KISSsoft 03/2017 – Tutorial 6

Shaft editor

KISSsoft AG
Rosengartenstrasse 4
8608 Bubikon
Switzerland

Phone: +41 55 254 20 50
Fax: +41 55 254 20 51
info@KISSsoft.AG
www.KISSsoft.AG

SHARING KNOWLEDGE

Table of Contents

1 Starting the Shaft Editor

1.1 Starting the Shaft Editor

For more information, refer to KISSsoft Tutorial 005, Shaft Calculation, Section 1.2.

1.2 Call the Shaft Editor, Settings

The main shaft calculation screen (Figure 1) consists of these tabs: **"Shaft editor"**, **"Basic data"** and **"Strength "**. In the Shaft editor you can model various shafts, taking into account their constraints and loads. These shafts can then be used in other calculations (deformation, strength etc., see Tutorial 005).

You can input fundamental settings (shaft position, speed, sense of rotation) directly in the **"Basic data"** tab. Click on the "Calculation/Settings" menu item to make further settings (for more information about calculation-specific settings: press "F1" Help/Manual):

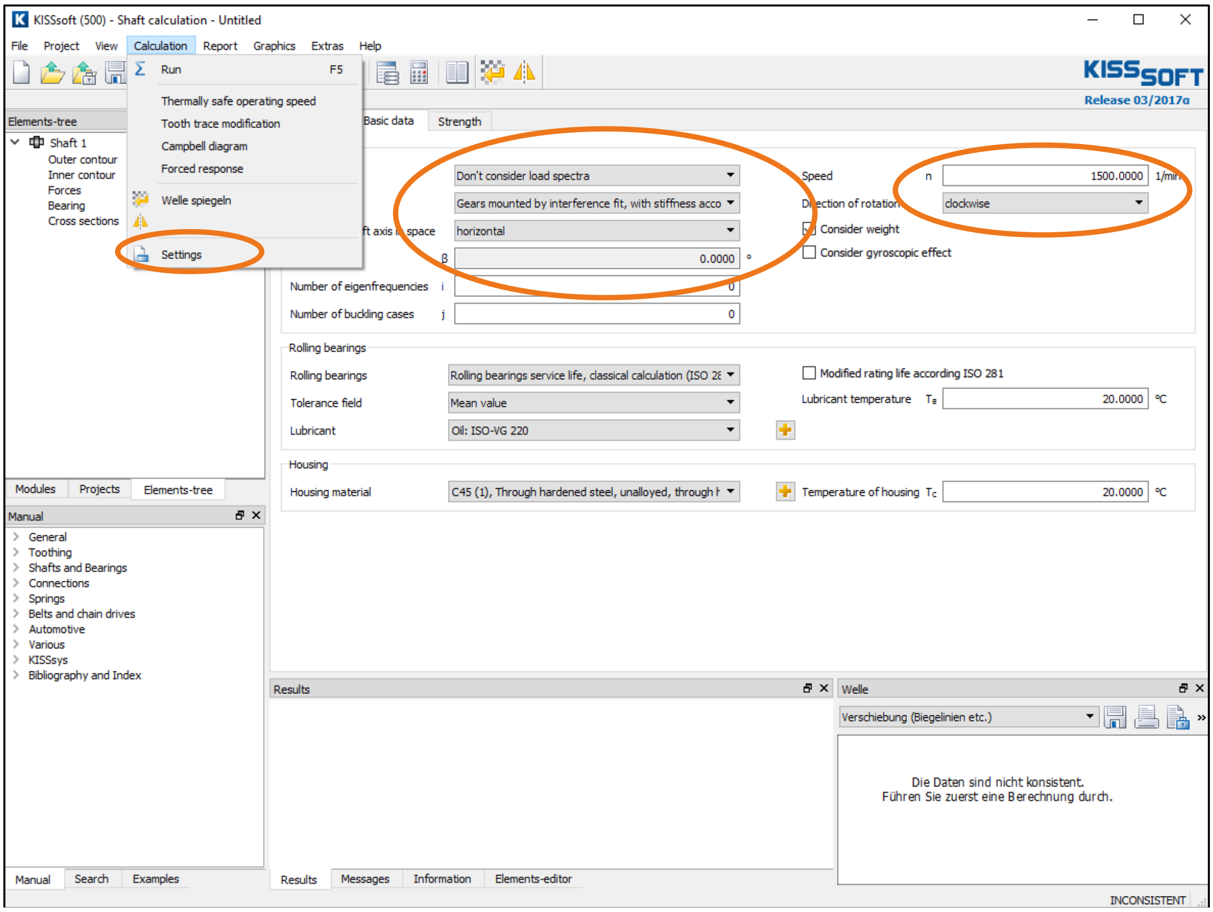


Figure 1. Shaft editor, module-specific settings, input speed, materials etc.

