

KISSsoft 03/2018 – Tutorial 5

Shaft analysis

Contents

1	Starting KISSsoft.....	3
1.1	Starting the software	3
1.2	Selecting a calculation	3
2	Analyzing a Shaft	4
2.1	Task, opening the example calculation	4
2.2	Results	6
2.3	Deformation analysis.....	7
2.4	Eigenfrequencies	9
2.5	Strength analysis of shafts	10
2.5.1	Data about the calculation method	11
2.5.2	General data	11
2.5.3	Information about a material	12
2.5.4	Selecting cross-sections for analysis	14
2.5.5	Analysis and results	16
2.6	Reports.....	16
3	Further Calculations	17
3.1	Additional calculations.....	17
3.2	Combination with roller bearing analysis.....	17

1 Starting KISSsoft

1.1 Starting the software

You can call KISSsoft as soon as the software has been installed and activated. Usually you start the program by clicking «Start→Program Files→KISSsoft 03-2018→KISSsoft 03-2018». This opens the following KISSsoft user interface:

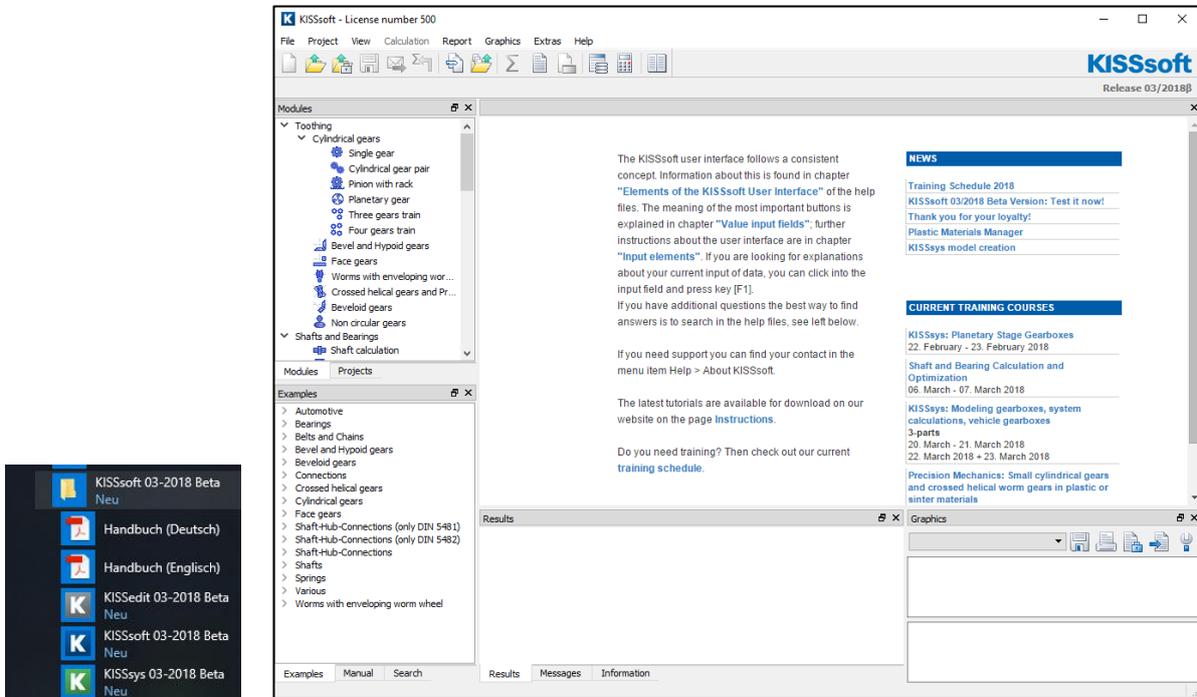


Figure 1. Starting KISSsoft, initial window

1.2 Selecting a calculation

In the Modules tree window, select the «**Modules**» tab to call the calculation for shafts:

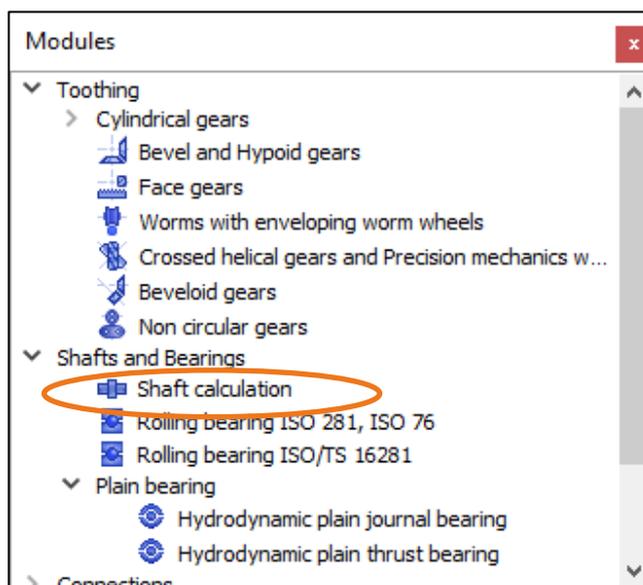


Figure 2. Selecting the «Shaft calculation» calculation module under Shafts and bearings

2 Analyzing a Shaft

2.1 Task, opening the example calculation

A shaft that has already been modeled, see Figure 3, is to be investigated mathematically. The following criteria are relevant:

- Shaft deformation
- Bending critical speed
- Static and fatigue strength

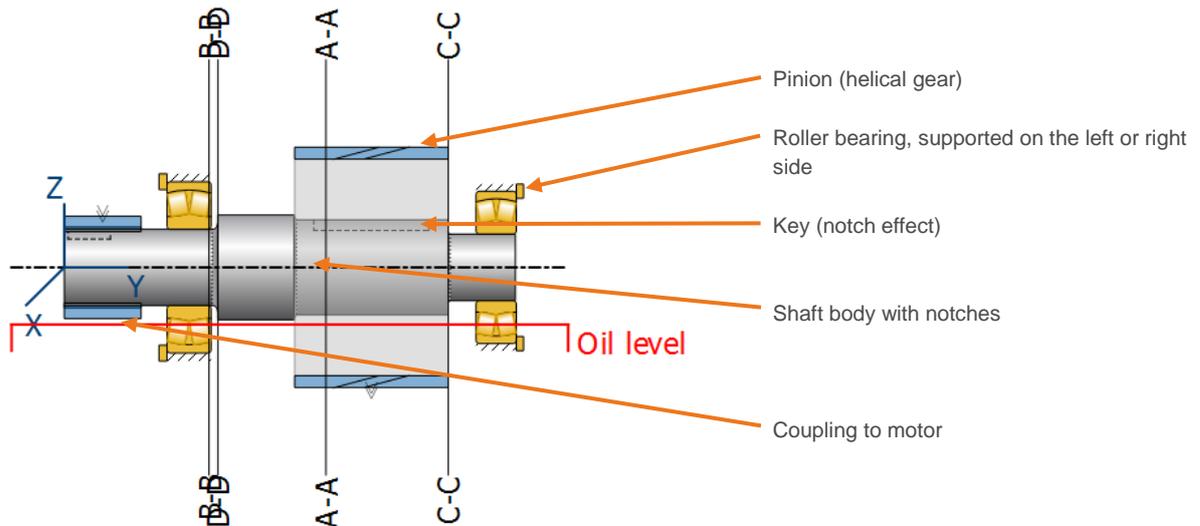


Figure 3. Analyzed pinion shaft

The shaft is driven by a motor attached to the coupling. The nominal power is 75W at a speed of 980 rpm. This power is taken from the system at the helical gear.

