': b = 0;
CASE 'one': b = 1;
DEFAULT: b = 5;
ENDSWITCH
```

For loops, there are four program variants:

```
// FOR Loop with optional intervals
FOR a = 1 TO 8 STEP 2 DO
    b = b + a;
    IF b>100 THEN
        BREAK;      // Ends the Loop
    ENDIF
NEXT

// WHILE Loop
WHILE b<100 DO
    b = b*10;
WHEND

// DO Loop
DO
    b = b*10;
UNTIL b>100;

// FORALL Loop is executed for all elements of an array
c = [1,2,3,4,5,6,7,8,9];
a = 0;
FORALL c d DO   // d receives each of the values of an element from c
    a = a + d;
NEXT
```

There is a special syntax for calling up functions that belong to objects. The standard method is to specify the object name followed by a point and the name of

the function. However, the name of an object can also be contained in a local variable. This allows you to change the object for the function call at runtime.

```
// The OBJ_GetMember function is called for Object1
Object1.OBJ_GetMember('variablenname');

// a is a local variable of Type String with the  name of an object
a = 'Object1';

// This calls a Service function for the object with the name a
b = a.OBJ_GetMember('variablenname');

// calls a function created by the user for Object1.
a.Userfunction();

// the function created by the user is called
// for the current object.
Userfunction();

// the function created by the user is called
// for the hierarchically superior object.
^.UserFunction();
```

The system searches for variable names relative to the current object. If `object.z` is used in an expression, the system will first of all attempt to find this variable below the current object. If it is not present, the search will continue in the hierarchically superior object (in accordance with `^.object.z`) and so on.

62.6.3 Important service functions

`OBJ_GetChildren()`	Supplies an array with all child objects.
`OBJ_GetName()`	Supplies the name of the object.
`OBJ_GetId()`	Supplies the ID of the object.
`OBJ_GetId()`	Supplies the ID of the object.
`OBJ_HasMember()`	Tests whether a variable is present
`OBJ_GetMember()`	Supplies the variable of the current object.
`OBJ_FindMember()`	Supplies the variable of the current or hierarchically superior object.

Table 61.3: Important service functions

62.6.4 Variable dialogs

In interpreter functions, variable dialogs can be generated for the input of variables. The call is:

```
res = CADH_VarDialog([“Title“, Width, Height, Pitch],
[Dialogelement1], [Dialogelement2], ...);
```

The title will be displayed in the title line of the dialog, and width and height show the dialog's dimensions in pixels. The pitch (between 0 and 1) describes the relationship between the width of the field description and the dialog width (default value 0,4). This definition of the dialog size can be followed by any number of arrays with the definition of the individual dialog elements.

The return value is an array. Its first value is `res[0] =1` if the dialog ends with `OK`, otherwise it will be zero. The other elements of the returned array supply the results of the input fields.

Below, the following convention is used to define the type of a variable: _str=String, _n=Int, _r=Real, _b=Bool. For example, in the case of Caption_str, this means that the variable Caption is of the type String.

62.6.4.1 Dialog elements for the variable dialog

The following dialog elements are available for the variable dialogs:

Horizontal grouping:

The horizontal grouping provides a framework in which the individual dialog elements are lined up beside each other. Their position must always be defined by a vertical group, which means that all dialog elements contained within a horizontal grouping must be defined in a vertical group. A horizontal group is defined as follows:

[C:VDGL_HORZ,Caption_str,DistAbove_n,DistAfter_n,[Dialogelem]]

- C:VDGL_HORZ: Type definition for horizontal grouping.
- Caption: Caption of the horizontal grouping. If "Caption" is not an empty string, a frame will be drawn around the horizontal group.
- DistAbove: distance above the horizontal group to the next dialog element.
- DistAfter: distance behind the horizontal group to the next dialog element. "DistAfter" and "DistAbove" are specified in pixels.
- [Dialogelem]: Element array for the definition of the dialog elements located in the horizontal grouping. This array may only contain elements of the type VDGL_Vert.

Vertical grouping:

The vertical grouping provides a framework in which the individual dialog elements will be lined up below each other. The width of the dialog elements is defined by the vertical group. A vertical group is defined as follows:

[C:VDGL_Vert,Caption_str,[XStart_r,XEnd_r],XPart_r,[Diag],Marg_n]

- C:VDGL_Vert: Type definition for vertical grouping
- Caption: Caption of the vertical grouping. The vertical grouping always has a frame drawn around it.
- [XStart,XEnd]: XStart and XEnd define a factor (between 0 and 1) for the width of the vertical group with reference to the width of the hierarchically superior dialog. Additionally, they define the X-position of the vertical group.
- XPart: Factor between 0 and 1 that defines the ratio between the prompted value and the input value for the dialog fields (the text assigned to an input field is called the "prompt"). If XPart=-1 the prompt will be positioned above the dialog element.
- [Diag]: Element array used to define the dialog elements located in the vertical grouping.
- Marg (margin): An optional parameter defining the displacement of the dialog elements in relation to the edge of the vertical group, which means that the dialog elements contain the distance "Marg" (margin) both from the left-hand and from the right-hand edge of the vertical group.

RealEditFeld:

Provides an edit box in which the user can input a floating comma number.

[C:VDGL_Real,Prompt_str,Preset_r,res,res,Places_n]

- C:VDGL_Real: Type definition of RealEditFeld.
- Prompt: Text assigned to the input field.
- Preset: preset value.
- res: Here, a space is reserved for two optional parameters which are not in use at present. However, these spaces must not be left empty in the definition (e.g. [C:VDGL_Real,Prompt,Preset,0,0,Places] would be a correct solution but not [C:VDGL_Real,Prompt,Preset,,,Places]).
- Places: This is an optional parameter defining the number of decimal places of the input field.
- ReturnVal: (return value). The return value is the input string.

IntEditFeld:

Provides an edit box in which the user can input a whole number.

[C:VDGL_Int,Promt_str,Vorgabe_n]

- C:VDGL_Int: Type definition of IntEdit field.
- Prompt: Text assigned to the input field.
- Preset: preset value.
- ReturnVal: (return value). The return value is the input string.

StringEditFeld:
Provides an EditBox in which you can input text.

[C:VDGL_Str,Promt_str,Vorgabe_str]

- C:VDGL_Str: Type definition of the StringEdit field.
- Prompt: Text assigned to the input field.
- Preset: Preset value.
- ReturnVal (return value): The return value is the input string.

Text display:
The system generates a text display. If an empty string is entered instead of text, the text field can also be used to define a distance.

[C:VDGL_Prompt,Prompt_str,Fieldheight_n]

- C:VDGL_Prompt: Type definition of text display.
- Prompt: Field text.
- Fieldheight: Height at which the text is displayed.

IntComboBox:
Provides a combo box in which the user can input a whole number.

[C:VDGL_IntCom,Prompt_str,[Entr_n],Sign_n/[Ind_n],0,0,AsVal_b]

- C:VDGL_IntCom: Type definition of IntComboBox.
- Prompt: Text assigned to the combo box.
- [Entr]: Element array of the available list items (in the case of an IntComboBox the components must be whole numbers).
- Sign/[Ind]: Here you have the option of using "Sign" to either set a constraint value, which is contained in the list, directly, or using "Ind" to select a value in a particular list position as a constraint value (the first element in the list is located at position 0)."Sign" or "[Ind]" are optional parameters.

- AsVal: If the optional parameter "AsVal" has been set and is not 0, the return value becomes the input. Otherwise the return value is the index of the selected entry.

IntEditComboBox:
Provides a editable combo box in which the user can input a whole number. Please note that the values entered here are whole numbers.

[C:VDGL_IntComE,Prompt_str,[Entr_n],Sign_n/[Ind_n]]

- see IntComboBox
- ReturnVal: (return value). The return value is the input string.

RealComboBox:
Provides a combo box in which the user can input a floating comma number.

[C:VDGL_RealCom,Prompt_str,[Entr_r],Sign_r/[Ind_n],0,0,AsVal_b]

- see IntComboBox

RealEditComboBox:
Provides a editable combo box in which the user can input a floating comma number.

[C:VDGL_RealComE,Prompt_str,[Entr_r],Sign_r/[Ind_n]]

- see IntComboBox
- ReturnVal: (return value). The return value is the input string.

StringComboBox:
Provides a combo box in which the user can input a string.

[C:VDGL_StrCom,Prompt_str,[Entr_str],Sign_str/[Ind_n],AsPos_n]

- see IntComboBox
- AsPos: Contrary to the IntComboBox, the return value here represents the index of the selected field, if the optional parameter "AsPos" has been set and is not 0. Otherwise the return value is the input.

StringEditComboBox:
Provides a editable combo box in which the user can input a string input.

[C:VDGL_StrCom,Prompt_str,[Entr_str],Sign_str/[Ind_n]]

- see IntComboBox
- ReturnVal (return value): The return value is the input string.

Code button:

Service button:

62.6.4.2 Example application of a variable dialog

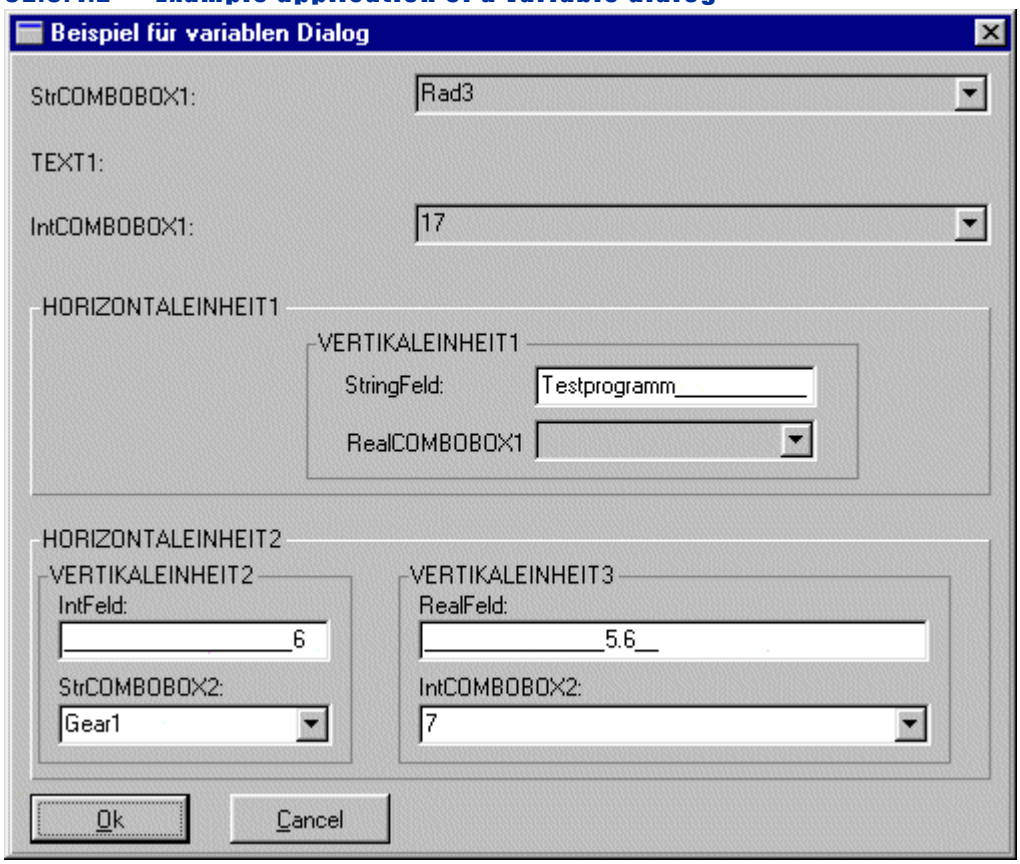


Figure 61.3: Example of a variable dialog

The section below uses the example of the program code for the variables dialog shown in Figure 61.3, in which the greatest possible number of elements are used:

```
//  VARIABLES DECLARATION

VAR res,result1,result2,result3,result4,result5,fullResult;

// DIALOG AND INPUT DATA

res = CADH_VarDialog(["Example of Variable Dialog",500,400,0.4],

      [C:VDLG_StrCom,"StrCOMBOBOX1:",["Gear1","Gear2","Gear3"],[2],0],
```

```
[C:VDLG_Prompt,"TEXT1:",30],
[C:VDLG_IntCom,"IntCOMBOBOX1:",[12,17,19],17,0,0,1],
// HORIZONTAL GROUP  WITH  ONE  VERTICALGROUP
[C:VDLG_HORZ,"HORIZONTAL UNIT1",20,10,
  [     // Warning: remember brackets!
    [C:VDLG_VERT,"VERTICAL UNIT1",[0.3,0.9],-1,
      [
        [C:VDLG_Str,"StringFld:","Test Program"],
        [C:VDLG_RealComE,"RealCOMBOBOX1",[5.3,7.1,9.1],[2]]
      ],
      20
    ]
  ]    // Warning: remember brackets!
],
// HORIZONTAL GROUP  WITH  TWO  VERTICAL UNITS GROUPS
[C:VDLG_HORZ,"HORIZONTAL UNIT2",10,10,
  [
    [C:VDLG_VERT,"VERTICAL UNIT2",[0.01,0.35],-1,
      [
        [C:VDLG_Int,"IntFld:",6],
        [C:VDLG_StrComE,"StrCOMBOBOX2:",["Gear1","Gear2"],[0]]
      ],
      10
    ],
    [C:VDLG_VERT,"VERTICAL UNIT3",[0.4,1],-1,
      [
        [C:VDLG_Real,"RealFld:",5.6,0,0,3,3],
        [C:VDLG_IntComE,"IntCOMBOBOX2:",[5,7,9],7]
      ]
```

```
          ]

        ]

      ]

    );

// res [0] contains 1 if  OK was pressed , or   else

IF res[0] THEN

    // READ RESULTS FROM DIALOG:

    result1 = res[1];    //res [1]= Gear3

    result2 = res[2];    //res[2]= TEXT1:

    result3 = res[3];   //res[3]= 17

    result4 = res[4];   //res [4]= [["Test Program",9.1]]

    result5 = res[5];   //res[5]= [[6,"Gear1"],[5.6,7]]

    fullResult=res;     //res=["Gear3","TEXT1:",17,[[''Test
Program",9.1]],[[6,"Gear1"],[5.6,7]]]

    CADH_Message(fullResult);

ENDIF
```

62.6.4.3 Interactions with variable dialogs

It is possible to interact with variable dialogs. Selections in lists, changes in input fields and selections in lists can trigger callbacks to a user-defined function. Then, it is also possible to change dialog elements from this callback routine.

You set a local function as a callback via the title input in the variable dialog:

```
res = CADH_VarDialog([[“Title“,PROC(Callback)], Width,
Height, Pitch], [Dialogelement1], [Dialogelement2],
...);
```

The local callback function will now be called up if there are changes in the dialog. The function is declared as follows:

```
PAR res;

PROC Callback
PAR handle, elemNo, event, eventPar;

IF TYP(elemNo)=STRING THEN
    IF elemNo='@combo' AND event=C:CBN_SELCHANGE THEN
        IF eventPar=0 THEN  // own input, enable input
            CADH_VarDialogAccess(handle,[['@input1',C:VDLG_ENABLE,TRUE]]);
```

```
        ELSE  // disable input, set value to zero
            CADH_VarDialogAccess(handle,[['@input1',C:VDLG_ENABLE,FALSE],
                                         ['@input1',c:VDLG_ASSIGN,0]]);
        ENDIF
    ENDIF
ENDIF
ENDPROC

res = CADH_VarDialog([['Title',PROC(Callback)], 400, 400 0.4],
                     [[C:VDLG_Real,'@input1'],'Input1:',2],
                     [[C:VDLG_StrCom,'@combo'],'Selection:',
                         ['own input','calculate'],[0],TRUE]);
```

A handle is transferred to the dialog as a code parameter, plus an element identifier, the event, and additional parameters. The possible events are:

Element type	**Event**	**Parameter**
Dialog	Initialization `WM_INITDIALOG`	none
Combobox	Selection `CBN_SELCHANGE`	Current value
Input field	Leave field `WM_KILLFOCUS`	Current value
Button	activated `BN_CLICKED`	none

Either the number of the element according to the index in the results array is transferred as the element number, or the name of the element is transferred. Like in the example, a name can be defined by transferring an array, with a type and name, into the array's first element for the dialog element.

Access from the callback routine to the dialog is via this function:

```
CADH_VarDialogAccess(handle, [[elemNo, action,
param],[elemNo, action, param],...]
```

Here, the following actions are permitted:

Action	**Description**	**Parameter**
`DLG_ASSIGN`	Assignment to input field	New value
`VDLG_SELECT`	Selection in combo box	[position]/value
`VDLG_ENABLE`	Activate or deactivate	TRUE/FALSE
`VDLG_SETFOCUS`	Focus on new element	Element's ID

If no action is specified, the value in the input field will be returned. The return takes the form of an array with as many elements as code parameters.

62.6.5 Defining 2D graphics

In `KISSsys` you can generate two-dimensional graphics for displaying results which are present in arrays. You can store the definition of the graphic in the `data` variable expression of the `kSys2DPlot` graphical element. Beam and line graphics can be displayed in parallel. The definition of the graphic consists of three parts:

- Axis system (1 or 2 axis systems can be defined)
- XY-line graphics
- Beam graphic

Below, each of these parts is described in more detail.

62.6.5.1 The definition of the axis system (af)

At least one axis system must be defined. The second one is optional. The definition for the axis system is as follows:

```
[ | Xaxisname_str , | min_x_r , | max_x_r ] , [ | Yaxisname_str , | min_y_r
, | max_y_r ] , [ axiscolor_str/array , | axiscross_x_r , axiscross_y_r ] ,
[ | scaleinterval_x_r , | scaleinterval_y_r , [ | exponential_x_n , |
exponential_y_n ]
```

where :

- **XAxisname**: Name of the X-axis.
- **YAxisname**: Name of the Y-axis.
- **min** : Minimum value of the axis (optional).
- **max** : Maximum value of the axis (optional).
- **axiscolour** : Color of the axis defined in a string (red ,green, blue, yellow, white, gray, cyan, brown, magenta, purple, black) or as an array [r_n , g_n , b)_n](where r, g, b represent the red, green and blue color values from 0 to 255 (optional).
- **axiscross** : The intersection point of the axes (optional).
- **scaleinterval** : Increment of the axis scaling.
- **exponential** : If 1 is input, the axis will be logarithmically subdivided.

62.6.5.2 The definition of an XY-line graphic (dg_l)

For an XY-line graphic the following information is required:

```
grouptype_n , [ dataarray_x_r ] , [ dataarray_y_r ] , [ | linename_str , |
|linecolour_str/array , | linestyle_n ] , | assignaxis_n
```

where :

- **grouptype** : = 1 (for lines graphic).
- **dataarray** : Contains the X or Y coordinates of the data.
- **linename** : Name of the element.
- **linecolour** : Line color.
- **linestyle** : Line type (0- solid, 1- interrupted, 2- dashed, 3- semicolon, 4- dash dot dot)
- **assignaxis** : Number 1 or 2 of the coordinates system

62.6.5.3 The definition of a beam chart (dg_b)

For a beam chart, a group of data is defined as follows:

```
grouptype_n , [ dataarray_1_r , ... , |dataarray_n_r ] , [
barcolour_str/array ] , | bargroupname_1_str , [ | barelementlabel_1_str ,
... , barelementlabel_1_str ] , | barclass_n
```

where :

- **grouptype** := 2 (for beam chart).
- **dataarray** : Contains the data for the group.
- **barcolour** : Color of the group's beams.
- **bargroupname** : Name of the group.
- **barelementlabel** : Names for individual elements.
- **barclass** : Display as group (=0) or sorted by elements (=1).

62.6.5.4 The entire definition

The entire definition must begin with the definition of the axis system. After this, you can list any number of definitions for line and beam charts. Each part definition must be enclosed in square brackets, just like the entire definition:

```
[ [af_1] , | [ af_2] , | [dg_l_1] , ..., | [ dg_l_ n1 ] , | [ dg_b_ 1 ] ,
..., [ dg_b_ n2 ]]
```

If lines and beams are to be used simultaneously, a second coordinates system will automatically be applied. This can, however, be changed by the definition of a second coordinates system. An example of the available options is listed as follows:

```
[
   [['x-AXIS'],['y-axis',0],[[40,250,150],[-1000,-10]],[30,20,0,0]],
   [['x-AXIS 2'],['y-axis 2',0],['blue',[0,0]],[30,20,0,0]],
   [1,[-1000,-500,0,500,1000],[5,20,40,55,71],['LINE1','red',0]],
   [1,[-1000,-500,0,500,1000],[2,20,46,60,83],['LINE2',[200,5,150],3]],
   [2,[5,25,16,10,4],['red',3],'group 1'],
   [2,[40,35,25,20,12],['red',3],'group 2']
```

The example shows two lines and two groups of beams in two separate coordinate systems.

62.6.5.5 Displaying the graphic

After the graphic has been defined in the `data` variable you can display the graphic with the graphical element's `Show` function. Later you can update it with the `Refresh` function in the menu or the graphics window.

Part X

Bibliography and Index

[1] A.G.M.A. - Fundamental Rating Factors and Calculation Methods for Involute Spur and Helical Gear Teeth. Norm 2001-B88, 1988

[2] Akahori H., Sato Y., Nishida T., Kubo A.: Prove di durata di Face Gear. (Testing the durability of face gears) Organi di trasmissione, 2002, No.12 or MTP2001-Fukuoka, The JSME Int. Conference, 2001, Japan.

[3] Basstein G., Sijtstra A.: Neue Entwicklung bei Auslegungen und Fertigung von Kronenrädern. Antriebstechnik, 32 (1993), No.11

[4] Bock G., Nocj R., Steiner O.: Zahndickenmessung an Getriebeschnecken nach der Dreidrahtmethode. Physikalisch-Technische Bundesanstalt, Braunschweig, 1974

[5] Decker K.H.: Maschinenelemente. Carl Hanser Verlag München, 10th Edition 1990

[6] Dietrich G., Stahl H.: Matrizen und Determinanten in der Technik. VEB Verlag Leipzig, 5th Edition, ca. 1960

[7] DIN ISO 15312: Wälzlager - Thermisch zulässige Betriebsdrehzahl - Berechnung und Beiwerte, October 2004

[8] DIN 732: Wälzlager - Thermische Bezugsdrehzahl - Berechnung und Beiwerte, May 2010

[9] DIN 743: Tragfähigkeitsberechnung von Wellen und Achsen. December 2012

[10] DIN 867: Bezugsprofile für Evolventenverzahnungen an Stirnrädern (Zylinderrädern) für den allgemeinen Maschinenbau und den Schwermaschinenbau. Issue February 1986

[11] DIN 2091: Drehstabfedern mit rundem Querschnitt: Calculation and Design. DIN Taschenbuch 29, Beuth Verlag Berlin, 2011

[12] DIN 2092: Tellerfedern: Berechnung. DIN Taschenbuch 29, Beuth Verlag Berlin, 2011

[13] DIN 2093: Tellerfedern: Masse, Qualitätsanforderungen. DIN Taschenbuch 29, Beuth Verlag Berlin, 2011

[14] DIN EN 15800: Zylindrische Schraubenfedern aus runden Drähten: Gütevorschrift für kaltgeformte Druckfedern. DIN Taschenbuch 29, Beuth Verlag Berlin, 2011

[15] DIN 2096: Zylindrische Schraubenfedern aus runden Drähten und Stäben: Gütevorschrift für warmgeformte Druckfedern. DIN Taschenbuch 29, Beuth Verlag Berlin, 2011

[16] DIN 2097: Zylindrische Schraubenfedern aus runden Drähten: Gütevorschriften für kaltgeformte Zugfedern. DIN Taschenbuch 29, Beuth Verlag Berlin, 2011

[17] DIN 2194: Zylindrische Schraubenfedern aus runden Drähten und Stäben: Kaltgeformte Drehfedern (Schenkelfedern), Gütenorm. DIN Taschenbuch 29, Beuth Verlag Berlin, 2011

[18] DIN 3960: Begriffe und Bestimmungsgrössen für Stirnräder und Stirnradpaare mit Evolventenverzahnung. Issue December 1987

[19] DIN 3961: Toleranzen für Stirnradverzahnungen, Grundlagen. 1978

[20] DIN 3967: Flankenspiel, Zahndickenabmasse, Zahndickentoleranzen. 1978

[21] DIN 3971: Begriffe und Bestimmungsgrössen für Kegelräder und Kegelradpaare. Issue July 1980

[22] DIN 3975: Begriffe und Bestimmungsgrössen für Zylinderschneckengetriebe mit Achsenwinkel 90 Grad. Issue July 1976

[23] DIN 3990: Tragfähigkeitsberechnung von Stirnrädern. Parts 1, 2, 3, 4, 5, 11, and 21. Issue December 1987

[24] DIN 3991: Tragfähigkeitsberechnungen von Kegelrädern. 1990

[25] DIN 5480: Zahnwellen-Verbindungen mit Evolventenflanken. Parts 1 to 15. March 1986

[26] DIN 6885: Passfedern. Blatt 1-3. 1968

[27] DIN 6892: Passfedern - Berechnung und Gestaltung. 2012

[28] DIN 7151: ISO Grundtoleranzen für Längenmasse bis 500 mm. 1964

[29] DIN 7190: Berechnung und Anwendung von Pressverbänden. February 2001

[30] DIN EN 13906-1: Compression springs: Calculation and Design. DIN Taschenbuch 29, Beuth Verlag Berlin, 2011

[31] DIN EN 13906-2: Tension springs: Calculation and Design. Beuth Verlag Berlin, 2013

[32] DIN EN 13906-3: Turnsprings: Calculation and Design. DIN Taschenbuch 29, Beuth Verlag Berlin, 2011

[33] DIN 31652: Hydrodynamische Radial-Gleitlager im stationären Bereich. DIN Taschenbuch 198, Beuth Verlag Berlin, 1991

[34] DIN 31653: Hydrodynamische Axial-Gleitlager im stationären Bereich. DIN Taschenbuch 198, Beuth Verlag Berlin, 1991

[35] DIN 31654: Hydrodynamische Axial-Gleitlager im stationären Bereich. DIN Taschenbuch 198, Beuth Verlag Berlin, 1991

[36] DIN 58400: Bezugsprofil für Evolventenverzahnungen an Stirnrädern in der Feinwerktechnik. Issue June 1984

[37] DIN 58405: Abmasse für die Feinwerktechnik, Part 2.

[38] Dubbel H.: Taschenbuch für den Maschinenbau. Springer Verlag Berlin, 15th Edition 1986

[39] Eschmann P.: Die Wälzlagerpraxis. R.Oldenburg Verlag Munich, 1978

[40] FAG: Standardprogramm. Catalog WL 41510, 3rd Edition, 1995

[41] FKM Guideline: Rechnerischer Festigkeitsnachweis für Maschinenbauteile. VDMA Verlag Frankfurt, 6th Edition 2012

[42] Hänchen R., Decker K.H.: Neue Festigkeitslehre für den Maschinenbau. Carl Hanser Verlag Munich, 3rd Edition 1967

[43] Hirn H.: Computergestützte Zahnradoptimierung. Fink GmbH, Druck und Verlag Pfullingen, 1999

[44] ISO 6336: Calculation of load capacity of spur and helical gears. Parts 1, 2, 3, 4, 5. Edition 2006

[45] ISO/DIS 10300: Calculation of load capacity of bevel gears. Parts 1, 2, 3. ISO 10300 Edition 2001, ISO/DIS 10300 Draft 2011

[46] Kissling U.: KISSsoft - eine praxisgerechte Maschinenelemente-Software. antriebstechnik 27 (1988), No. 12, p. 34-40

[47] Kissling U.: Auslegung von Maschinenelementen. CIM Management 11 4, 1995

[48] Kissling U.: Technische Berechnungen auf Personal Computern. VDI-Z 130 (1988), No. 5, p. 45-52

[49] Kissling U.: Sicher dimensioniert. antriebstechnik 6 (2007), p. 64-68

[50] Kissling U., Beermann S., Hirn T.: Kronenräder: Geometrie und Festigkeit, antriebstechnik 10 (2003)

[51] Klingelnberg-Werknorm 3028: Auslegung eines Kegelradgetriebes ohne Achsversatz. Issue No. 2

[52] Klingelnberg-Werknorm 3029: Auslegung eines Kegelradgetriebes mit Achsversatz. Issue No. 2

[53] Klingelnberg-Werknorm 3030: Tragfähigkeits-Berechnung für Spiralkegelräder. Issue No. 1

[54] Klotter K.: Technische Schwingungslehre, Volume 2. Springer Verlag Berlin, 2nd Edition 1960

[55] Kollmann F.: Shaft-Hub Connections. Springer Verlag Berlin, 1984

[56] Lachenmaier, Sepp: Auslegung von evolventischen Sonderverzahnungen für schwingungs- und geräuscharmen Lauf von Getrieben. VDI Verlag Düsseldorf, WZL Reihe 11 Nr. 54, 1983

[57] Lang O., Steinhilper R.: Gleitlager. Konstruktionsbücher Volume 31, Springer Verlag Berlin, 1978

[58] Linke H.: Stirnradverzahnung. Carl Hanser Verlag München, 1996

[59] MAAG-Taschenbuch. 2nd Extended Edition, Zürich, 1985

[60] Massa E.: Costruzione di macchine. Editori Masson Italia, Milano, 1981

[61] Matek W., Muks D., Wittel H.: Roloff/Matek Maschinenelemente. Vieweg Verlag Braunschweig, 11th Edition 1987

[62] Matek W., Muks D., Wittel H., Becker M., Jannasch D.: Roloff/Matek Maschinenelemente. Vieweg Verlag Braunschweig, 15th Edition 2001

[63] Matthias K.: Schraubenkräfte in einer Flanschverbindung. Maschinenbau, Berlin 34 (1985) 11, p. 517.

[64] Niemann G.: Maschinenelemente, Volume 1. Springer Verlag Berlin, 2005

[65] Niemann G.: Maschinenelemente, Volume 2. Springer Verlag Berlin, 1983

[66] Niemann G.: Maschinenelemente, Volume 3. Springer Verlag Berlin, 1985

[67] NIHS 20-25: Uhrenindustrie, Swiss standard SN 282 025, October 1993

