KISSsoft AG - +41 55 254 20 50
Uetzikon 4 - +41 55 254 20 51
8634 Hombrechtikon - info@KISSsoft. AG
Switzerland - www. KISSsoft. AG

KISSsoft Tutorial: Shaft Analysis

1 Starting KISSsoft

1.1 Starting the software

Once you have installed and activated KISSsoft either as a test or licensed version, follow these steps to call the KISSsoft system. Usually you start the program by clicking "Start→Program Files→KISSsoft 03-2011→KISSsoft". This opens the following KISSsoft user interface:

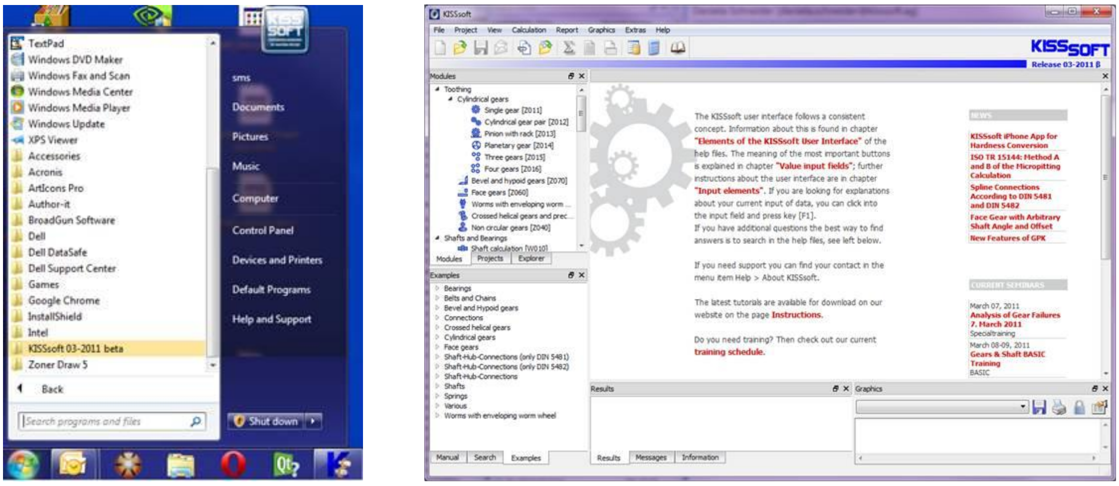


Figure 1.1 Starting KISSsoft, initial window

1.2 Selecting a calculation

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

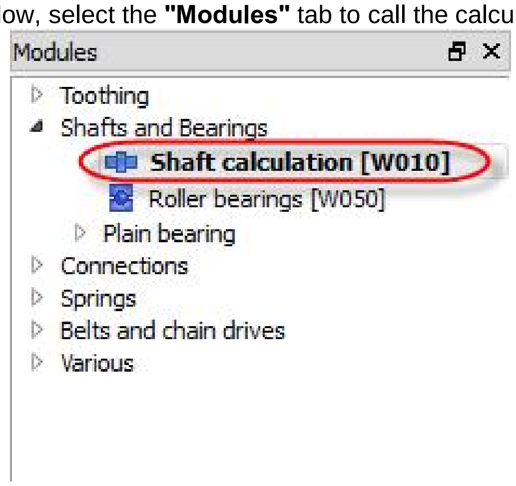


Figure 1.2 Selecting the "Shaft" calculation module under Shaft-Hub-Connections

2 Analyzing a Shaft

2.1 Task, opening the example calculation

You are to mathematically investigate a shaft that has already been modeled, see Figure 2.1. The following criteria are relevant:

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

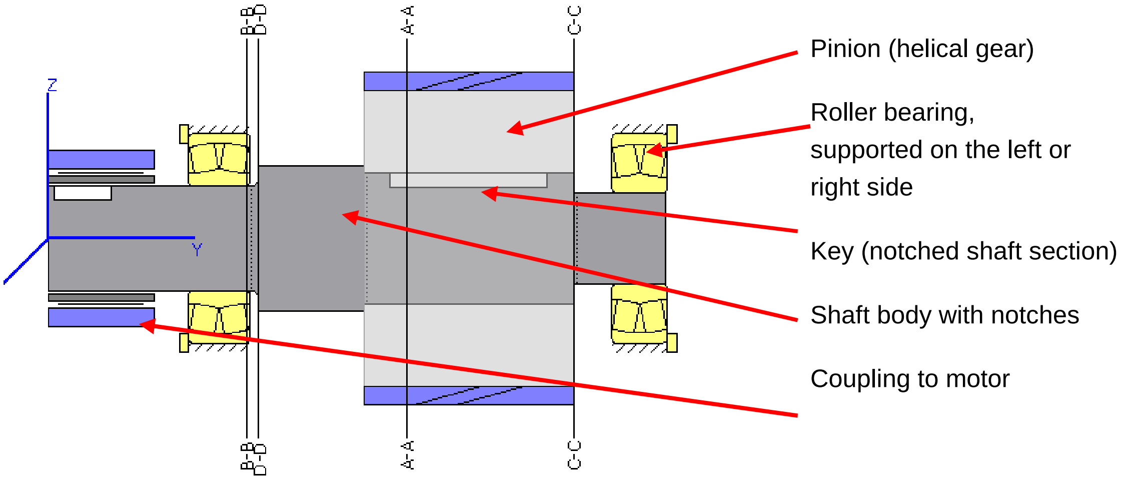


Figure 2.1 Shaft to be analyzed

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 present in KISSsoft as an example file called **Shafts 1. W10**. You can either open this file directly in the help index window by double-clicking the left-hand mouse button on the **"Examples"** tab or by selecting **"File"→ "Open"** by clicking on the file in the appropriate KISSsoft subdirectory, …\example, with **"Open"**.