This shaft is already present in KISSsoft as an example file called **Shafts 1.W10**. You can either open this file directly in the help index window «**Examples**» tab by double-clicking on it with the left-hand mouse button or by clicking «**File**»→ «**Open**» and then selecting the file in the appropriate KISSsoft subdirectory ...\\example by clicking «**Open**».

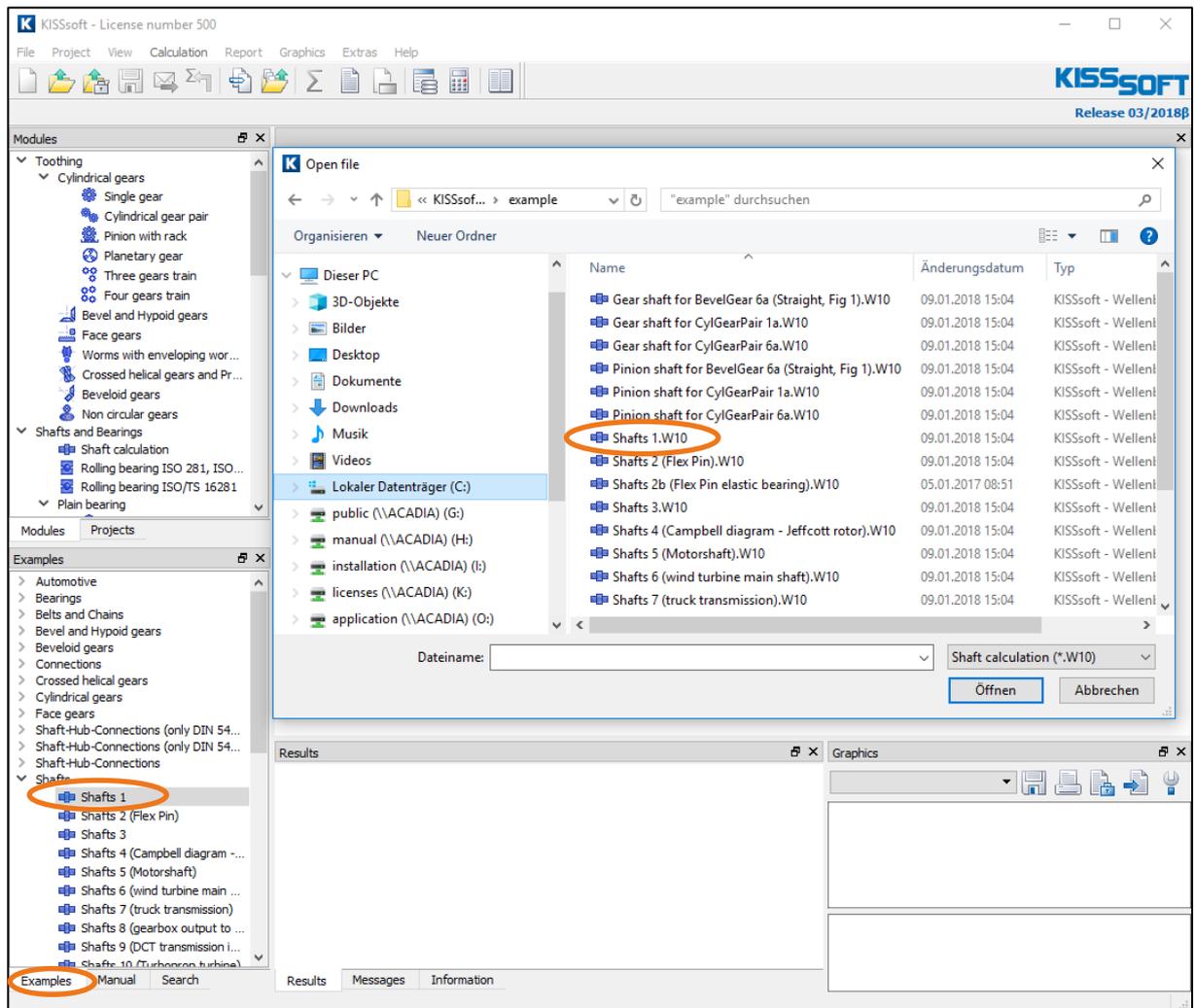


Figure 4. Opening the example analysis

Click the «**Shaft editor**» tab to view the shaft tab as shown in Figure 5.