[68] Obsieger: Zahnformfaktoren von Aussen- und Innenverzahnungen. Zeitschrift Konstruktion 32 (1980), p. 443-447.

[69] Weber C., Banaschek K.: FVA report 129 and 134, Elastische Formänderung der Zähne und der anschliessenden Teile der Radkörper von Zahnradgetrieben, FVA 1955

[70] Rules for The Classification of Naval Ships (FREMM 3.1), Bureau Veritas, March 2004

[71] SKF: Hauptkatalog 4000 T. 1989 Edition

[72] Spinnler, Prof.: Manual de calcul d'organes des machines. EPFL Lausanne, 1990

[73] VDI 2226: Festigkeitsberechnung metallischer Bauteile.

[74] VDI 2227: Festigkeitsberechnung.

[75] VDI 2230: Systematische Berechnung hochbeanspruchter Schraubenverbindungen, Sheet 1. February 2003

[76] VDI 2545: Zahnräder aus thermoplastischen Kunststoffen. 1981 Edition

[77] KISSsoft: Klassische Anleitungen zu den Berechnungsmodulen: KISSsoft Gear Pump Analysis, Hombrechtikon, 2005

[78] Boresi A.P., Schmidt R.J.: Advanced mechanic of material , 6th. Edition, John Wiley and Sons, Inc., 2002, ISBN 0-471-39138-7.

[79] Karlheinz Roth, Evolventen-Sonderverzahnungen zur Getriebeverbesserung, Springer, 1998

[80] Hoechst High Chem, Technische Kunststoffe - Berechnen, Gestalten, Anwenden, B.2.2, Hoechst AG, 1992

[81] Theissen, J.: Berechnung der Sicherheit gegen Graufleckigkeit von Industriegetrieben auf der Grundlage des neuen Rechenverfahrens nach FVA 259. Dresdner Maschinenkolloquium, TU Dresden, Sept. 2003. Tagungsband p.195-212, ISBN 3-86130-201-2.

[82] FVA-Informationsblatt No. 54/7, Testverfahren zur Untersuchung des Schmierstoffeinflusses auf die Entstehung von Graufleckigkeit bei Zahnrädern, FVA Vereinigung, Frankfurt, 1999

[83] Feulner, R.: Verschleiss trocken laufender Kunststoffgetriebe, Lehrstuhl Kunststofftechnik, Erlangen, 2008

[84] DIN 32711 : Welle-Nabe-Verbindung - Polygonprofil P3G. Issue March 2009

[85] DIN 32712 : Welle-Nabe-Verbindung - Polygonprofil P4C. Issue March 2009

[86] Decker: Maschinenelemente, Funktion, Gestaltung und Ber echnung, Hanser Verlag Munich, 2001

[87] Klingelnberg, J.: Kegelräder Grundlagen, Anwendungen, Springer Verlag Berlin Heidelberg, 2008

[88] N. Norden, On the compression of a Cylinder in Contact with a Plane Surface, National Bureau of Standards, 1973

[89] R. Annast, Kegelrad-Flankenbruch, Technische Universität München, 2002

[90] VDI 2241: Schaltbare fremdbetätigte Reibkupplungen und -bremsen, Sheet 1: 1982; Sheet 2: 1984.

[91] R. Burdick, Manufacturing Single-Enveloping Worm Gear Sets, Gear Solutions, April 2003

[92] T. Harris, J. H. Rumbarger, C. P0. Butterfield, Wind Turbine Design Guideline DG03: Yaw and Pitch Rolling Bearing Life. 63 pp.; NREL Report No. TP-500-42362, 2009

[93] DET NORSKE VERITAS, Calculation of gear rating for marine transmissions, Norway, 2003.

[94] E. Haibach, Betriebsfestigkeit, Verfahren und Daten zur Bauteilberechnung, 2nd Edition, Springer Verlag 2002

[95] Winter H., Podlesnik B.: Zahnfedersteifigkeit von Stirnradgetrieben, parts 1 to 3, Antriebstechnik 22 1983

1

12.17.5.6 Calculating the displacement volume of gear pumps - II-438
18. Beveloid gears - II-550

2

2D geometry - II-574

3

3D export - II-608
3D geometry - I-179, II-584
3D interface to ASCON Kompas - I-238
3D interface to Autodesk Inventor - I-200
3D interface to CATIA - I-230
3D interface to CoCreate - I-233
3D interface to Creo Parametric (ProEngineer) - I-217
3D interface to Solid Edge - I-192
3D interface to Solid Works - I-184
3D interface to ThinkDesign - I-235
3D interface to Unigraphics NX - I-206
3D interfaces - I-173
3D view - IX-1070

A

Abbreviations used in gear calculation - II-298, II-634
Accuracy grade bevel gears - II-487
Accuracy of the tooth form - II-558
Add your own texts in the results window - I-98, I-244
Addendum angle and root angle - II-463, II-465
Addendum reduction - II-500, II-506
Add-in (menu items in CAD) - I-201, I-207
Add-in functions (calls) - I-189, I-198, I-204, I-216
Adding and deleting files - I-93
Adding manufacturing data - I-184, I-189, I-192, I-198, I-200, I-204
Adding manufacturing data on the drawing - I-212
Adding new bolt types to the database - IV-960
Adding tip chamfer - II-355
Adding tip rounding - II-354
Adding variables in tables - IX-1076
Additional inputs for DIN 6892 Method B - IV-822
Additional strength calculation of all variants - II-420

Adhesives - I-132
AGMA 925 - II-402, II-573
Allow large profile shift - II-429
Allow simplified calculation in accordance with DIN 3990/ISO 6336 - II-431
Allowances for racks - II-620
Alternating bending factor - II-288, II-293, II-475
Ambient density - III-651
Amplitude of contact stiffness - II-600
Analog to DIN 3991, Method B - II-503
Analog to ISO 10300, Method B - II-502, II-503
Angle error - II-557
Angle modifications - II-466
Angle of flank normal - II-581
ANSI 92.1 and ISO4156/ANSI 92.2M - IV-870
Answers concerning geometry calculation - II-612
Answers to Frequently Asked Questions - I-239, II-611, III-786, IV-959
Answers to questions about strength calculation - II-621
Application factor - II-267, II-272, II-476, II-503, II-522, II-542, IV-807, IV-824, IV-834, IV-846, **IV-863**, IV-876, IV-888, VI-1012
Application factor and summand for operational behavior - VI-1006
Application factor f1 - VI-998
Arc-like end relief I and II - II-337
Arc-like profile modification - II-332
Areas of application for the FKM guideline - VIII-1031
Assembly - II-581
Assumptions made for the calculation - V-982
Automatic calculation of load factor q - IV-956
Automatic calculation of the dishing angle ψ - IV-957
Automatic change of reference profiles - II-616
Automatically - II-349
Automotive - VII-1016
Average surface pressure - III-767
Axial clearance - III-684
Axial offset - II-497
Axial spanning with nut - IV-808
Axial/transverse module - II-514
Axis alignment - II-570, II-596
Axis alignment if load spectrum is applied - II-442

B

Background - VIII-1033
Ball/pin diameter shaft/hub - IV-871
Base size - III-652
Basic data - II-250, II-393, II-452, II-494, II-514, II-534, III-673, IV-858, IV-954
  Materials - I-139
Basic data inputs - IV-907
Basic installation - I-40, I-42, I-242
Basic material Glued and Soldered joints - I-131

Basic materials - IV-948
Beam profiles - I-139
Bearing - III-645, III-647, III-667
Bearing application factor - II-477, II-505
Bearing calculation General - III-678, III-724
Bearing calculation with inner geometry - III-734, III-754
Bearing data - III-754
Bearing data tab - III-752
Bearing force curve and direction of the bearing forces - II-600
Bearing manufacturer - III-686
Bearing power loss - II-524
Bearing ring deformations - III-755
Bearing width - III-777
Bearings coefficient - V-966
Bearings with radial and/or axial force - III-747
Belt length - VI-999
Belt length and number of teeth on belt - VI-1007
Belts and chain drives - VI-995
Bending critical speed - III-695
Bending stress values - V-981
Bevel and Hypoid gears - II-448
Bevel gear - generating a 3D model - I-182
Bevel gear factor at flank and root - II-478
Bevel gears
  – Determine permitted overloads - II-628
Bevel gears with cyclo-palloid® gear teeth - II-483
Bevel gears with Palloid toothing - II-485
Bibliography - X-1106
Bibliography and Index - X-1105
Bolt data - IV-917, IV-921
Bolted joint under axial and shearing force - IV-908
Bolted joint under axial load - IV-909
Bolts - IV-904
  Bore - I-136
  Coefficients of friction classes - I-137
  Nuts - I-138
  Nuts strength class - I-137
  Strength class - I-136
  Thread type - I-137
  Tightening factor - I-136
  Type - I-138
  Washer - I-138
Bolts and pins - IV-897
Bolts/pins - I-134
Boundary safety coefficient - IV-944
Bracket connection - IV-951
Buckling - III-674, III-696

C

Calculate flank safety with 0.85*b (ISO 10300) - II-476
Calculate form diameter from tooth form - II-436, II-613
Calculate lubrication factor with oil temperature - II-438
Calculate moment of inertia from tooth form - II-438
Calculate number - V-988
Calculate scuffing - II-289
Calculate the internal temperature and the flash temperature - II-437
Calculate torsion moment of resistance - II-443
Calculating a pinion type cutter - II-360
Calculating and generating a report - I-89, I-99
Calculating axial forces on bearings in face-to-face or back-to-back arrangements - III-748
Calculating cylindrical gears manufactured using tools specified in DIN 3972 - II-614
Calculating force on bearings with a contact angle - III-692
Calculating Shafts - III-672, III-688
Calculating the displacement volume of gear pumps - II-438
Calculating the reference profile - II-360
Calculating the thermal nominal speed - III-736
Calculating the thermally permissible operating speed limit - III-738
Calculation - III-782, IV-806, VII-1021, VIII-1056
Calculation elements - IX-1084
Calculation in accordance with AGMA 421.06 (High Speed Gears) - II-630