2 Modeling a Shaft

2.1 General Remarks

To model a shaft in the KISSsoft system, you must input the main dimensions, notch geometry, external loads and supports/constraints and critical cross sections. You define these values in the **"Elements-tree"**. To do this, select the appropriate element and right-hand mouse click to display a list of elements that can be added to the calculation. You can also create sub-elements for axially symmetrical contours (cylinders and cones).

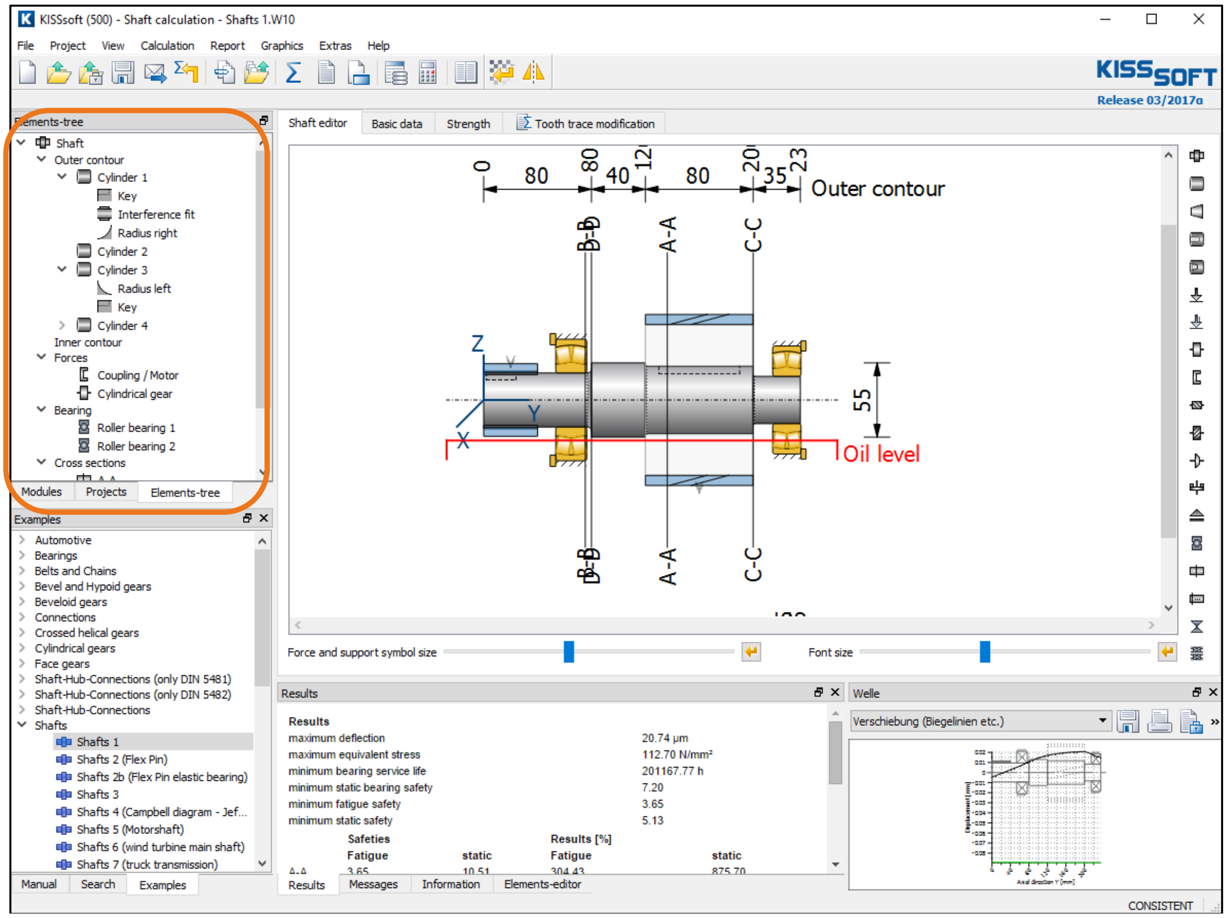
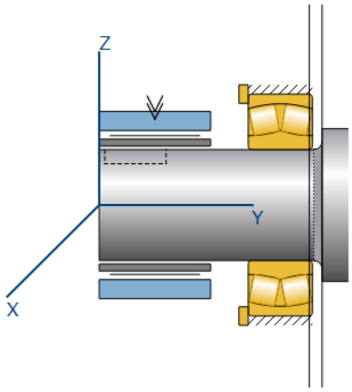