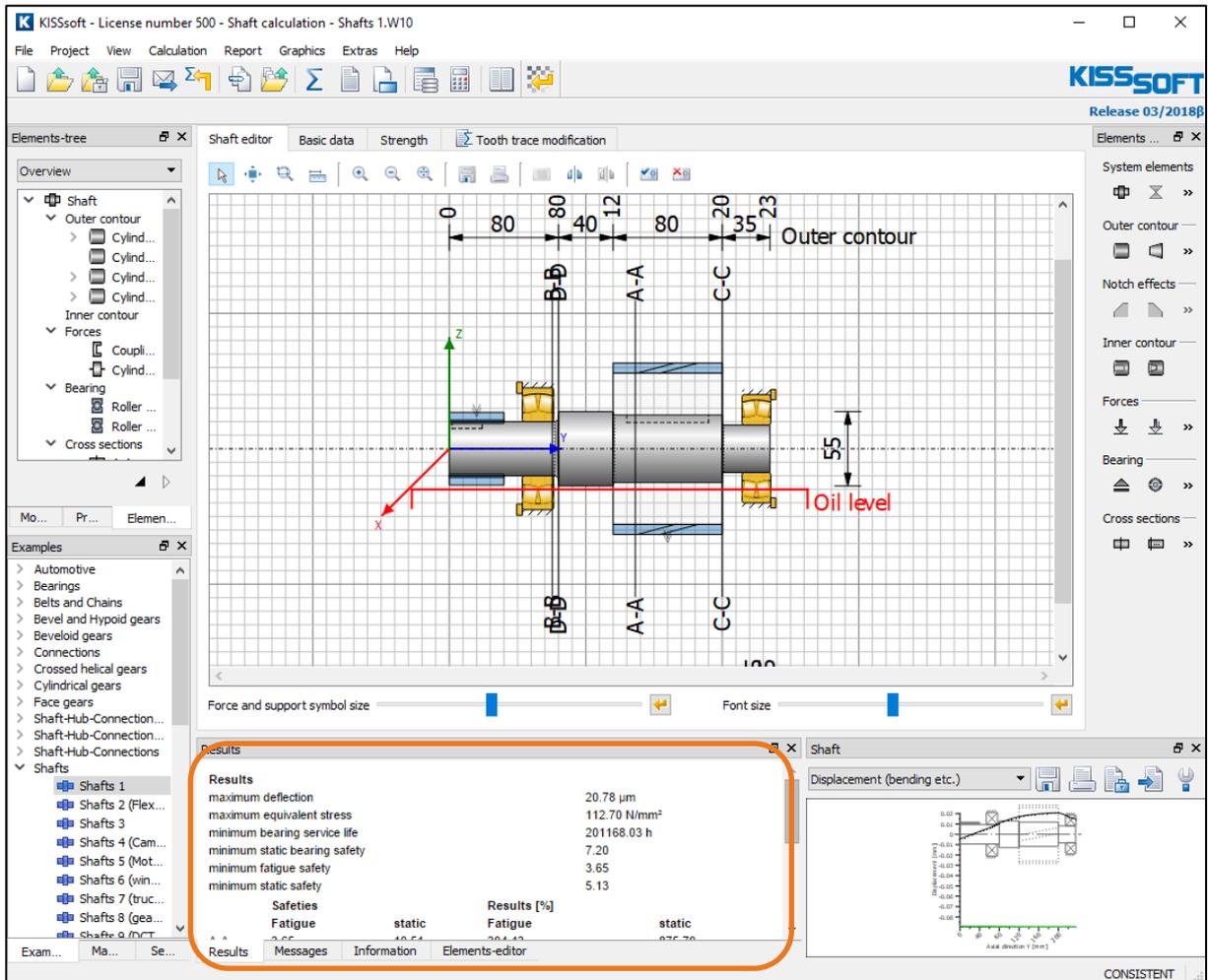


Figure 5. Opened shaft analysis

You will find details of how to generate a shaft model with the help of the graphical shaft data input interface in KISSsoft Tutorial 006.

When you open the shaft file, the system performs an initial calculation using the settings entered. Once you have finished defining the shaft, click  in the menu 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.

2.2 Results

All important results are listed in the Results tab. To open this window, click the  icon in the upper right-hand corner. You can then resize the window as necessary.

Results						
Results						
maximum deflection	20.78 μm					
maximum equivalent stress	112.70 N/mm^2					
minimum bearing service life	201168.03 h					
minimum static bearing safety	7.20					
minimum fatigue safety	3.65					
minimum static safety	5.13					
	Safeties		Results [%]			
	Fatigue	static	Fatigue	static		
A-A	3.65	10.51	304.43	875.70		
B-B	3.80	5.44	316.91	453.51		
C-C	4.01	13.21	334.53	1100.98		
D-D	4.10	5.13	341.42	427.46		
Bearing service life	S0	Lnh	Lnmh	Lnrh	Lnmrh	pmax_i
Roller bearing 1	7.20	6938 h	15701 h	37827 h	201168 h	1687 N/mm^2
Roller bearing 2	9.72	24420 h	104534 h	39560 h	283174 h	1611 N/mm^2
Bearing reaction force	Component		X	Y	Z	Rxz
Roller bearing 1	F		-3.088 kN	3.263 kN	4.977 kN	5.857 kN
	M		0.000 Nm	0.000 Nm	0.000 Nm	0.000 Nm
Roller bearing 2	F		-1.501 kN	0.000 kN	7.255 kN	7.409 kN
	M		0.000 Nm	0.000 Nm	0.000 Nm	0.000 Nm

Figure 6. Zoomed display of the results

2.3 Deformation analysis

In the «**Basic data**» tab is a group ,General', you will find a drop-down list "Gears". This is where you input how the gears modeled on the shaft are to be taken into account for the calculation:

- Ignore the mass and stiffness of the gears on the shaft
- Consider gears as masses only (the gear is mounted loosely on the shaft and, although it transfers its own weight to the shaft, along with the external loads, it does not stiffen the shaft)
- Consider gears as masses and as stiffness (the gear is rigidly attached to the shaft and forms a single unit with the shaft)
- Gears mounted by interference fit with stiffness according to ISO (like list item «consider gears as mass and as stiffness» but calculated with a smaller diameter)

The screenshot shows the 'Basic data' tab with the following settings:

- General:**
 - Load spectra: Don't consider load spectra
 - Gears: **Gears mounted by interference fit, with** (highlighted)
 - Position of shaft axis in space: horizontal
 - Angle: β 0.0000 °
 - Number of eigenfrequencies: i 0
 - Number of bucking cases: j 0
 - Speed: n 980.0000 1/min
 - Direction of rotation: clockwise
 - Consider weight
 - Consider gyroscopic effect
 - Consider deformation due to shearing (Timoshenko beam, not Euler-Bernoulli be)
- Rolling bearings:**
 - Rolling bearings: Rolling bearing service life from inner geo
 - Tolerance field: Mean value
 - Lubricant: Oil: ISO-VG 220
 - Impurity: Oil lubrication with filtration, ISO 4406 -/1
 - Modified rating life according ISO 281
 - Lubricant temperature T_a : 30.0000 °C
- Housing:**
 - Housing material: C45 (1), Through hardened steel, unallo
 - Temperature of housing T_c : 20.0000 °C

Figure 7. Method of considering the gears

After making your selection, call the «**Calculate**» function by clicking  in the menu bar or pressing «F5» to trigger the calculation. To display deflection as a graphic, either click the «**Graphics**» → «**Shaft**» → «**Displacement (bending etc.)**» menu or select the «**Displacement (bending etc.)**» option in the shaft docking window drop-down list.

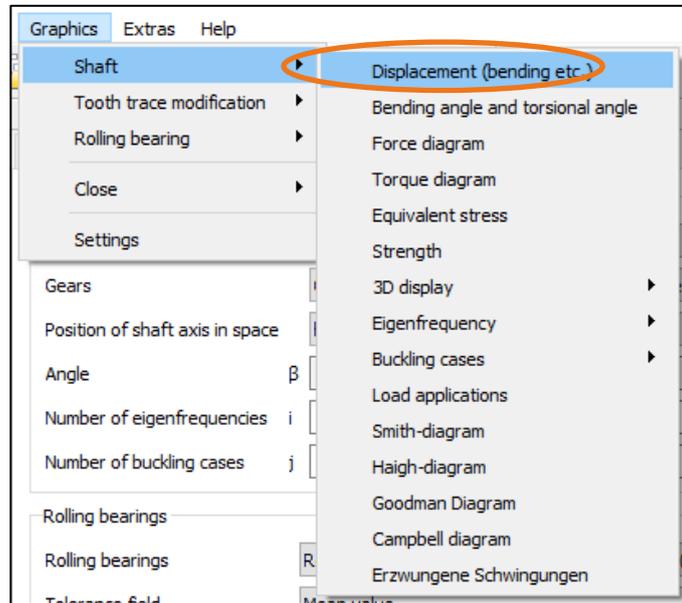


Figure 8. Options for displaying shaft analysis as a graphic

Note that a graphics window is already open in the lower right-hand corner of the user interface. Click «**Displacement (bending etc.)**» to display the bending curve.