Calculation in accordance with Klingelnberg, Gleason and Oerlikon - II-450, II-469
Calculation in accordance with SKF Catalog 1994 - III-742
Calculation in accordance with SKF Catalog 2004 - III-739
Calculation method - II-266, II-330, II-470, II-501, III-698
Calculation method for friction - III-686
Calculation methods - III-764
Calculation of contact analysis taking the shafts into account - II-298, II-380
Calculation of flank safety - II-432
Calculation of KHβ with manufacturing errors - II-308
Calculation of size coefficients for small gears - II-440
Calculation of spline connections as described in DIN 5480 with diameter centering - IV-856
Calculation of the wear coefficient kw for steel - II-264, II-605
Calculation of volume specific heat - III-784
Calculation reports - I-99
Calculation using Methods B or C (DIN 3990, 3991) - II-622
Calculation using your own Woehler line - II-436
Calculation variables - I-107
Calculation with enhanced formulae (differs from standard) - II-530
Calculation with experimental data - III-707
Calculation with normal module instead of axial module - II-514, II-529
Calculation with operating center distance and profile shift according to manufacture - II-437
Calculations - II-314, II-315, II-316, II-317, II-435, II-489, II-529
Calculations - IV-816
Campbell diagram - III-722

Center distance - II-251, II-253, II-515, II-535, VI-998, VI-1006
Center distance - VI-1013
Center distance tolerances - II-325
Center distance tolerances - I-130
Chain drives - VI-1010
Chain profiles ISO606 - I-132
Chain type - VI-1011
Change the output of angles in reports - I-240
Changes of the parameters for generation - I-192
Changing base settings in the interface - I-220, I-228
Characteristic number - II-487
Check changes in safeties if the center distance changes - II-632
Check if mounting of planets is possible - II-433
Checking the contact pattern - II-549, II-576
Checking the meshing - II-557
Circular pitched teeth - II-358
Clamped connections - IV-815
Clamped parts inputs - IV-924
Classic method - IX-1072
Classification of bearings - III-725
Coefficient for minimum tip clearance - II-430
Coefficient for minimum tooth thickness at tip - II-430
Coefficient of friction for hypoid gears - II-489
Coefficient of thermal expansion for housing - II-399
Coefficients of friction - IV-796, IV-805, IV-931
COM Interface - I-165
Comment field - I-78
Comments - I-106
Comparing types - III-730
Comparison of a FEM calculation with spiral-toothed gear wheel calculation - II-631
Compression springs - V-964
Compression springs standard - I-130
Condition query IF ELSE END - I-87, I-110
Conditions for using shaft calculation files - II-382
Conditions I - II-410
Conditions I/II - II-424
Conditions II - II-408, II-412
Configuration - II-312
Configuration - VI-1012
Configuration tool - I-59
Configuring Tensioning pulleys - VI-1005
Configuring Tensioning Pulleys - VI-998
Conical interference fit - IV-804
Conicity - IV-811
Connecting roller bearing - III-671
Connection elements - III-670, IX-1087
Connection, general - III-671
Connections - IV-791
Consider gyroscopic effect - III-679
Consider weight - III-679

Constraints data - IV-929
Constraints on various bearings - III-668
Constructed involute - II-318
Contact analysis - II-596, II-614
Contact Analysis - II-346, II-371
Contact analysis of a planet system - II-385
Contact analysis/Face load factor - II-371, II-442
Contact line (face gear) - II-592
Contact lines on tooth flank - II-598
Contact spring stiffness - II-488
Context menu - I-75, I-78, II-348
Convert external pressure with multiple interference fit - IV-799
Converting or inputting Gleason toothing data - II-455
Coupling - III-665
Creating a new bolt type - IV-960, IV-962
Creating and modifying tables - IX-1075
Creating Models in KISSsys - IX-1071
Creating, opening and closing projects - I-92
Cross section types - III-659, III-712, III-787
Cross sections - III-645, III-647, III-672, III-710, III-718
Crossed helical gears and precision mechanics worms - II-528, II-532
Crowning - II-339, II-345
Custom roller profile - III-754
Cutter radius - II-469
Cutter/Tool
  Hobbing cutter - II-313, II-349
  Pinion type cutter - II-315
Cutting teeth on an existing shaft - I-220, I-225
Cycloid - II-358
Cylindrical gear pairs - II-300
Cylindrical gears - II-248, II-473
Cylindrical interference fit - IV-792

D

Database Tool and External Tables - I-114, II-269, II-317, II-537, III-698, IV-800, IV-812, IV-818, IV-829, IV-840, IV-852, IV-858, IV-868, IV-881, IV-894, IV-900, IV-945, IV-960, V-967, V-974, V-982, V-988, VIII-1043, VIII-1062
Deep toothing or cylindrical gears with a high transverse contact ratio - II-434, II-612
Default values for addendum coefficients - II-480
Default values for tip base clearance - II-480
Define details of geometry - II-536
Define details of strength - **IV-863**, IV-865
Define grinding wheel for gear n dialog window - II-327
Define load spectrum - II-272, II-287, II-474, II-541, II-542, II-566, III-704, VIII-1042
Define misalignment for individual elements - II-388
Defining 2D graphics - IX-1102
Defining input and output - I-159
Defining Shafts - III-642

Defining sub elements - III-647, III-656
Defining the scoring load level (oil specification) - II-624
Defining your own default files - I-49, I-70, I-92, I-160
Definition coefficients of sliding friction and velocities - VII-1025
Definition in [COCREATE] - I-57
Definition of spring forces - VII-1024
Definitions and dimensions of standard cutters for palloid toothing - II-486
Definitions in [CADEXPORT] - I-55
Definitions in [CATIA] - I-56
Definitions in [GRAPHICS] - I-54
Definitions in [HICAD] - I-58
Definitions in [INTERFACES] - I-55
Definitions in [INVENTOR] - I-56
Definitions in [LICENSE] - I-54, I-68
Definitions in [PATH] - I-40, I-44, I-46, I-49, I-50, I-51, I-67, I-95, I-99, I-104
Definitions in [PROENGINEER] - I-57, I-219
Definitions in [REPORT] - I-53, I-101
Definitions in [SETUP] - I-42, I-43, I-44, I-47, I-48, I-52
Definitions in [SOLIDEDGE] - I-56, I-192
Definitions in [SOLIDWORKS] - I-56, I-176
Definitions in [THINK3] - I-58
Deflection and Bearing Forces, Distribution and Force of Torque - III-690
Deleting a database entry - I-120
Description of database tables - I-130
Description of the public interface - I-155
Determine the equivalent torque (for load spectra) - II-632
Diagram view - IX-1069
Dialog elements for the variable dialog - IX-1094
Difference between cylindrical gear calculation following ISO 6336 or DIN 3990 - II-621
Differences between different gear calculation programs - II-621
Dimension of the integral worm shaft - II-525
Dimensioning - II-509
DIN 3967 - II-323
DIN 5480 - IV-870
DIN 58405 - II-323
DIN 743 (2012) - III-701
Directory structure - I-46
Disc spring standard - I-138
Disc springs - V-985
Disconnect license from the network - I-68
Discretized model - II-376
Display critical bearing - III-685
Displaying elements in 3D graphics - IX-1088
Displaying the graphic - IX-1104
Dissipated Heat Flows - III-736
Distances for eccentric clamping/load - IV-927
Do not cancel if geometry errors occur - II-325, II-429
Docking window - I-70, I-72, I-75
Documentation point - III-672

Downloading a license file - I-41, I-42, I-43, I-44
Drawing - II-581
Drawing data - I-70, I-100, II-563
Drawing data - V-969
Drawing data - V-977
Drawing data - V-983
Drawing number - III-650
Dynamic factor - II-291, II-478
Dynamic load capacity - III-733

E

Eccentric crowning - II-340
Eccentric profile crowning - II-335
Editing the control file - II-581
Effect of partial load - II-384
Effect of profile modifications - II-618
Effect of torsion on the body of the gear - II-383
Effective belt width - VI-1007
Effective number of V-belts - VI-999
Effective/Actual - IV-870
Efficient interfaces - I-156
Eigenfrequency - III-674, III-694
Element Assistant - IX-1073
Element overview - III-650
Elements editor - III-645, III-649, III-650, III-787
Elements for shafts - IX-1086
Elements list - III-645, III-648
Elements of the KISSsoft User Interface - I-69
Elements tree - III-645, III-647, III-650
Elliptical deformation - II-367
Elliptical root modification - II-357
Enhanced service life calculation in accordance with ISO 281 - III-757
Enter safeties - III-706
Entry curve as specified by Hirn - II-356
Equivalent stress for sizings - III-686
Estimate the strength of asymmetrical spur gear toothings - II-631
Evaluation - II-586
Example
  Interference fit calculation - I-124, I-158, I-162
Example application of a variable dialog - IX-1098
Example of a call from Excel - I-167
Explicitly reading (importing) and generating data - I-161
Export individual teeth - II-559
Export shaft geometry - III-648, III-662
Expressions in variables - IX-1081, IX-1090
Extended functionality for developers - IX-1080
Extended service life calculation in accordance with Supplement to DIN ISO 281 (2007) - III-744, III-754