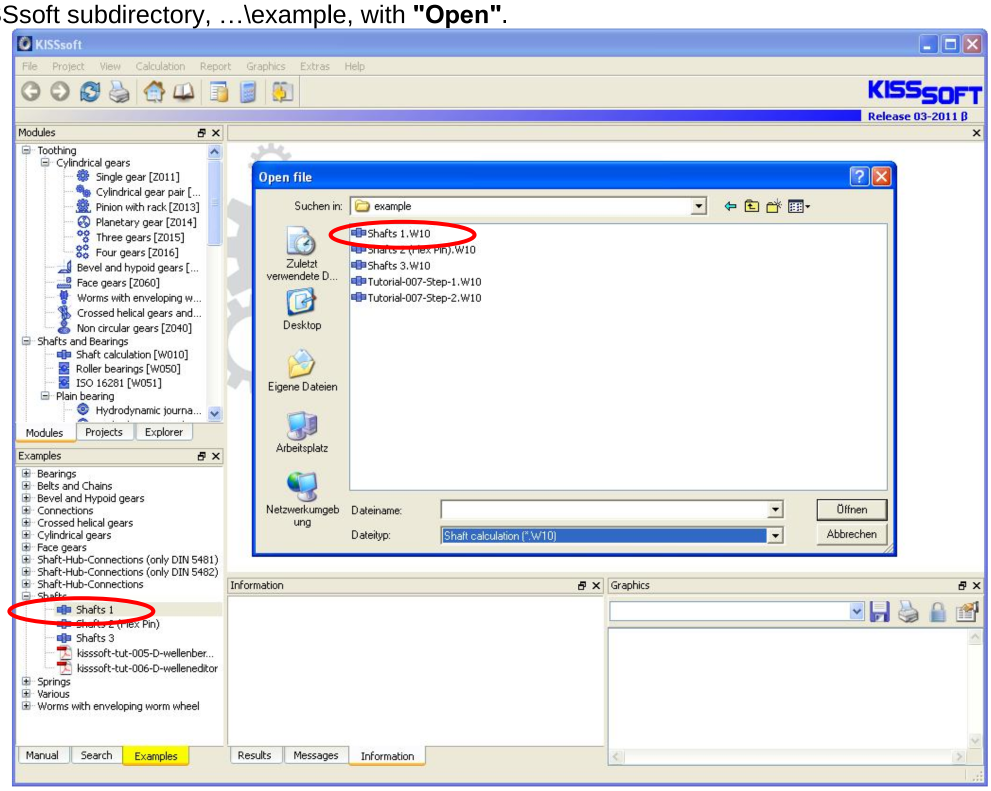


Figure 2.2 Opening the example calculation

Click the **"Shaft editor"** tab to view the tab as shown in Figure 2.1.