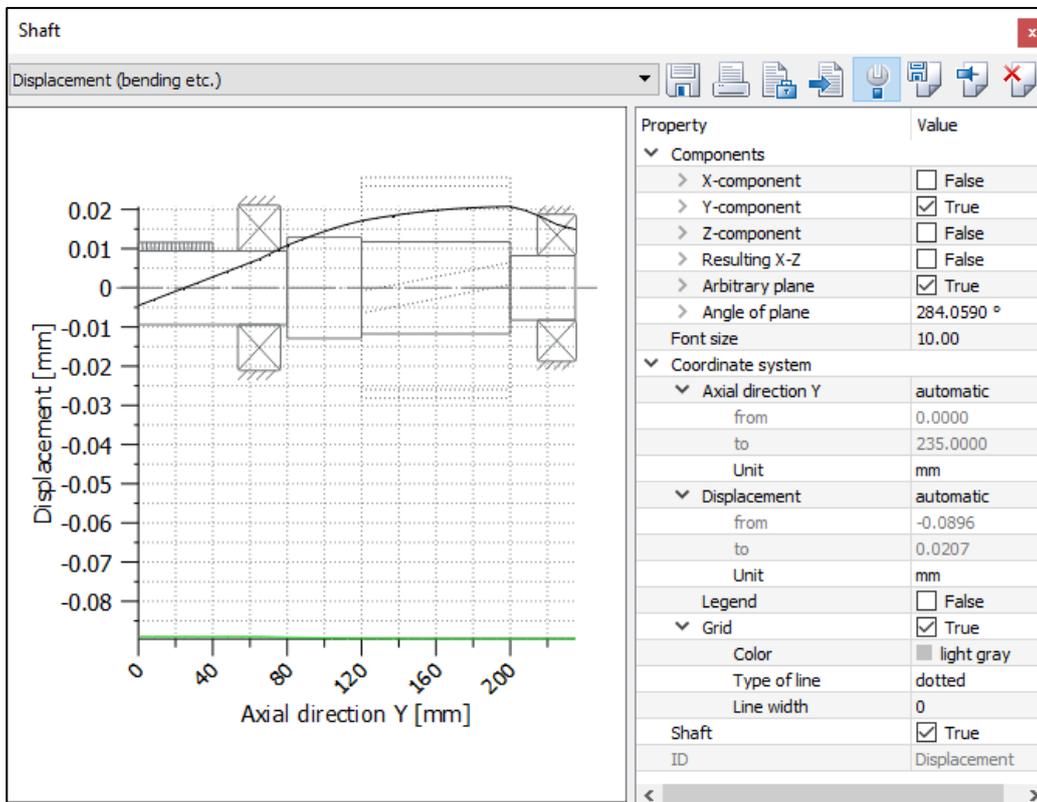


Figure 9. Display produced by selecting the Displacement option

The maximum displacement shown in the results window as the result of helical tothing and the associated moments on the x- and z- axes result in a tilting such that the maximum displacement at plane $\alpha = 284^\circ$ $ux' = 20.7 \mu\text{m}$. This is the largest value from all possible planes.

Alternatively, select «Report» → «Diagrams of bending» to view a list of calculated values.

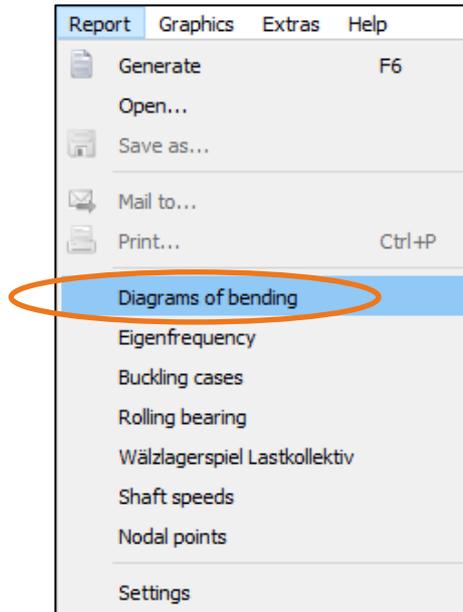


Figure 10. Calling the Diagrams of bending report

2.4 Eigenfrequencies

The first three critical frequencies are to be defined in this example. Once again, open the «Basic data» tab and select how the gears are to be considered (here only as masses, using conservative assumed values). Then

input the number of eigenfrequencies to be calculated (three) and click the «Calculate» icon  to trigger the calculation process. (For more information about the calculation, press «F1».)

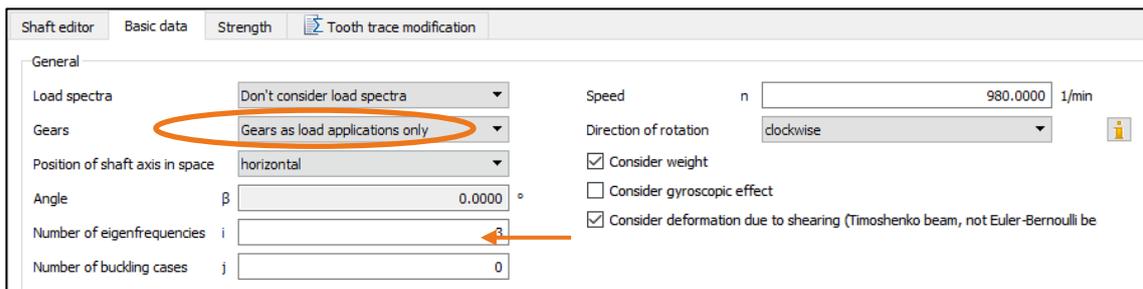


Figure 11. Inputting the number of eigenfrequencies

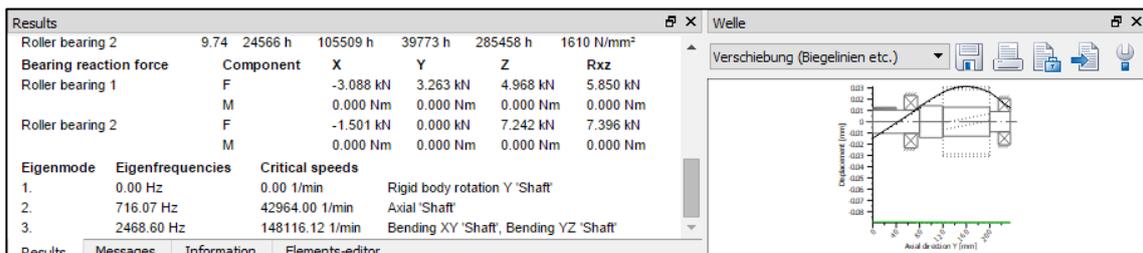


Figure 12. Result of calculating the number of rotations

The first 3 resulting eigenfrequencies along with their frequencies are displayed in the results window. The type of eigenfrequency involved here is also shown. The first eigenfrequency is a rigid body rotation at 0 Hz. Click the «**Property Browser**» to make different settings. You can select which eigenfrequency is to be displayed, and for which angle (torsion) or which plane (bending):