Figure 2. The five most important elements required to create a shaft

Elements	Data about	Necessary data	To define	Color
Outer/inner contour	Shaft section	Diameter and length of a shaft section, surface roughness	A, I_{xx}, I_{zz}, I_p, W_{xx}, W_{zz}, W_p	Gray
Cylinder/Cone	Notch	Type of notch, geometry	α_k	White
Forces	Loads	Centric or eccentric force/vectors of moment, machine elements for load introduction	F_y, Q_x, Q_z, M_{bx}, M_{bz}, T	Blue
Bearing	Bearings	Selection of roller bearings, information about bearing stiffness, degrees of freedom	Reaction forces, constraints	Yellow
Cross sections	Critical cross section	Effect of notch, position, geometry, surface roughness	Notch factors, strains	Black

2.2 Co-ordinates system

Note that the coordinates system is right-handed and Cartesian



Positive x-axis: Out of the screen
Positive y-axis: Along the length of the shaft, from left to right
Positive z-axis: From bottom to top
Position of contact: Positive angle counter clockwise (looking towards the positive direction of the y-axis), with zero position in the x-axis.

You can display or hide the coordinates system by activating/deactivating the "Show coordinate system" checkbox in the **"Module specific setting"**.

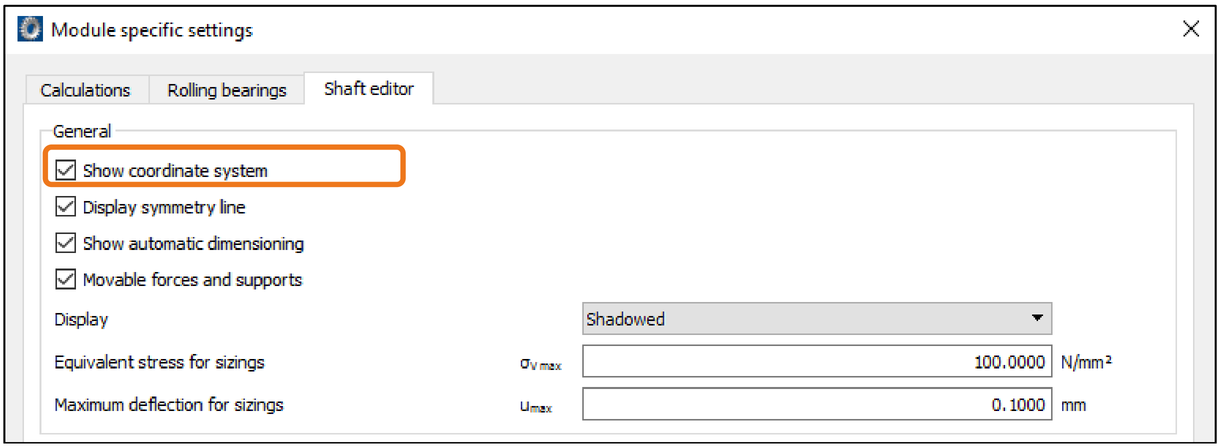
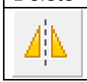


Figure 3. Shaft editor tab in Module specific settings dialog

Alternatively you can, with the right mouse button, click the corresponding sub-menu in the shaft editor selected for the activation (see Figure 4).

2.3 Editing functions, screen view

The following editing functions are available in the graphical Shaft editor:

Function	Explanation
+ key, - key, Home key	Zoom in/zoom out/full screen
Mouse left-click	Select element/move element (when flag is enabled)
Mouse right-click	Zoom in/zoom out/full screen
Delete	Delete selected element
	Shaft mirror

To move the forces and bearing icons, the corresponding sub-menu can be selected for activation by right mouse click.