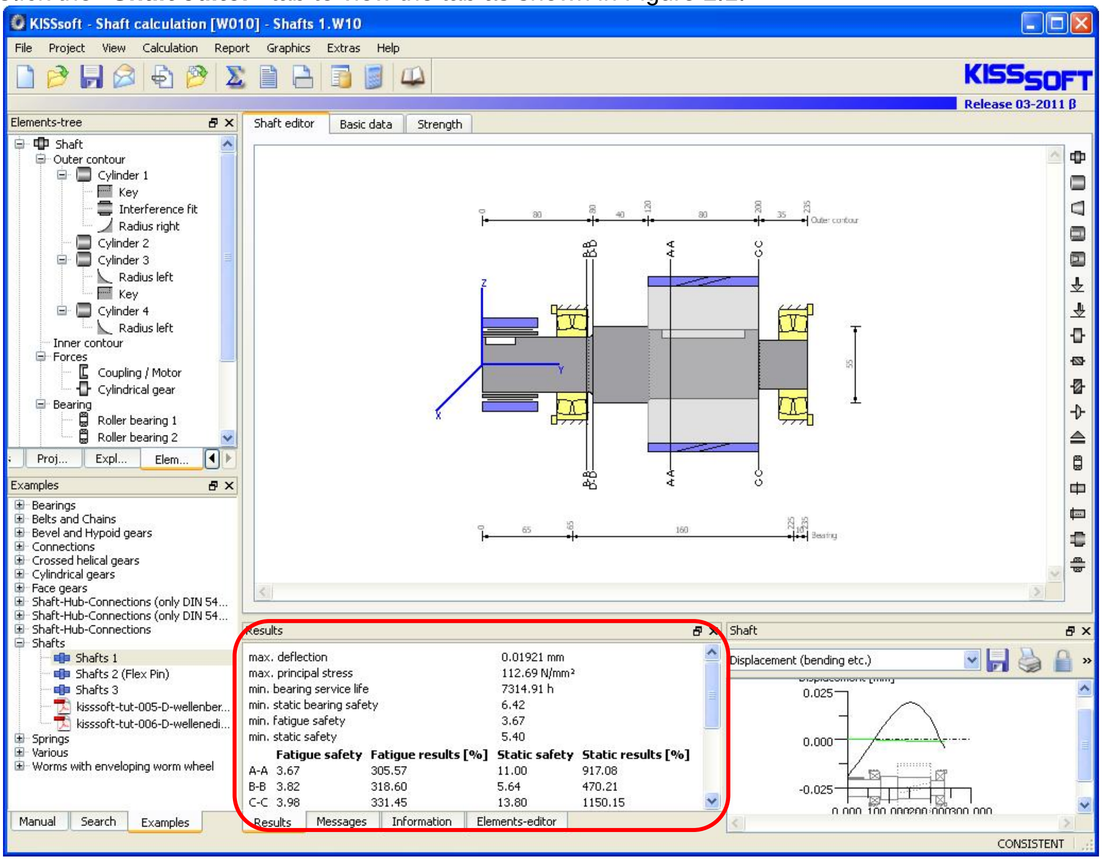


Figure 2.3 Opened shaft calculation

You will find details of how to generate a shaft model with the help of the graphical shaft data input interface in KISSsoft Tutorial No. 006.

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

max. deflection	0.01921 mm
max. principal stress	112.69 N/mm²
min. bearing service life	7314.91 h
min. static bearing safety	6.42
min. fatigue safety	3.67
min. static safety	5.40

	Fatigue safety	Fatigue results [%]	Static safety	Static results [%]
A-A	3.67	305.57	11.00	917.08
B-B	3.82	318.60	5.64	470.21
C-C	3.98	331.45	13.80	1150.15
D-D	4.12	343.58	5.40	449.90

Bearing service life	S0	Lnh	Lnmh
Roller bearing 1	6.42	7283 h	7315 h
Roller bearing 2	9.74	24565 h	45444 h

Bearing reaction force	Component	X	Y	Z	R
Roller bearing 1	F	-3.088 kN	3.264 kN	4.968 kN	5.850 kN
	M	0.000 Nm	0.000 Nm	0.000 Nm	0.000 Nm
Roller bearing 2	F	-1.501 kN	0.000 kN	7.242 kN	7.396 kN
	M	0.000 Nm	0.000 Nm	0.000 Nm	0.000 Nm

Figure 2.4 Enlarged display of results

2.3 Deformation analysis

In the **"Basic data"** tab, click the drop-down list to open the "Strength" group. This is where you input how the gears modeled on the shaft are to be taken into account in 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, along with the external loads to the shaft, it does not stiffen the shaft)
- Consider gears as mass and as stiffness (the gear is attached rigidly to the shaft and forms a single unit with the shaft)