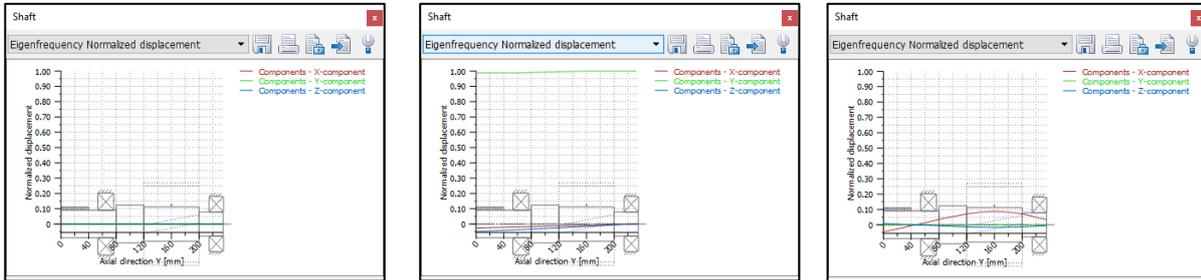


Figure 13. The three resulting eigenfrequencies

This graphics can also be shown in 3D.

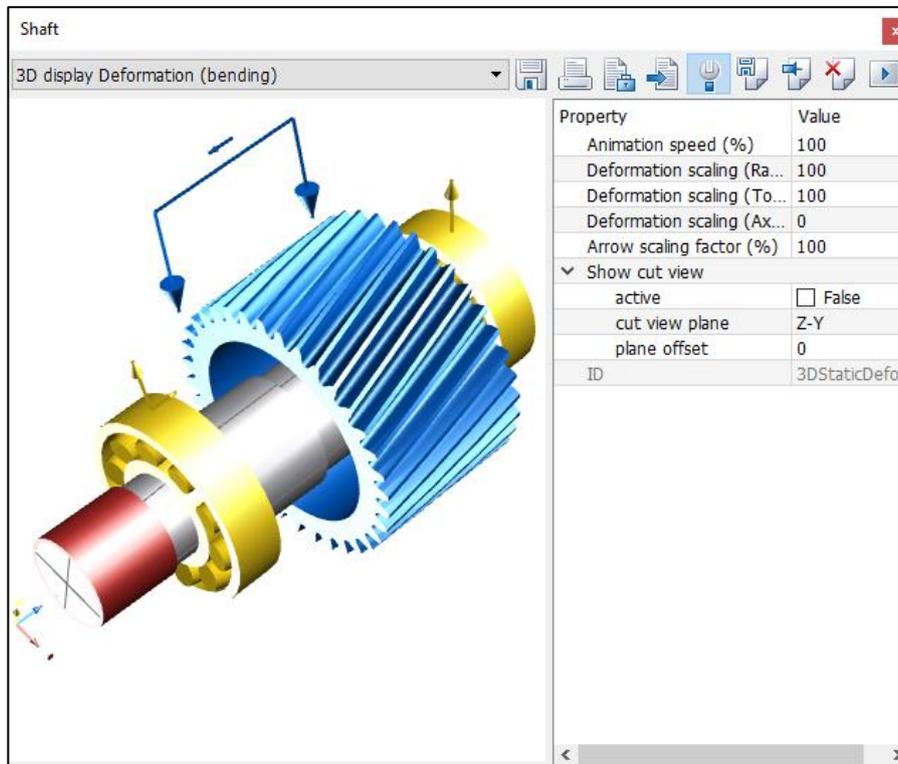


Figure 14. 3D display of deformation (bending)

2.5 Strength analysis of shafts

Open the «**Strength**» tab (see Figure 15) to determine the settings for strength analysis. KISSsoft currently provides four methods for strength analysis: according to Hänchen and Decker, DIN 743, AGMA 6101 and the FKM Guideline. This example uses DIN 743.

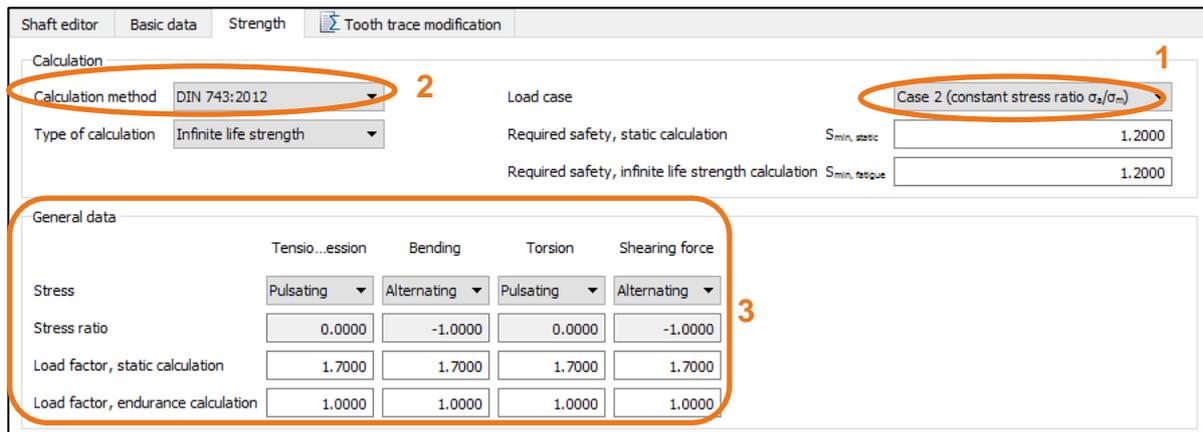


Figure 15. Buttons involved in strength calculation

- (1) Load case
- (2) Select the calculation method/type
- (3) General data

To determine which cross sections are to be analyzed, first select «**Cross sections**» by right-clicking this element in the «**Elements tree**». The software then automatically defines these cross-sections (depending on the geometry notches, interference fits etc.). Alternatively, you can input your own values here.