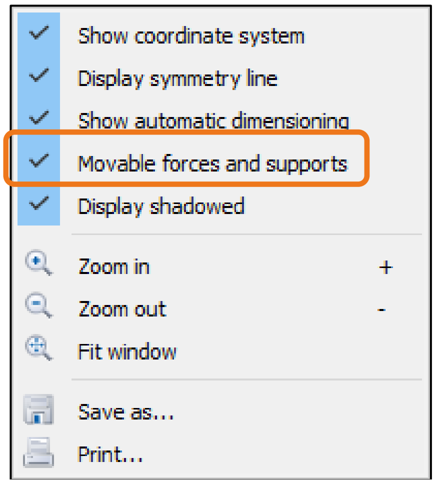


Figure 4. Selection of sub-menu with right mouse click

On the right-hand side of the Editor you see the following icons that make it easier to input the elements:

Add shaft
Add cylinder
Add conus
Add bore
Add conical bore
Centric force/vector of moment
Eccentric force/vector of moment
Add machine element, cylindrical gear, etc. ...
…
…
…
…
Add bearing
Add roller bearing
Add limited/free cross section
Add general joint
Add connecting bearings

2.4 Entering main dimensions

To define a shaft section, select the element in the **"Elements-tree → Outer contour"** or select the element in the shaft editor, or select the element in the shaft editor. (Figure 5).

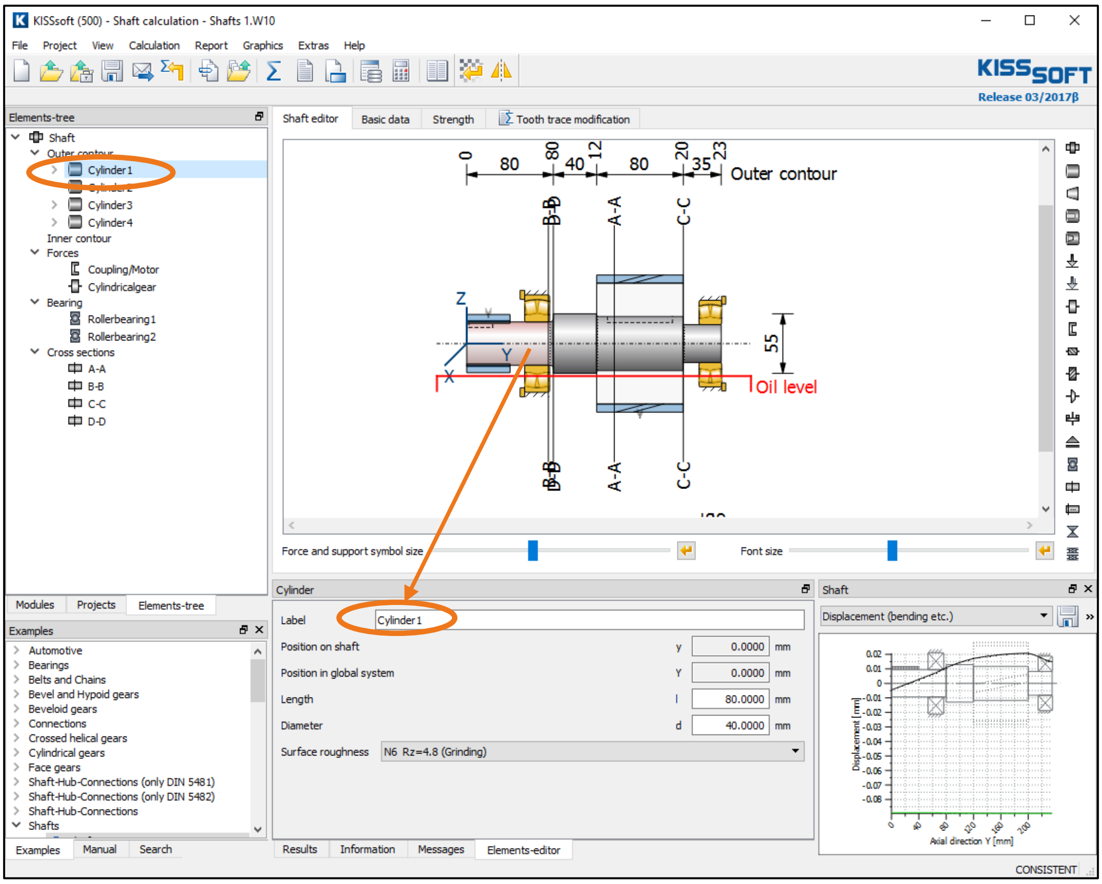


Figure 5. Dialog in which you define a shaft section (cylinder)

You then define the diameter, length and surface roughness in the Elements-editor. Every new shaft section you add can be positioned either before (to the left) or after (to the right) of existing sections. To add a shaft section to an existing section, first select the existing section with a left-hand mouse click and then right-hand mouse click and select **"paste before"** or **"paste after"** to insert the new shaft section. To modify an existing element in the Elements editor, simply select it with a left-hand mouse click.