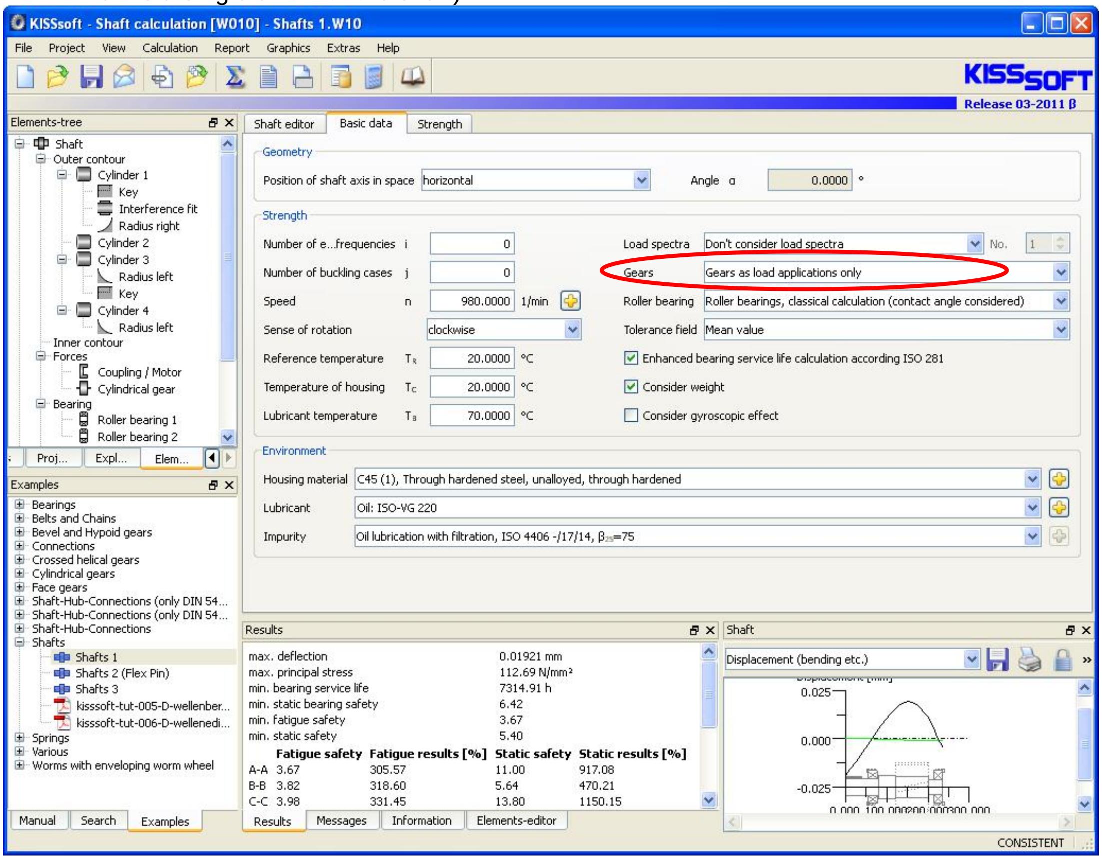


Figure 2.5 How gears are considered

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

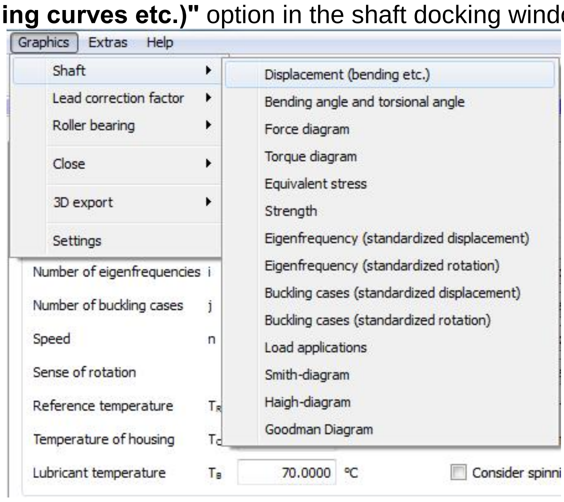


Figure 2.6 Options for displaying shaft analysis as a graphic

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.

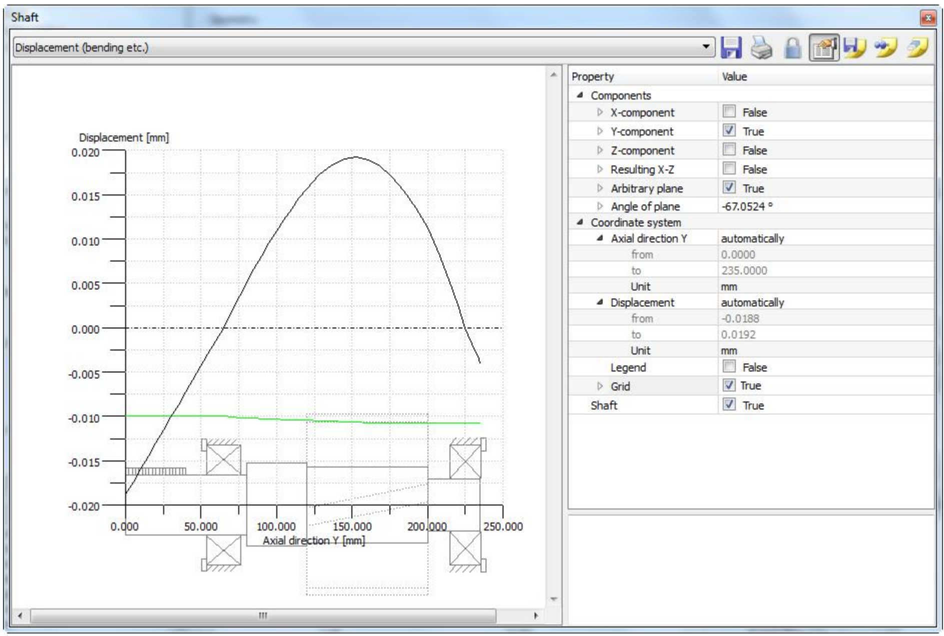


Figure 2.7 Display produced by selecting the Displacement option

The maximum displacement shown in the results window, as the result of helical toothing and the associated moments on the x and z axes, results in a deflection such that the maximum displacement at plane α=-67.05°, is ux'=19.2 μm. This is the largest value from all possible planes.

Alternatively, select **"Report"→ "Diagrams of bending"** to display a list of calculated sizes.

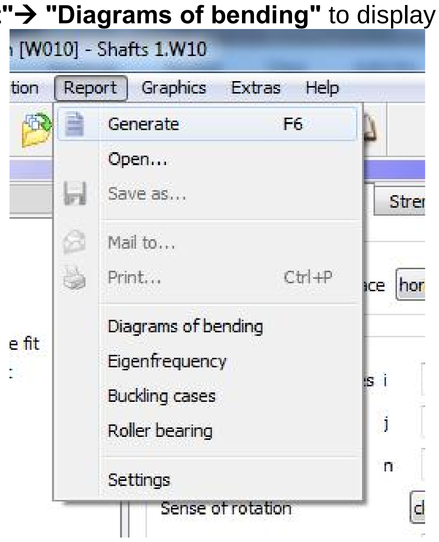


Figure 2.8 Calling the Diagrams of bending report

2.4 Selecting eigen frequencies

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 eigen frequencies to be calculated (three) and click the "Calculate" icon, Σ, to trigger the calculation process. (For more information about the calculation, press "F1".)