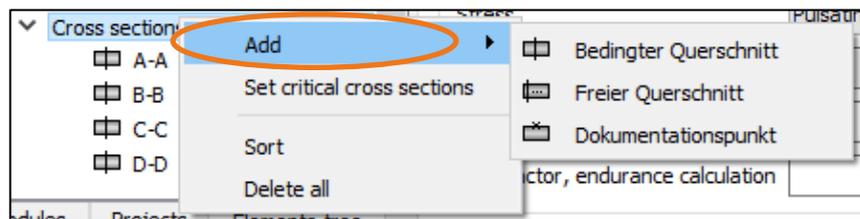


Figure 16. Calling the cross sections to be analyzed - automatic sizing

2.5.1 Data about the calculation method

You can use the default settings for this example.

2.5.2 General data

This is where you input the details of the load type, and the required proof (only static or proof of fatigue strength).

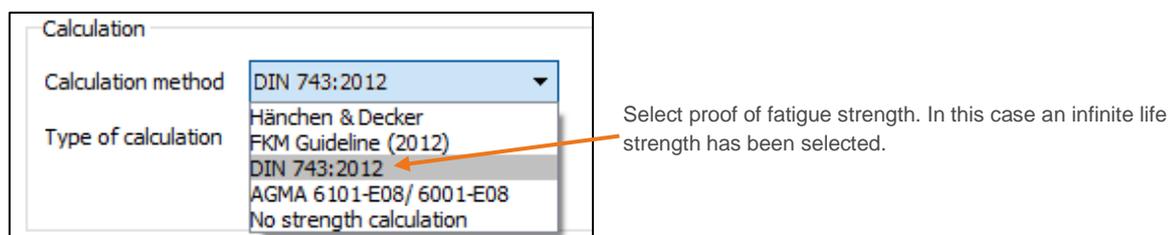


Figure 17. «Strength» tab: «Calculation» group - Details about the proof

Type of load: rotary bending, pulsating torsional moment (this places a pulsating tension-pressure load on the helix angle of the gear)	Load factors for static proof			
	Load factors for the endurance proof			
General data				
Stress	Bending Alternating	Torsion Pulsating	Tension/Compression Pulsating	Shearing force Alternating
Stress ratio	-1.0000	0.0000	0.0000	-1.0000
Load factor static calculation	1.7000	1.7000	1.7000	1.7000
Load factors endurance calculation	1.0000	1.0000	1.0000	1.0000

Figure 18. «Strength» tab: «General data» - details of the type of load

In the case of shafts that are to be loaded with different frequency for bending and torsion (for example, quickly rotating drive shafts with pinions that are stopped and started from time to time), you must first decide whether a constant load (here for torsion) or a pulsating load is to be assumed. To avoid errors, both calculations can be carried out, and the lower safety value can be taken as valid.

2.5.3 Information about a material

In the «Basic data» tab in the shaft calculation module (Figure 11) a housing material is defined. You define the material used for the shaft when you define the shaft itself. You can also change this data by selecting a shaft in the Elements editor.

The screenshot shows the 'Elements-editor' window with the following fields:

- Label: Shaft
- Drawing number: W-007
- Position in global system: Y, 0.0000 mm
- Operating temperature: T, 20.0000 °C
- Ambient density: ρ , 1.2000 kg/m³
- Speed: n, 980.0000 1/min
- Direction of rotation: clockwise
- Material: 42 CrMo 4 (1), Through hardened steel, alloyed, through hardened
- Raw diameter: d, 60.0000 mm
- Surface factor (DIN, FKM): None
- State during heat treatment: Pre-turned to actual diameter
- Material properties (DIN, FKM): with reference diameter
- Temperature factor (AGMA) k_t : 1.0000
- Miscellaneous effects factor (AGMA) k_e : 1.0000
- Import results into the report:

Figure 19. Shaft material

However, you must input more details for the strength calculation to define the influencing factors (for example, technological size coefficient).

Raw diameter	d	60.0000	mm	<input checked="" type="checkbox"/>
Surface factor (DIN, FKM)		None		
State during heat treatment		Pre-turned to actual diameter		
<input type="checkbox"/> Wärmebehandlung von Hohlwellen im vollen Zustand (DIN, FKM)				
Material properties (DIN, FKM)		with reference diameter		
Temperature factor (AGMA) k_s		1.0000		<input type="checkbox"/>
Miscellaneous effects factor (AGMA) k_z		1.0000		<input type="checkbox"/>
<input type="checkbox"/> Own data for Woehler line (S-N curve)				
<input checked="" type="checkbox"/> Import results into the report				

Figure 20. Material details

To define the technological size coefficient $K_{1,def}$, select one of these three options:

- **Returned to actual diameter:** The raw diameter has no effect on the technological size coefficient. The value $K_{1,def}$ is recalculated for each cross section based on the actual diameter size.
- **Raw diameter:** $K_{1,def}$ is determined once from the raw diameter and applied to every cross-section.
- **Returned to actual diameter (shoulders K1 from d):** The same as «Returned to actual diameter», however for shoulders the smallest diameter d is used to determine $K_{1,def}$.

State during heat treatment	Pre-turned to actual diameter
<input type="checkbox"/> Wärmebehandlung von Hohlwellen im vollen Zustand (DIN, FKM)	
Material properties (DIN, FKM)	with reference diameter

Figure 21. Additional information about material – heat treatment

Material properties (DIN, FKM)	with reference diameter
Temperature factor (AGMA) k_s	R_p, R_m acc. database, σ_w for reference diameter
Miscellaneous effects factor (AGMA) k_z	R_p, R_m acc. database, σ_w constant

Figure 22. Additional information about material – how KISSsoft determines the critical material characteristics, that are relevant for strength analysis

- **with reference diameter:** values are taken from database (at reference diameter) and multiplied with K_1
- **R_p, R_m according to database, σ_w for reference diameter:** The values R_p and R_m are determined according to size (excluding K_1), and the fatigue limit σ_w is determined for the reference diameter entered in the database and then multiplied with K_1 .
- **R_p, R_m according to database, σ_w constant:** The values R_p and R_m are determined according to size, the fatigue limit σ_w without the influence of the geometric size coefficient is taken from the database. The size coefficient K_1 is not taken into account.
- **R_p, R_m according to database, σ_w calculated from R_m :** The values R_p and R_m are taken from the database according to size, σ_w is determined from the tensile strength R_m according to the standard. Click the **Own data for Woehler line** checkbox to define your own Woehler line.

The surface roughness used to define the influencing factor for surface roughness has already been determined in the graphical definition of the shaft. For more information, refer to the separate Tutorial 006.

2.5.4 Selecting cross-sections for analysis

A maximum of 20 cross-sections can be checked in one calculation. All the cross-sections listed in the Elements tree or in the Shaft editor are analyzed.