If a main element is active in the Elements-tree, e.g. “Outer contour”, the Elements-list is shown (Figure 6). There, it is possible to modify the elements, too.

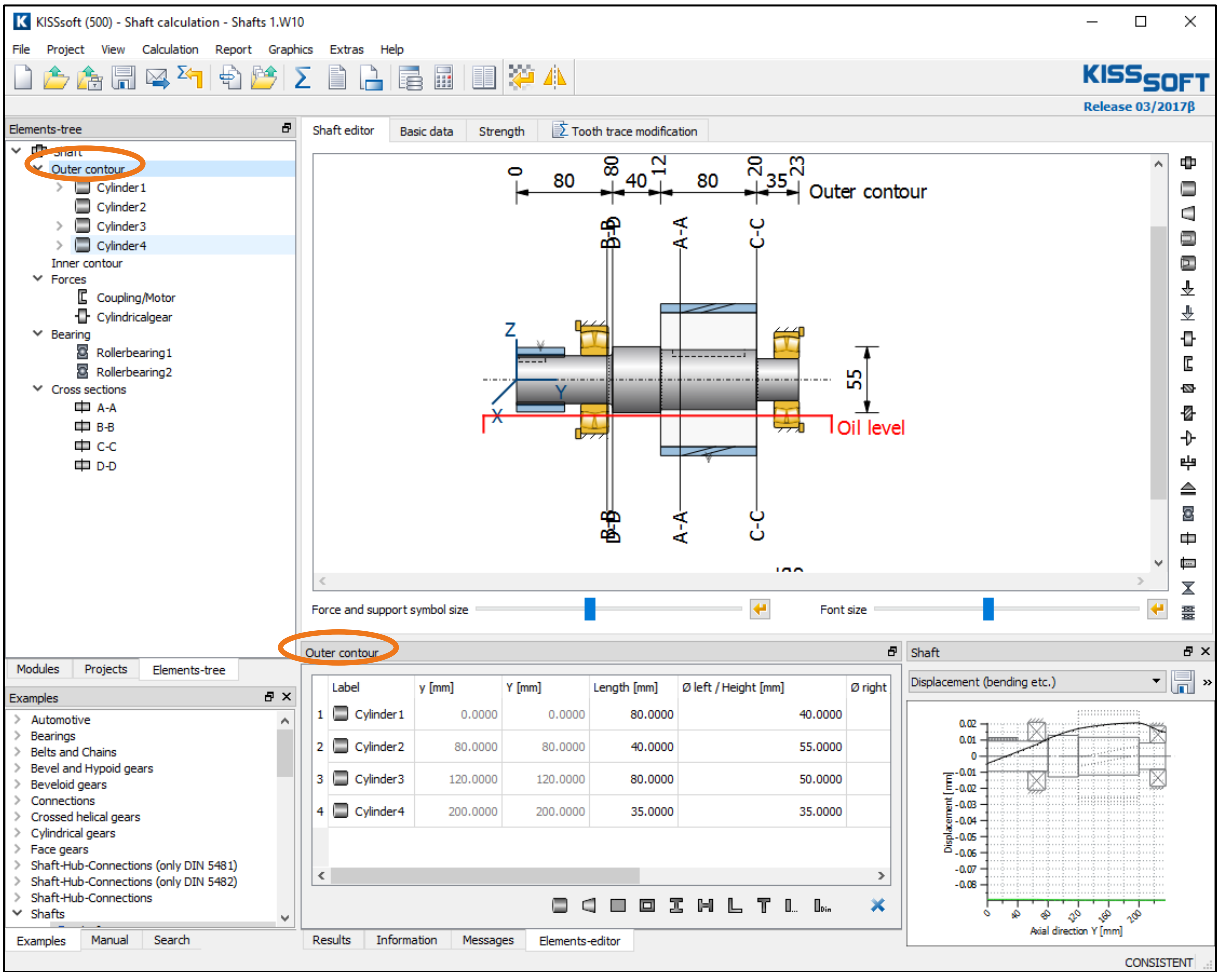


Figure 6. Call the Elements-list

Bores (**"Inner contour → cylindrical bore"**) are also added to the existing shaft, from left to right. However, if you only want the bore to be present on the right-hand end of the shaft, first define a bore with diameter zero from the left-hand end of the shaft.