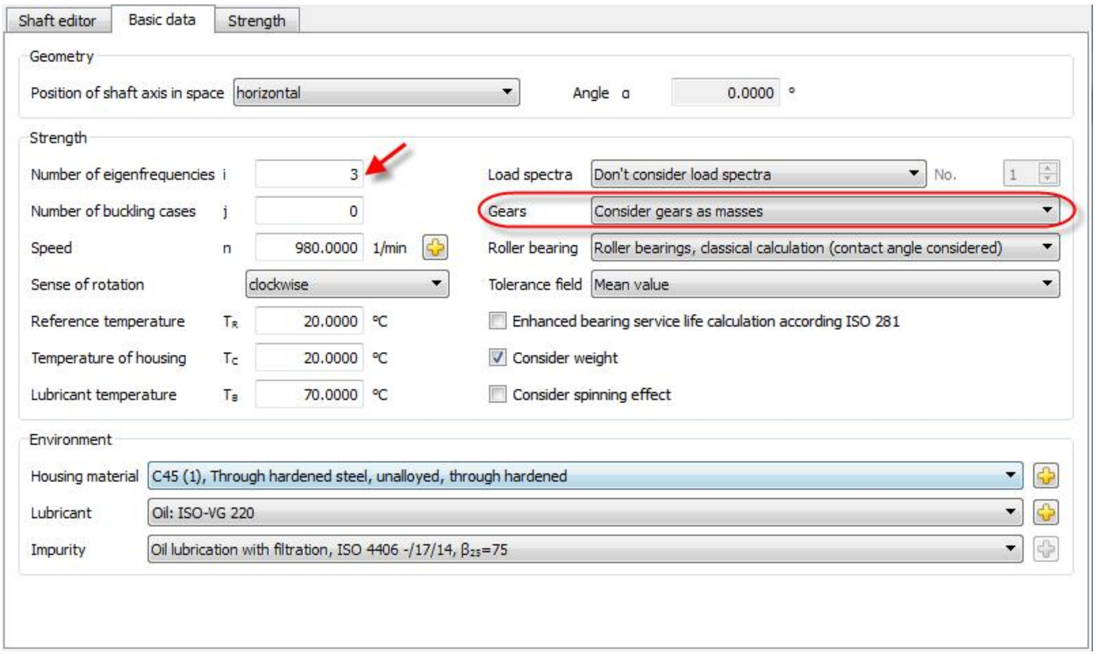


Figure 2.9 Inputting the number of eigen frequencies

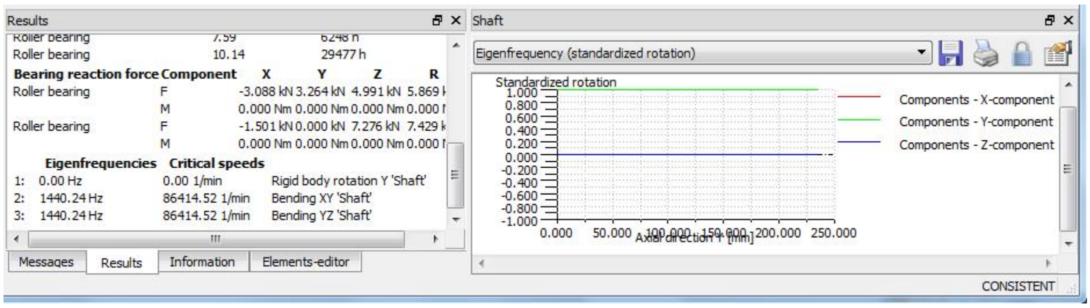


Figure 2.10 Result of calculating the number of rotations

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

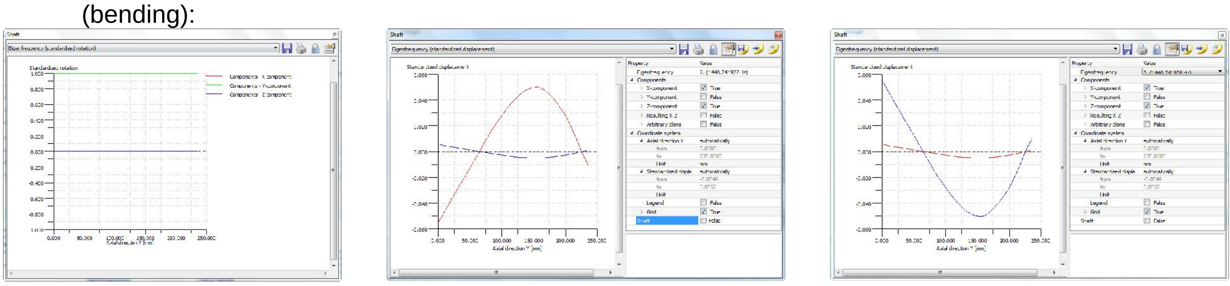


Figure 2.11 The three resulting eigen frequencies

2.5 Strength analysis of shafts

Open the **"Strength"** tab (see Figure 2.12) to determine the settings for strength analysis. At present, KISSsoft has three strength calculation methods: Hänchen and Decker, DIN 743 and the FKM Guideline. This example uses DIN 743.