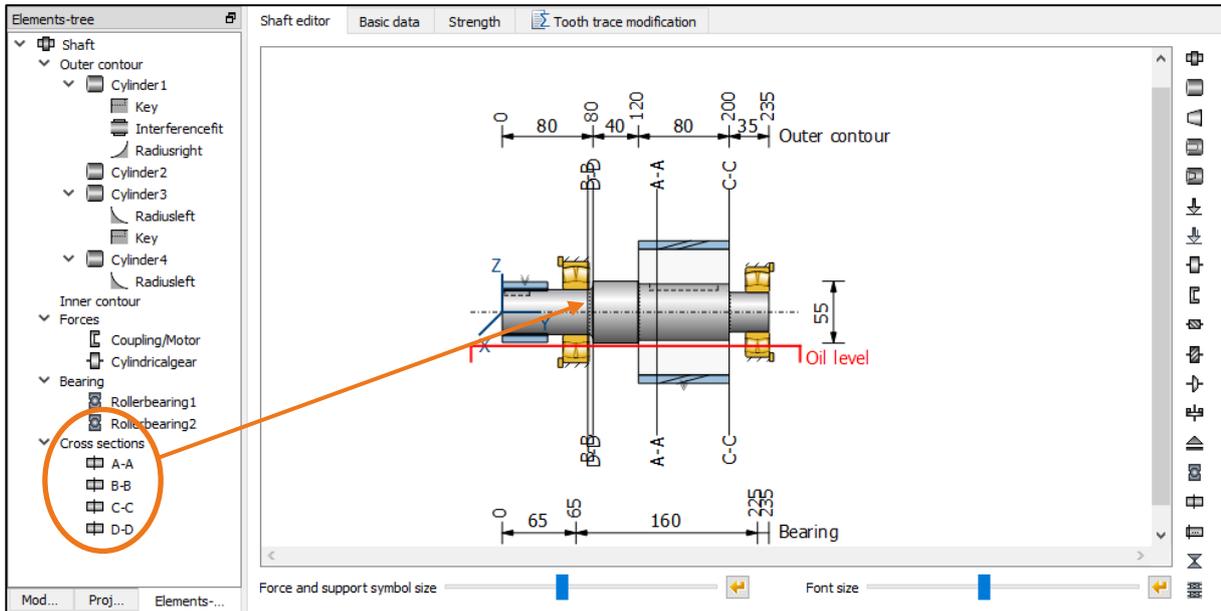


Figure 23. Cross sections to be analyzed

The position (Y-coordinate) and notch must be defined in each cross section. KISSsoft can then automatically select the cross-sections that are to be examined (using the equivalent stress and the notch factor). To do this select «**Cross sections**» with a right-hand click on the «**Elements tree**» and select «**Sizing**» (see Figure 16). The KISSsoft system then automatically finds the critical cross-sections.

You can add and analyze tree types of cross-section. In the case of a limited cross section, the notch factor is included in the geometry you enter.

A free cross section is independent of the geometry you input. In this case, you can select any value as the notch factor.

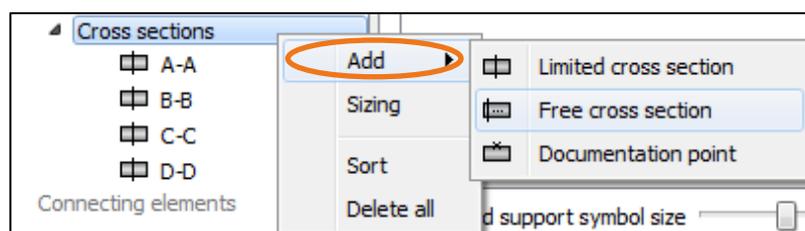


Figure 24. Inserting an additional cross-section

- You can input the Y-position for both types of cross-section (here 85 mm)
- Complete the notch details (see Figure 27), (here: cross hole with 5 mm diameter)
- Click «Shaft -editor» to examine the position of the new cross section (the cross section can also be moved using the mouse if the «Movable forces and supports» flag is set), see Figure 25.