Extending an existing bolt series - IV-960
External diameter and throat radius - II-519
External tables - I-114, I-122, I-130, I-131, I-132, I-134, I-136, I-137, I-138, I-139, I-140, I-142, I-143, I-152, I-153, II-260, II-322, II-325, II-500
Eyes screen - V-972

F

f0r and f1r coefficients - III-736
Face gear - 3D geometry - I-180
Face gears - II-490
Face load factor - II-288, II-296, II-505
Face width ratio - II-481
Facewidth - II-253, II-463, II-516, II-535, II-596
Factors - II-291, II-477, II-505
Failure probability - III-685, III-747
Fatigue Limits for New Materials - III-789
File interface - III-752
Fine Sizing - II-409
FKM-Richtlinie, Edition 2012 - III-702
Flange connection with torque and forces - IV-909
Flank breaking - II-369
Flank curvature radii - II-580
Flash temperature - II-587, II-602
FOR loop - I-107, I-111
Forces - III-645, III-662
Form factors - II-275, II-286, II-601
Formatting - I-99, I-104, I-105
Formula entry and angle input - I-87
Fp-Tolerance as specified in tables in DIN3962 - II-429
Free cross section - III-672, III-787
Frequency of load - III-704
Friction clutches - VII-1018
Friction coefficients f0 and f1 - III-737, III-742
Functionality of the software - VIII-1031
Functions - IX-1091
Functions tables - I-123

G

Gear pump - II-393, II-607
Gear teeth in the case of an existing basic solid - I-184, I-192
Gear teeth in the case of existing shaft data - I-200
Gear teeth when existing shaft data is present - I-206, I-210, II-323
Gear tool - II-575
Gear tooth forms - II-276, II-574
Gears - III-677, III-695

General - I-38, I-101, II-428, II-449, II-506, II-507, II-527, II-581, IV-856, VIII-1031, IX-1066
General bearing - III-667
General calculation procedure for KHbeta as specified in ISO 6336-1, Appendix E. - II-311
General entries - III-718
General settings - VIII-1044
Generate - II-553
Generate a cylindrical gear with a pinion type cutter - II-351
Generate a cylindrical gear with an imported hobbing cutter - II-351
Generate a cylindrical gear with an imported pinion type cutter - II-353
Generate a rack with a hobbing cutter - II-361
Generate a rack with imported hobbing cutter data - II-361
Generate a SA worm - II-363
Generate cylindrical gear with hobbing cutter - II-349
Generate rack with a pinion type cutter - II-361
Generate rack with imported pinion type cutter - II-362
Generate with the other gear in the pair - II-360
Generating 3D shafts - I-177, I-184, I-192, I-200, I-206, I-238
Generating a database entry - I-119
Generating a face gear with a pinion type cutter - II-360
Generation of 3D gears - I-175, I-184, I-189, I-192, I-198, I-200, I-204, I-206, I-210, I-218, I-230, I-234, I-236, I-238
Generation of 3D model - II-443
Geometry - II-552, VII-1017
Geometry data - I-164
Geometry details - II-259, II-467, II-499, II-518, IV-862
Geometry of chain sprockets - VI-1015
Geometry of clamped parts - IV-924
Geometry standards - IV-858
Geometry-Fine Sizing for 3 gears - II-420
Global settings - KISS.ini - I-50, I-51, I-67, I-68, I-99, I-104
Glued and soldered joints - IV-946
Graphics - II-419, II-425, III-759, IV-884, VII-1026
Graphics list - II-610
Graphics menu - II-569
Graphics window - I-70, I-75
Grinding notch - II-281

H

Hänchen & Decker - III-698, III-709
Hand of gear for gear teeth - II-250
Handling bending and torsion using the results for the shaft - II-385
Hardening depth - II-588
Hardening depth, known by its abbreviation - II-271, II-284, II-547
Hardness conversion - VIII-1052
Head forms - V-992
Header and footer - I-101

Heat development - II-601
Heat transfer coefficient - III-773
Heat transfer surface - III-772
Height of face gear - II-500
Helix angle - II-460
Helix angle at reference circle - II-250, II-251, II-496
Helix angle modification - II-338, II-339, II-341
Helix angle reference circle gear 1 - II-534
Helptext viewer - I-74, I-83
Hertzian pressure - VIII-1049
Housing material - III-679
How can I test the software? - I-242
How to use KISSsoft - II-557
Hydrodynamic plain radial bearing - III-763
Hydrodynamic plain thrust bearing - III-779
Hypoid gears with cyclo-palloid gear teeth - II-483

I

Implementation in KISSsoft - VIII-1037
Import and export data with the database tool - I-121
Important service functions - IX-1093
Importing existing KISSsoft calculations - IX-1086
Importing the shaft geometry - III-648, III-660
Improve tooth form - II-558
Impurity - III-679
Individual names for elements - IX-1078
Influence factors - IV-899
Initial parameters - I-67
Inner contour - III-645, III-647, III-662
Input data - II-552
Input different load cycles for bending and torsion (for finite life calculations) - III-684
Input elements - I-87
Input file - I-160
Input format for data in imported files - II-554
Input materials for gear calculations in the database - I-241
Input normal diametral pitch instead of normal module - II-429
Input of number of teeth with decimal places - II-252, II-429
Input quality - II-428
Input window - III-645, III-711
Inputting the stress values on the proof point and on the neighboring point - VIII-1038
Inputting Tolerances - IV-795
Inside diameter - II-500, II-537
Inspecting V-belts - VI-1001
Installation on the server - I-43
Installing KISSsoft - I-39
Insufficient scuffing safety - II-623
Integrating the KISSsoft Add-in - I-201, I-207, I-221, I-237
Integrating the KISSsoft Add-in (menu items in CAD) - I-186, I-194

Interactions with variable dialogs - IX-1100
Interfaces between calculation programs and CAD - Overview - I-156
Internal teeth - differences in the reference profile if you select different configurations - II-617
Intersecting notch effects - III-718, III-787
Introduction - II-247
ISO 1328 - II-323
ISO 6336 - II-440
Iterative calculation of load distribution - III-684

J

Joint - III-671

K

Key - IV-819
Key standard - I-134
Kinematics - II-600
KISSsoft Calculation Modules - I-70, I-85
KISSsys - IX-1064
  Calculation Systems - IX-1065

L

Language settings - I-47, I-70
Lastkollektiv mit negativen Elementen - III-676
Lead angle at reference circle - II-514
Leg springs - V-979
Length factor - IV-837, IV-849, IV-891
Licensing - I-40, I-42
Licensing the KISSsoft system. - I-44
Lifetime factors - II-286
Lifetime factors as defined in ISO 6336 - II-274, II-475, II-544
Limit dimensions - V-989
Limited cross section - III-672
Limiting the number of teeth - II-411
Limiting the root diameter - II-411
Limiting the tip diameter - II-410
Limiting values - V-993
Limiting values in the calculation - III-785
Linear drive train - VIII-1053
Linear end relief I and II - II-337, II-338, II-345
Linear profile modification - II-355
Linear tip and root relief - II-331, II-336
Linear tip and root relief with transition radii - II-334
Linking the individual slices - II-379
List of key words used - I-126, I-128

List tables - I-126
Load - II-266, II-470, II-501, II-521, II-538, III-757
Load application - IV-927
Load capacity of roller bearings - III-733
Load case - III-707
Load cases - VIII-1039
Load distribution - III-759
Load distribution coefficient - II-288, II-292
Load factor - IV-826
Load factor for endurance calculation - III-710
Load factor for static analysis - III-709
Load spectra - III-676
Load spectra - I-132
Load spectra with negative elements - II-289
Load spectrum with changing torque - II-295, II-625
Load tab - III-757
Lubricant temperature - III-676
Lubricants - I-134
Lubrication - II-265, III-679
Lubrication arrangement - III-768
Lubrication coefficient - II-279

M

Machining allowance for cylindrical gear - I-130
Magnetic tension - III-666
Main input area - I-82
Main screen - IV-821, IV-822, VIII-1037
Main settings - II-381
Maintain root circle when changing profile shift - II-430
Managing database entries - I-116, I-119
Manual and Search - I-74
Manufacture - II-469
Manufacturing a gear - II-575
Manufacturing drawing - II-581
Manufacturing process - II-464, II-468
Manufacturing process for bevel and hypoid gears - I-131
Manufacturing tolerances - II-564
Manufacturing type - V-972
Mass - III-666
Master gear - II-400
Material - III-652
Material and lubrication - II-260, II-500, II-519, II-521, II-537
Material Disc spring calculation - I-141
Material Interference fit - I-141
Material of bolts - I-141
Material of enveloping worm wheels - I-140
Material of gears - I-143
Material of plain bearings - I-140

Material of shaft-hub connection - I-142
Material pairing factor (strengthening an unhardened gear) - II-624
Material properties - III-653
Material Shaft calculation - I-142
Material Spring calculation - I-140
Material Welded joints - I-141
Materials - I-241, II-260, IV-800, IV-812, IV-829, IV-840, IV-852, IV-868, IV-881, IV-894, IV-900, IV-945, VIII-1043, VIII-1062
Materials - I-139
Materials - IV-818
Materials - V-966
Materials - V-974
Materials - V-982
Materials - V-987