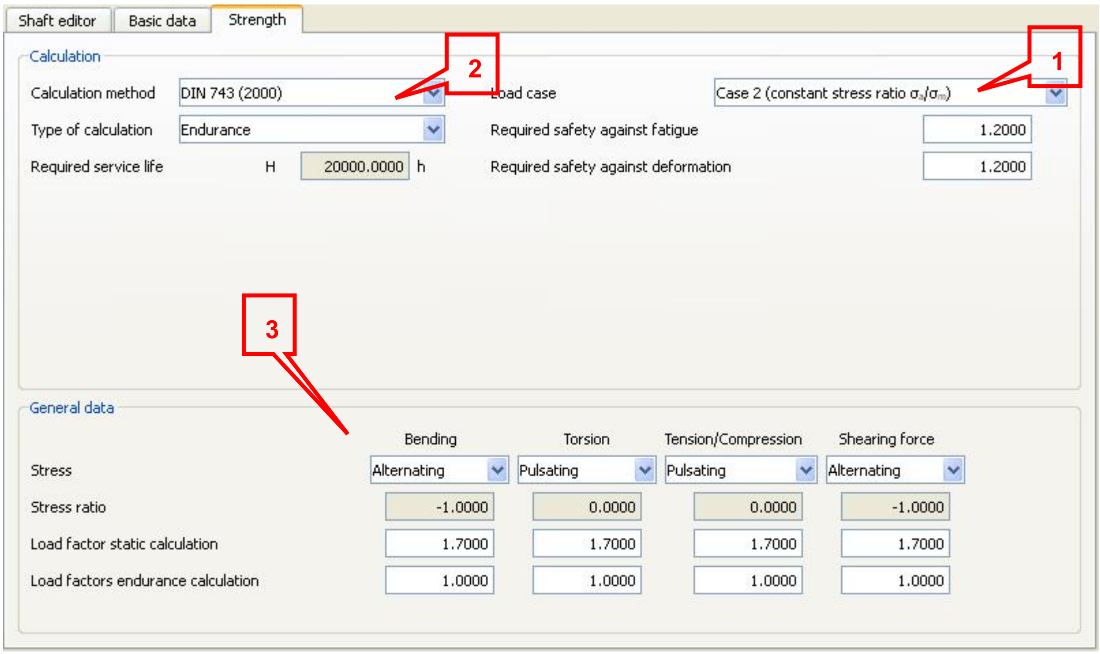


Figure 2.12 Buttons involved in strength calculation

(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.

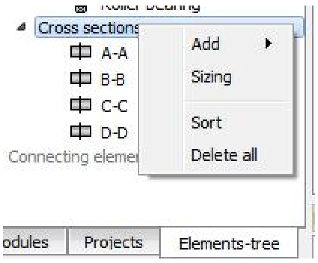


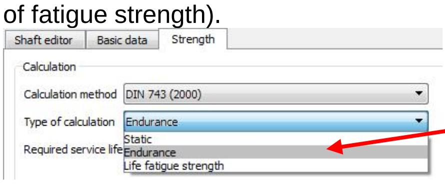
Figure 2.13 Calling the cross sections to be analyzed - automatic sizing

2.5.1 Data used for 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).



Selecting the proof of fatigue strength. In this case select a proof of endurance limit.

Figure 2.14 "Calculation" group in the "Strength" tab - 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

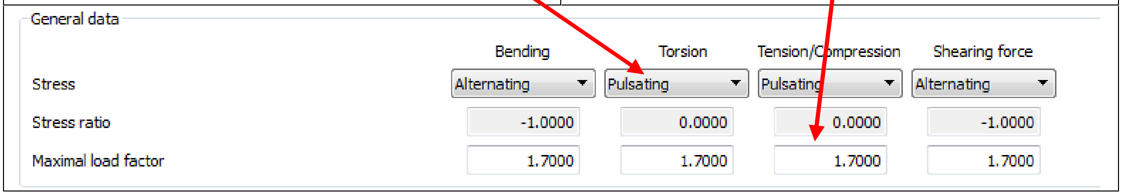


Figure 2.15 "General data" group in the "Strength" tab - details about the type of load

In the case of shafts that are to be loaded with different frequencies for bending and torsion (for example, quickly rotating drive shafts with pinions that are stopped and started once in a while), 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 applicable.

2.5.3 Information about a material

The **"Basic data"** tab in the shaft calculation module, see Figure 2.9, is where the 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.