2.5 Inputting Notch Geometry

Before you can add a notch, you must select a shaft section (left-hand mouse click on the section, to display that section in red) or a suitable element in the **"Elements-tree"** on which the notch is to be positioned. First right-hand mouse click on shaft section in the Elements-tree and then select the appropriate notch. Here you can select one of the various notch types that are available (for example, circumferential groove). After you select the notch type, you see a dialog in which you can define the geometry and position (reference dimension: the left end of the selected shaft section) of the notch.

The sub-elements **"Radius"**, **"Chamfer"** and **"Relief groove"** do not need to be positioned because these are set automatically on the selected end (left/right) of the current shaft section.

To add a general notch effect, select **"General notch effect"**. You input the corresponding notch factors directly in the **"Elements-editor"**.

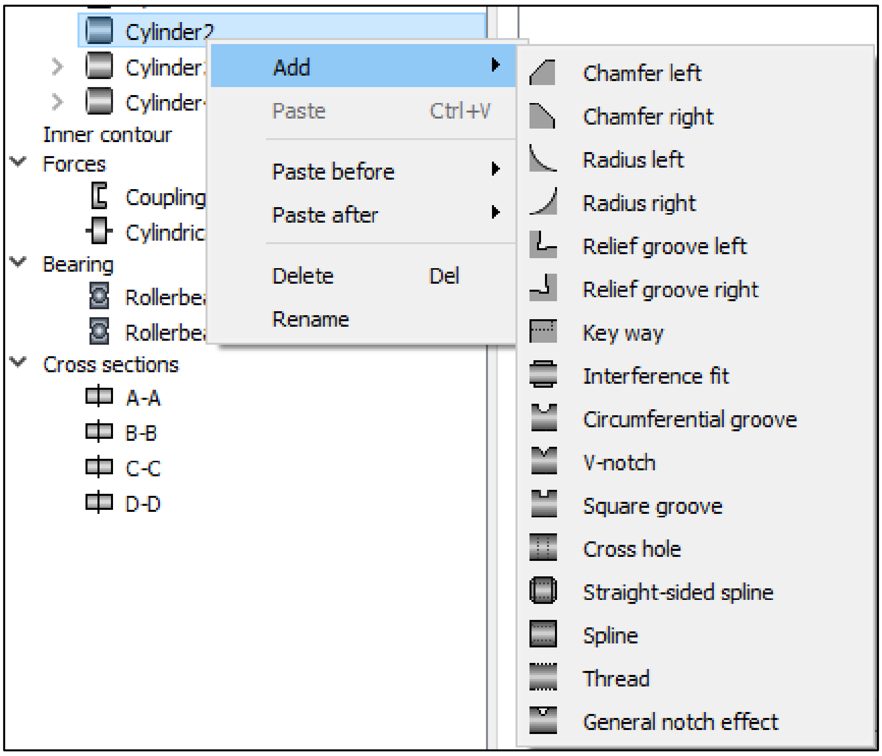


Figure 7. Dialog for defining sub-elements

2.6 Inputting Loads

2.6.1 General forces

Select **"Forces→Centrical load"** or **"Forces→Eccentric load"** in the **"Elements-tree"** to define a load vector (three forces, three moments), that have either an effect on the mid line of the shaft or an eccentric effect. You can also predefine linear loads. The force vector is shown symbolically in the graphic as a single arrow in the negative z-direction and not as a vector.

2.6.2 Machine elements

In addition to allowing the input of general force vectors, KISSsoft also provides a range of predefined machine elements. Here, the resulting forces on the shaft are determined from the geometry of the machine elements and the predefined power. As you no longer need to convert the helix angle and reference diameter, for example, manually, the load data you input is significantly less prone to errors.
In **"Forces"**, select a machine element. The dialog shown in Figure 8 then appears.