Maximal no of solutions - II-411
Maximum deflection for sizings - III-687
Maximum service life coefficient - III-685
Maximum Speeds - III-743
Measurement grid - II-421
Menus, context menus and the Tool bar - I-70, I-72, I-73, I-92, I-93, I-96
Menus, context menus and the Tool Bar - IX-1079
Meshing - II-549, II-575
Message output - IX-1070
Messages - I-90
Method Crown Gear (DIN 3990) - II-502
Method ISO 6336-B/Literature - II-501, II-502
Methods used for strength calculation - II-448, II-459, II-470, II-501, II-538
Minimum distance between 2 planets - II-433
Minimum safeties - II-487
Modification for mold making - II-365
Modification for pinion type cutter - II-366
Modification for wire erosion - II-366
Modification of S-N curve (Woehler line) in the range of endurance limit - II-284
Modifications - I-132, II-274, II-282, II-317, II-320, II-326, II-342, II-357, II-506, II-543
Modified rating life according ISO 281 - III-678
Modifying the selected 3D model - I-219, I-224
Modifying the teeth on an existing shaft - I-227
Module ratio - II-463, II-482
Module specific inputs - III-765
Module specific settings - III-681, IV-958
Moment of friction - III-686, III-739
Multi-bolted joint with any bolt position - IV-911
Multi-Spline standard - I-139

N

Necessary entries in the input window - II-410
Network version with dongle (protection key) - I-43

Network version with the license code - I-44
Node density - III-683
Non circular gears - II-551
Non-identical (mirrored symmetry) tooth flanks - II-616
Non-linear shaft - III-681
Normal force curve - II-598
Normal force distribution - II-598
Normal module - II-250, II-494, II-534, IV-859
Normal module (middle) - II-457
Normal module ranges for Klingelnberg machines (cyclo-palloid) - II-484
Notch effects on hollow shafts - III-788
Notch factors - III-704
Notches on the inner contour - III-788
Notches on the outer contour - III-788
Note about - II-388
Notes - II-549
Notes about profile modification - II-346
Notes on calculations in accordance with the Klingelnberg standard - II-483
Notes on face gear calculation - II-509
Number of blade groups, tool - II-469
Number of buckling cases - III-674
Number of eigenfrequencies - III-674
Number of links - VI-1014
Number of load cycles - II-280, II-475, II-545, VIII-1039
Number of load peaks - IV-867
Number of radial sealing rings, worm shaft - II-524
Number of strands - VI-1011
Number of teeth - II-252, II-462, II-515, IV-860
Number of teeth with common multiples - II-619

O

Occurring flank pressure - IV-836, IV-848, IV-878, IV-890
Offset (Center dist.) - II-462
Oil level and Lubrication type - III-741, III-750
Oil temperatures - III-774
Oil viscosity, depending on temperature - II-590
Only geometry calculation - II-501
Open interfaces concept in KISSsoft - I-157
Opening the calculation file - I-215
Opening the calculation file for the created gear - I-190, I-199, I-205
Operating backlash - II-396
Operating data - IV-907, VII-1018
Operations - II-348, II-349
Optimal tip relief - II-284, II-547
Outer contour - III-645, III-647, III-655
Output file - I-160
Overview of the available CAD interfaces and their functionality - I-174, I-179

Overview of the bevel gear manufacturing process and the terminology used in it - II-449
Own data for Woehler line (S-N curve) - III-654
Own Input - II-324, III-671
Own inputs - IV-827

P

Page layout - I-101
Pairing an external gear to an inside gear that has a slightly different number of teeth - II-613
Parasolid Export of 3D Shafts - I-180
Part safety coefficient - IV-943
Permissible decrease in quality - II-522
Permissible mass decrease - II-524
Permissible maximum wear of tooth thickness - II-432
Permissible pressure - IV-828, IV-839, IV-851, IV-880, IV-893
Permissible static stress - III-733
Permissible thickness of lubrication film - III-778
Permissible tooth thickness decrease - II-524
Permitted values - IV-902
Pinion - Face gear with Z1 > Z2 - II-510
Plain bearing - III-725
Planetary stages - II-304
Planets - II-433
Plastic - II-431
Polygon - IV-874
Polygon effect - VI-1013
Polygon standard - I-134
Position - III-651
Position of shaft axis in space - III-663, III-673, III-675, III-679
Position of tensioning pulley (x/y) - VI-1000
Position of the tensioning pulley x/y - VI-1009
Possible Sizings/Suggestions - VI-1005
Power loss - II-601
Power, torque and speed - II-273, II-474, II-503, II-523, II-542
Power-on time - II-439, II-527
Preamble - I-159
Precision mechanics - II-612
Preliminary treatment and grinding allowance - II-318, II-319, II-327
Pressure angle at normal section - II-250, II-458, II-496, II-514, II-534
Pressure angle at normal section an - IV-859
Pressure angle driving/driven flank
  Hypoid gears - II-458
Pressure angle modification - II-336, II-341
Pressure curve - III-759
Pressure curve for each rolling body - III-762
Pretension - II-283
Procedure for toothing creation - I-184, I-192, I-200

Process for calculating thermally permitted operating speed (DIN 732-2) - III-737
Profile and tooth trace diagram - II-576
Profile crowning (barreling) - II-335
Profile modification - II-274, II-342, II-475, II-543
Profile modification optimization - II-424
Profile modifications - II-328, II-331
Profile shift coefficient - II-253, II-352, II-463, II-498, II-516, II-535, II-586, IV-860
Profile shift coefficient (on the pinion) - II-497
Programming in the Interpreter - IX-1090
Progressive profile modification - II-333, II-355
Project Management - I-49, I-70, I-73, I-91
Project properties - I-93, I-96
Properties - I-75, I-77, I-78, I-81, III-725
Properties dialog - IX-1080
Properties of the most important bearing types - III-728
Proposal for the hardening depth EHT - II-568
Protective layer thickness, Aluminum - III-706
Protective layer thickness, aluminum, chapter 4.3.4, Figure 4.3.4 - VIII-1040
Pure thrust bearing - III-670

Q

Quality - II-256, II-429, II-464, II-498, II-517, II-535, IV-861

R

Range tables - I-125
Ratio facewidth to center distance - II-435
Ratio facewidth to normal module - II-434
Ratio facewidth to reference circle, gear 1 - II-435
Reading (importing) a cylindrical gear - II-354
Reading (importing) a worm into the axial section - II-364
Reading (importing) the rack - II-363
Reduced stiffness on the side edges - II-378
Reference cone apexes on the outside/inside of the unworked part - II-468
Reference diameter gear 2 - II-458
Reference gearing - II-528
Reference profile - II-312, II-317, II-480, II-555
Reference profiles - I-130
Reference temperature - II-398, III-651, III-675
References - IX-1081, IX-1084
Registering the interface - I-230
Registering the server - I-165
Relationship of calculations with elements - IX-1085
Relative structure coefficient (scuffing) - II-279, II-345, II-475, II-544
Relative water absorption during swelling - II-398
Relaxation - V-968, V-976
Report - II-425, II-560

Report menu - II-562
Report settings - I-101
Report templates - I-76, I-99, I-104, I-161, I-240
Report Viewer - I-70, I-82, I-99
Required safeties - II-273, II-405, II-441, II-474, II-503, II-523, II-530, II-542, VIII-1047
Required safeties for cylindrical gears - II-612, II-622
Required service life - II-474, III-685
Required transverse contact ratio - II-434
Requirements placed on the third party program - I-160
Restore previous stages of the calculation - I-245
Restoring a database entry - I-120
Resulting shearing force - IV-864
Results - II-408, II-417, II-424, II-443
Results and Reports - I-97
Results from FEM calculation - IV-915
Results of a calculation - I-98
Retain tip circle when modifying profile shift - II-429
Retaining rings (self-locking rings, Seeger rings) - IV-953
Rights - I-50
Roller bearing - III-677, III-725, III-727, III-751
Roller bearing - I-144
Roller bearing basic data - I-144
Roller bearing Internal geometry - I-147, III-734
Roller bearing tolerance - I-152
Roller bearing Tolerance classes - I-152
Roller bearings - III-667
Roller Bearings (Inner Geometry) - III-734, III-751
Root diameter allowances - II-324
Root radius - II-358
Rough sizing - II-404, II-418, II-481
Rules - I-63
Running KISSsoft via an add-in - I-189, I-198, I-204, I-209

S

Safety against deformation/fracture - III-705
Safety against fatigue/deformation - III-708
Safety against micropitting - II-602
Safety factor curves - II-590
Safety factor for the calculation of the shear stress at EHT - II-439, II-568
Scope of a report - I-99, I-101, I-105
Scuffing and sliding velocity (face gear) - II-594
Seam length - IV-940
Selecting the type of roller bearing - III-728
Selection of hobbing cutters - I-131
Selection of pinion type cutters - I-138
Selection of the part form - VIII-1037
Sense of rotation - III-652, III-663, III-675

Server functionality - I-165
Service life - II-271, II-503, II-522, II-541, II-566, III-704, III-744
Service life calculation with load spectra - III-745