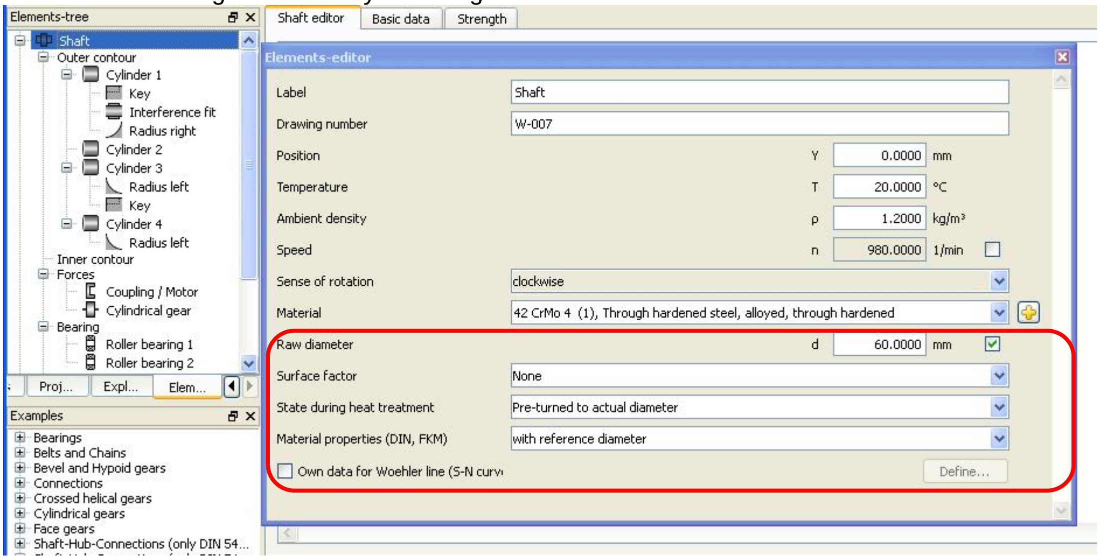


Figure 2.16 Shaft material

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

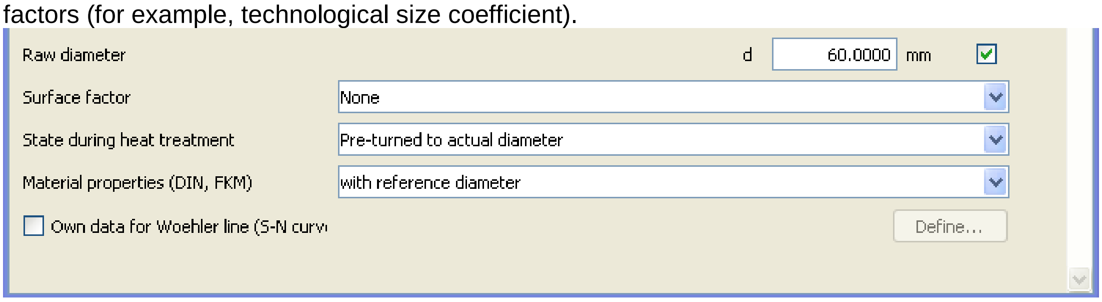


Figure 2.17 Material details

To define the technological size coefficient $K_{1,\,deff}$, 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 to all cross sections.

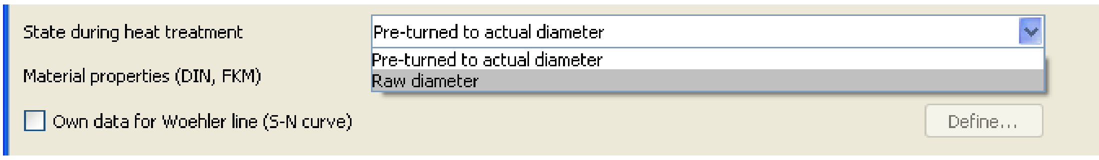


Figure 2.18 Additional information about material - heat treatment

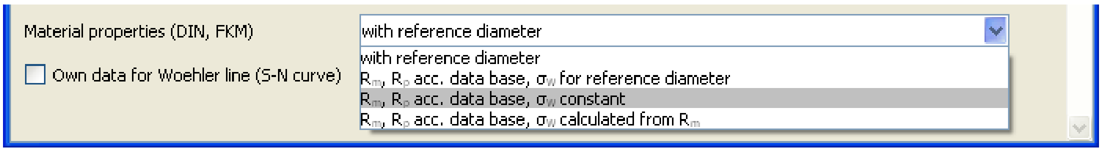


Figure 2.19 Additional information about material - how KISSsoft is to determine the critical material characteristics that are relevant for strength analysis

- **with reference diameter:** values are taken from database (at reference diameter) and multiplied with K1
- **R_p, R_m as stated in the database, σ_W 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.
- **R_p, R_m as stated in the database, σ_W 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 factor K_1 is not taken into account here.
- **R_p, R_m as stated in the database, σ_W calculated from R_m**: the values R_p and R_m are determined from the database according to size, σ_W is determined from the yield point R_m in accordance with the standard. Set a flag in the "**Own data for Wöhler 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 No. 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.

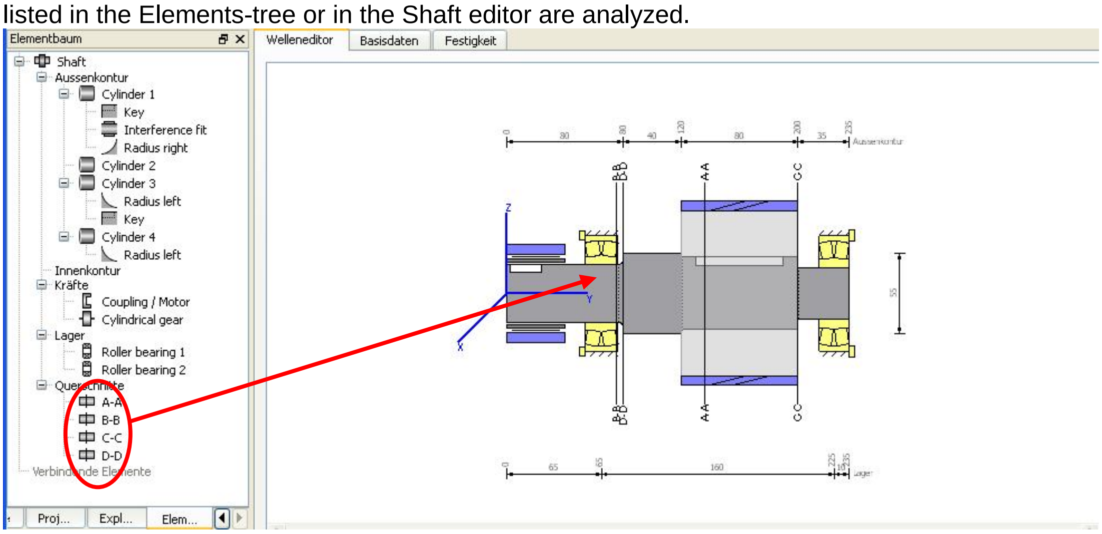


Figure 2.20 Cross sections to be examined

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"** and then right-hand click on the **"Elements-tree"** and select **"Sizing"** (see Figure 2.13). The KISSsoft system then automatically finds the critical cross sections.