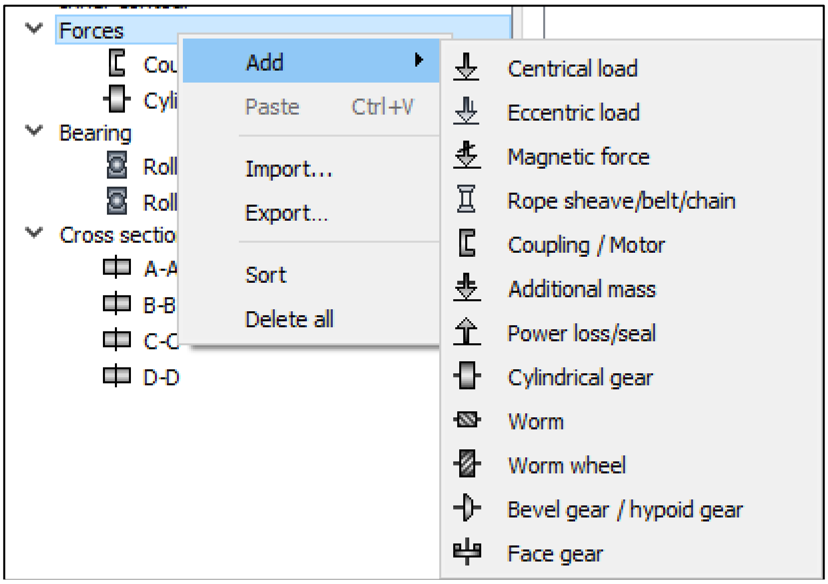
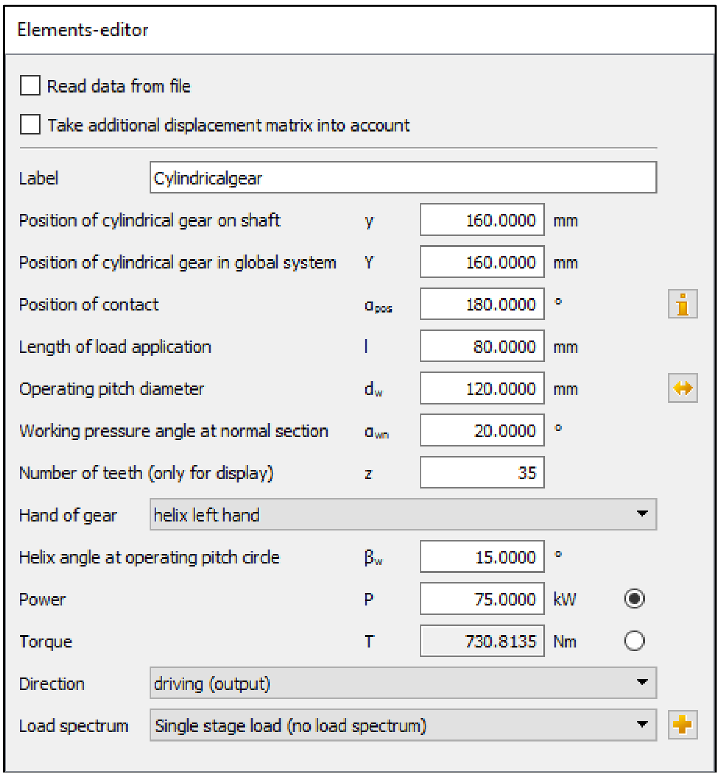


Figure 8. Adding machine elements

Here you can now define and position the machine element. You can also define the load and its effective direction.



Elements-editor for defining the **"Cylindrical gear"** element
Direction: (power flow)
Driving: The shaft drives a system. Power is drawn from the machine element.
Driven: the shaft is driven from outside. Power is applied to the machine element.
Position of contact:
From the positive x-axis, counter clockwise (to the positive z-axis.)
Position of the cylindrical gear on the shaft (y-coordinate)

Figure 9. Definition of a machine element, using the example of a cylindrical gear

Two machine elements are used in this example: one coupling/motor (Figure 10) and a cylindrical gear (Figure 9).