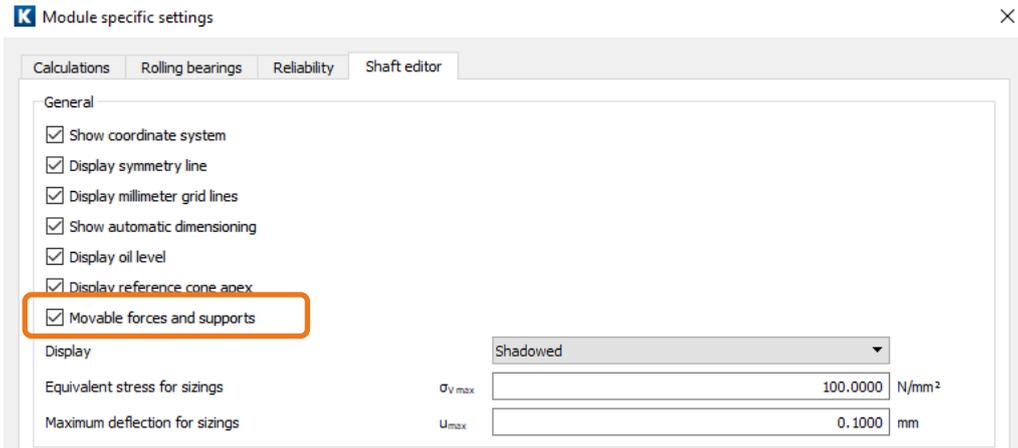


Figure 25. Setting of movable forces and supports

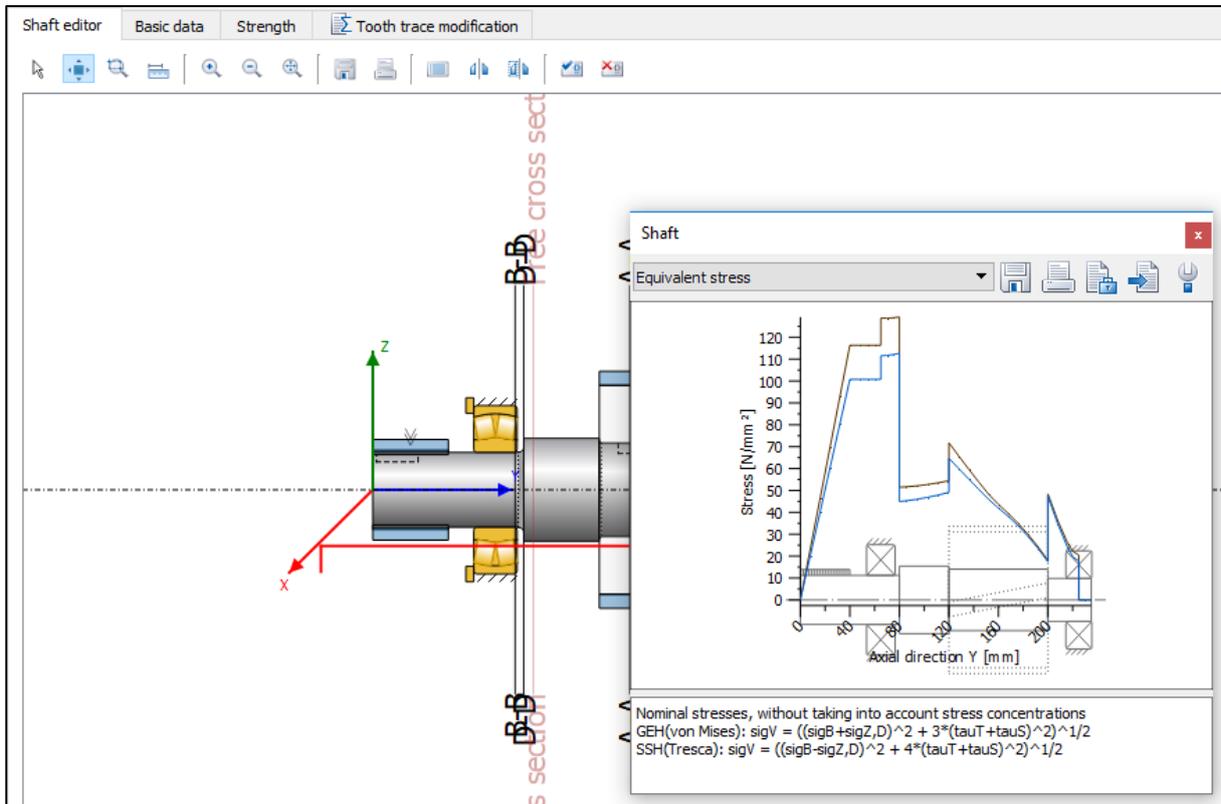


Figure 26. Graphic with cross sections (active cross section: red) and equivalent stress

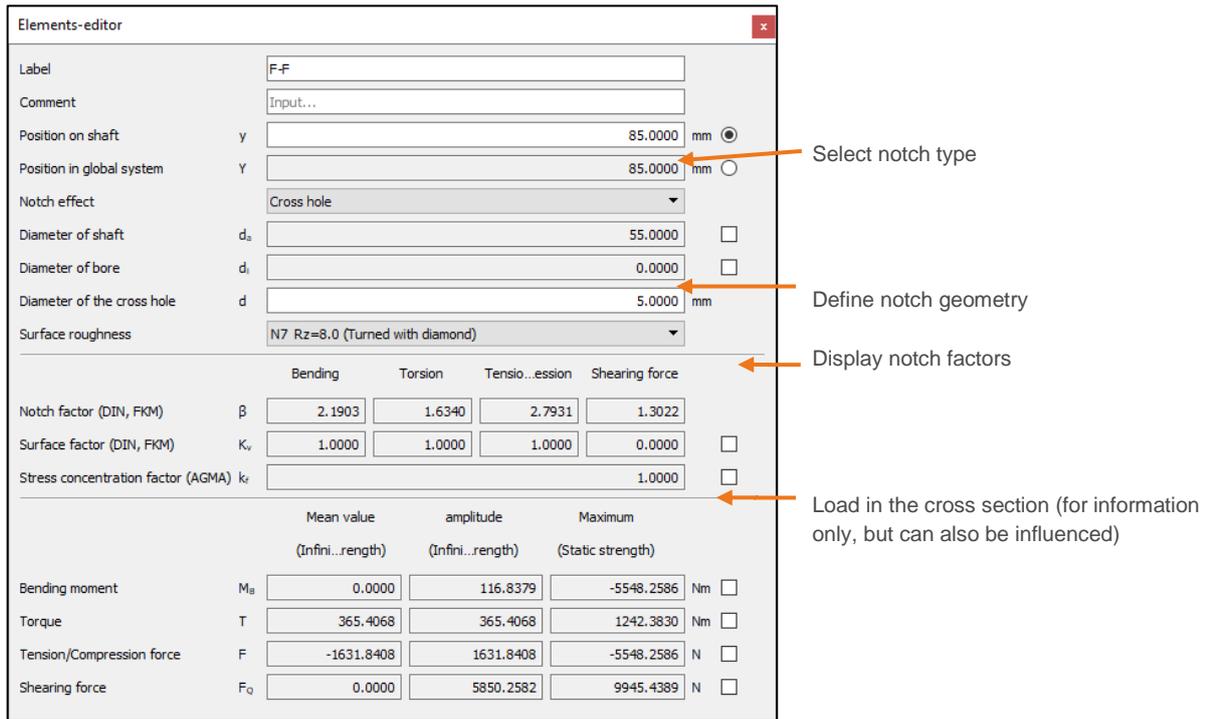


Figure 27. Definition of notch in cross section F-F

2.5.5 Analysis and results

Once all the cross sections are defined, click «Calculate»  to trigger the strength analysis. The resulting safety factors then appear in the lower part of the window. As stated in DIN 743, the minimum permitted safety is 1.20. However, this value only covers any uncertainties in the procedure and may need to be increased to prevent any damage or if you do not have precise data about load assumptions.

2.6 Reports

For a detailed description of the calculation results, click «Report» → «Generate» in the main shaft calculation window. To output additional information about a specific calculation, click the calculation report:

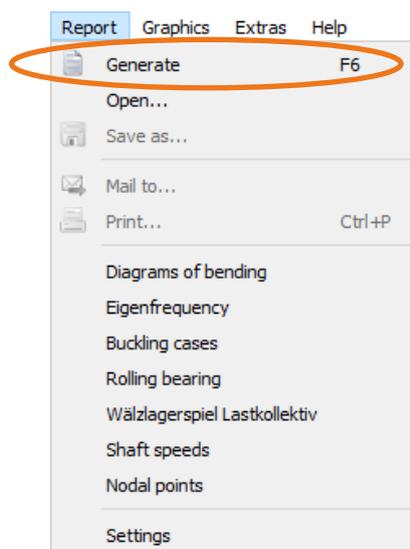


Figure 28. Selecting the report

3 Further Calculations

3.1 Additional calculations

- Torsion-critical speed: see section 2.4 Eigenfrequencies
- Buckling: calculate the buckling load (axial pressure) or necessary load increase factor
- Calculation of the tooth trace modification: this analysis returns a proposal for the crowning of, for example, a pinion to compensate for the deformation of a shaft due to deflection/torsion
- Calculation with load spectra: you can perform finite life analysis using different load spectra and different modifications of the Miner rule (elementary, extended, consistent)
- Calculation of the Campbell-diagram
- Calculation of the roller bearings and journal bearings, see section 3.2
- Calculation of multiple shafts

3.2 Combination with roller bearing analysis

The shaft analysis module in the KISSsoft system also has a user-friendly, integrated method of calculating bearings and roller bearings that are modeled on a shaft. In this case, the bearing loads are defined automatically on the basis of the external loads on the shaft and are therefore available for use in a service life analysis (for more information, see Tutorial 007).