You can add and analyze two 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 2.21 Adding another cross section

- You can input any value as the Y-position for both types of cross section (here 85 mm)
- Complete notch details (see Figure 2.23) and click "OK" to confirm this (here: cross bore with 5 mm diameter)
- Click "Graphics" to view the position of the new cross section (you can also move the cross section with the mouse). See Figure 2.22

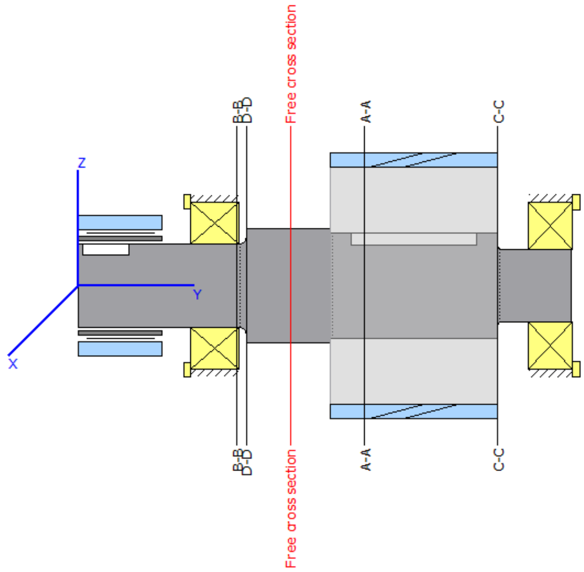


Figure 2.22 Graphics with cross sections and equivalent stress (active cross section: red)

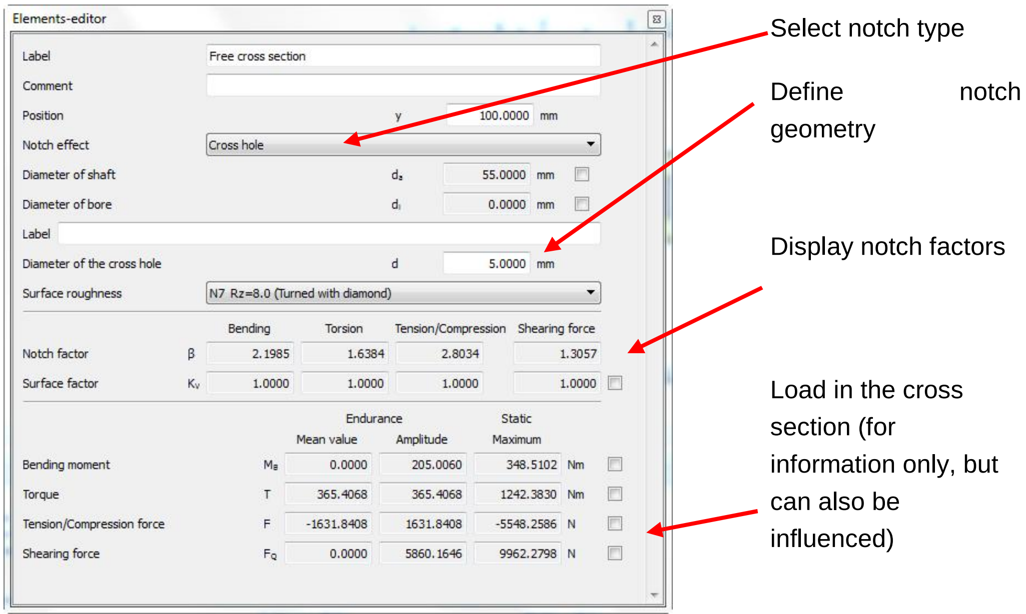


Figure 2.23 Definition of notch in cross section FF

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:

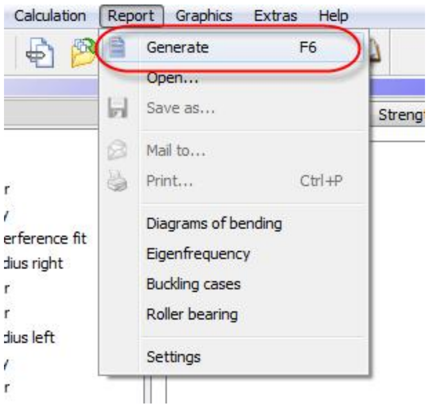


Figure 2.24 Selecting reports

3 Further Calculations

3.1 Additional calculations

- Torsion-critical revolutions: see section 2.4 Selecting eigen frequencies
- Buckling: calculate the buckling load (axial pressure) or necessary load increase factor
- Calculation of crowning: this analysis returns a proposal for the crowning of, for example, a pinion to compensate for the deformation of a shaft due to bending/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 bearings and roller bearings, see section 3.2
- Calculation of other 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).