Service life of files - I-161
Setting Up KISSsoft - I-45
Settings - II-325, II-428, II-489, II-507, II-527, II-548, II-608, II-609, IV-801, IV-813, IV-830, IV-841, IV-853, IV-882, IV-895, IV-901, IV-934, IV-949, VII-1027, VIII-1044, VIII-1051
Settings - IV-817
Settings - V-975
Settings - VIII-1061
Shaft angle - II-461, II-499, II-528, II-537
Shaft connections - IV-952
Shaft editor - III-645, III-650
Shafts and Bearings - III-641
Shape of flank - II-518
Share factor - IV-838, IV-850, IV-892
Shear stress - V-993
Shear stress values - V-965, V-971
Show automatic dimensioning - III-686
Show coordinate system - III-686
Simplified view of the gears - I-190
Single contact stiffness - II-599
Single normal pitch deviation - II-488
Single user version with dongle (protection key) - I-42
Single user version with license code - I-43
Sizing - III-712
Sizing modifications - II-342
Sizing of torque - II-567
Sizing the bearing clearance - III-775
Sizings - II-318, II-405, II-406, II-434, II-497, II-508, III-783, IV-803, IV-814, IV-831, IV-842, IV-854, IV-883, IV-896, IV-903, IV-950, V-978, V-984, V-994, VI-1010, VIII-1061
Sizings - IV-817
Sizings - V-969
Small no. of pittings permissible - II-279, II-475
Smoothing the tooth form curvature to calculate Hertzian pressure in the contact analysis - II-376
Solders - I-133
Sommerfeld number - III-776
Special features in KISSsoft - IV-906
Special toothing - II-614
Specific sliding - II-586, II-601
Spectra - VIII-1042
Speed - III-651, III-652, III-674
Speed/number of teeth/transmission ratio - VI-1012
Spline (geometry and strength) - IV-843, IV-855
Spline (strength) - IV-843, IV-866
Spline shaft - IV-832
Spline Standard - I-132

Spring design - V-981
Springs - V-963
Standard - VI-1011
Standard and special tabs - I-86, I-89, I-99, II-250, II-266, II-291, II-349
Standard profiles - IV-833, IV-844, IV-875, IV-886
Standard radius at shoulder - III-682
Start and end block - I-102
Starting KISSsoft - I-66
State during heat treatment - III-652
Static calculation - II-540
Static calculation on shearing - II-540
Static strength - II-501
Stiff connection - III-671
Stiffness curve - II-599, III-761
Storage locations - I-95
Storage locations and descriptions - I-104
Storage strategies for calculations - IX-1085
Straight line flank - II-359
Strength - III-697
Strength calculation as defined in VDI 2736 - II-540
Strength calculation in acc. with Hirn - II-538
Strength calculation in acc. with Hoechst - II-539
Strength calculation in acc. with ISO 6336/Niemann - II-538, II-539
Strength calculation method - IV-863
Strength calculation methods - II-521
Strength calculation using mean position in tolerance field (of tooth form) - II-439
Strength calculation with several geometries on one gear - II-627
Strength details - II-273, II-475, II-523, II-543
Strength details (AGMA) - II-268, II-273, II-286
Strength parameters in accordance with DIN - III-707
Strength parameters in accordance with FKM - III-705
Strength parameters in accordance with Hänchen and Decker - III-704
Strength values - V-965, V-971, V-980, V-986, V-992
Strength verification with local stresses - VIII-1030
Stress - III-708
Stress curve - II-601
Stress curve (face gear) - II-593
Stress ratio - III-708
Stress ratio R - IV-867
Stress ratios - VIII-1040
Stress values - V-986
Stripping strength - IV-933, IV-935
Structure of KISSsys - IX-1066
Student version - I-42
Summary - II-443, II-565
Support of gearing - II-524
Surface factor - III-652
Surface factor KV, section 4.3.3, Table 4.3.7 - VIII-1042
Surface roughness - III-712, VIII-1043
Surface roughness at tooth root - II-487

Surface roughness of housing - III-686
Surface roughness of shafts and shaft-hub connections - I-134
Swing angle controlled tightening - IV-932
Synchronization - VII-1016
System Assistant - IX-1074
System of units - I-48, I-70
System settings - IX-1089

T

Table view - IX-1069, IX-1081
Tables - I-88
Take into account deformation due to shearing and shear correction coefficient - III-682
Take into account user specific additions - II-438
Take protuberance into account - II-439
Take shot peening data into account in calculating the strength of toothed gears - II-629
Taking double helical gearing into account in the shaft calculation - III-790
Taking the results into account in the report - III-654
Technical Explanations - IV-930
Technical notes (toothed belts) - VI-1003
Temperature - III-651, VIII-1040
Temperature duration - III-705, VIII-1040
Temperature of housing - III-651, III-676
Templates - IV-872
Temporary files - II-560
Tension springs - V-970
Tensioning pulley diameter - VI-999
Tensioning pulley tooth number - VI-1007
Tensioning pulleys - VI-1011
Test version - I-42, I-242
Text formatting features - I-106
The active working project - I-46, I-49, I-73, I-92, I-94, I-95
The definition of a beam chart (dg_b) - IX-1103
The definition of an XY-line graphic (dg_l) - IX-1103
The definition of the axis system (af) - IX-1102
The entire definition - IX-1103
The existing elements - IX-1083
The FKM Guideline
Rechnerischer Festigkeitsnachweis für Maschinenbauteile - VIII-1033
The influence of the face load factor KHß for tooth trace deviation fma is due to a manufacturing error. - II-624
The info window - I-73, I-87
The Messages window - I-73, I-90
The module tree - I-72
The project tree - I-49, I-70, I-73, I-91
The Results window - I-73, I-86, I-98
The Shaft element - III-650
The user interface - IX-1068, IX-1069, IX-1070

Theoretical contact stiffness - II-591
Theoretical involute/form grinding - II-358
Theory of Contact Analysis - II-373
Thermal expansion coefficients - III-766
Thermal reference speed - III-735
Thermally permissible service speed - III-735
Thickness factors from the shaft diameter - III-718
Thickness of lubrication film and specific oil film thickness - II-573
Tightening technique - IV-922
Tip alteration - II-320
Tip diameter allowances - II-324
Tolerance calculation - VIII-1029
Tolerance field - III-678
Tolerances - II-322, II-526, II-555, IV-869
Tolerances - V-967
Tolerances - V-975
Tolerances - V-983
Tolerances standard - I-139
Tool bar and context menu - I-75, I-76
Tooltips and status bar - I-71, I-84, I-88, I-89, I-98
Tooth contact stiffness - II-277, II-342, II-442
Tooth form - II-347, II-585, IV-873
Tooth system - II-585
Tooth thickness - II-447
Tooth thickness at tip - II-614
Tooth thickness modification factor - II-463, II-517
Tooth thickness tolerance - II-322, IV-869
Tooth thickness tolerances - I-152
Tooth trace modification - II-299, II-345, III-719
Tooth trace modifications - II-328, II-336
Toothed belt standard - I-153, VI-1004
Toothed belts - VI-1002
Toothing - I-80, II-246
Topological modification - II-341
Torque curve - II-599
Torque curve/Frequency of change of load direction - IV-835, IV-847, IV-877, IV-889
Torsion critical speed - III-695
Torsion-bar springs - V-990
Transmission accuracy level number - II-287
Transmission error - II-596
Transmission error acceleration - II-597
Transmission error amplitude - II-598
Transverse coefficient - II-291
Tree view - IX-1068
Triangular end relief I and II - II-340
Twist - II-341
Type - II-450, II-452
Type of bolted joint - IV-920
Type of calculation - III-702
Type of load spectrum - II-288

Type of loading/Frequency of change of load direction - IV-866
Type of modification - II-328, II-506
Type of oil lubrication - III-686

U

Undercut or insufficient effective involute - II-613
Underlying principles of calculation - II-329, II-449, II-491, II-512, II-533, IV-856
Unit switch - I-88
Use alternative algorithms for the tooth form calculation - II-430
Used files - I-160
Usefulness of the service life calculation - VIII-1033
User-defined settings - I-59

V

Value input fields - I-48, I-73, I-87, I-88
Values on the X-axis of diagrams - II-430
Variable dialogs - IX-1093
Variable external diameter of the hub - IV-810
Variable hub external diameter - IV-798
Variables - IX-1081, IX-1083
Variants - IX-1081, IX-1084
Variations in rolling as defined in DIN 58405 - II-615
Various - VIII-1028
Varying qualities - II-429
V-belt - VI-996
V-belt Standard - I-131
V-belt standards - VI-997
V-belts data - VI-997
VDI 2737
  Calculation of gear rim - II-260, II-439
Viewer with neutral format interface - I-179
Viewing database entries - I-116, I-119

W

Warning
Washers - IV-922
Ways in which KISSsys can be used - IX-1066
Wear - II-604
Wear iteration - II-379
Weld seam boundary stress - IV-942
Welded joints - IV-937, IV-938
Welded seam equivalent stress - IV-941
What licenses are available? - I-243
Width and circumferential factor - IV-868
Woehler line - VIII-1039

Woehler line for material - II-589
Woodruff Key - IV-885
Woodruff Key standard - I-134
Worm wheel - generating a 3D model - I-183
Worms with enveloping worm wheels - II-511

Z

Z-Y coefficients and the technology factor - II-309