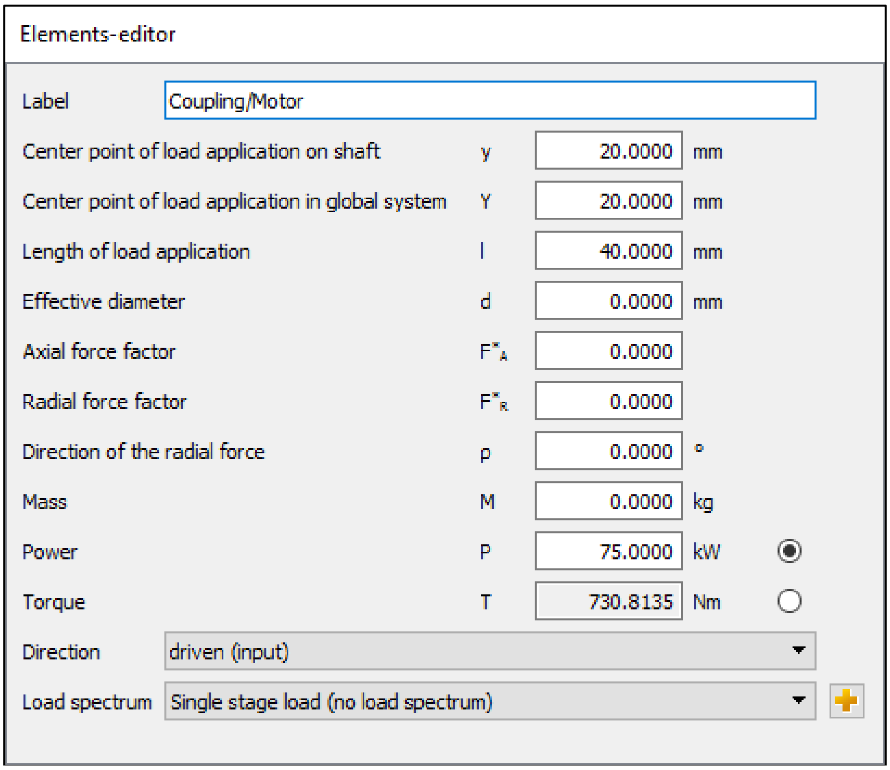


Figure 10. Definition of the coupling

Here power is applied to the shaft from outside, which is why it is **"driven"**.
You must ensure that the total power applied to and drawn from the system equals zero, otherwise, this warning message appears:

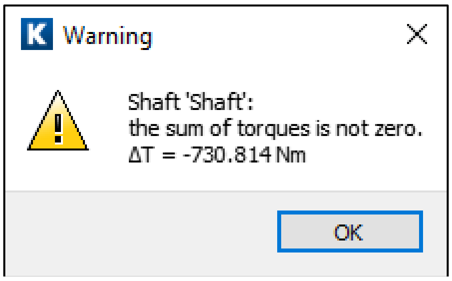


Figure 11. Warning message to tell you that the power balance in the system is not equal

2.7 Adding Bearings

In KISSsoft system you can define either general bearings or roller bearings.

2.7.1 General bearings

A bearing is defined in the Elements-tree. In the Elements-tree, click on "Bearings" and then right-hand mouse click on **"Support"** or **"Roller bearing"** to add these elements. A simple dialog appears in the Elements-editor. You can then position the bearings (input the bearing's y-coordinate, measured from the left-hand end of the shaft). In the next step, enter the bearing type.

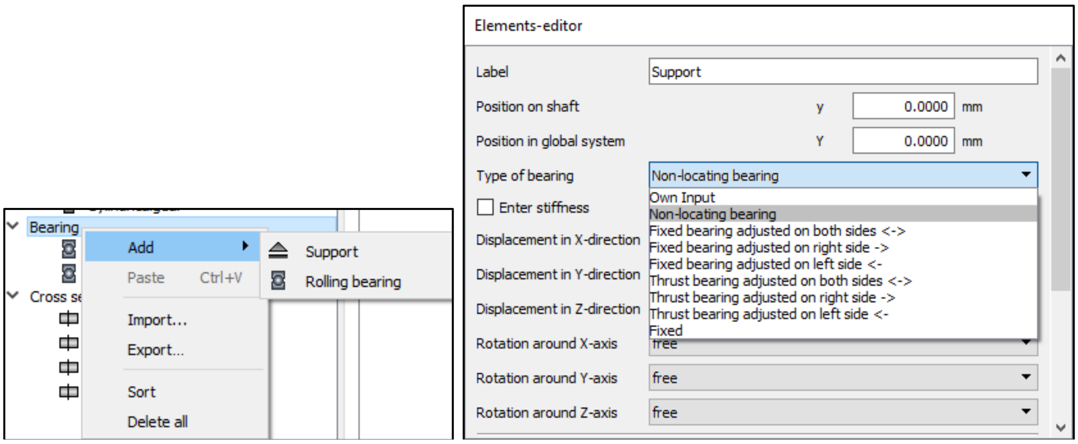
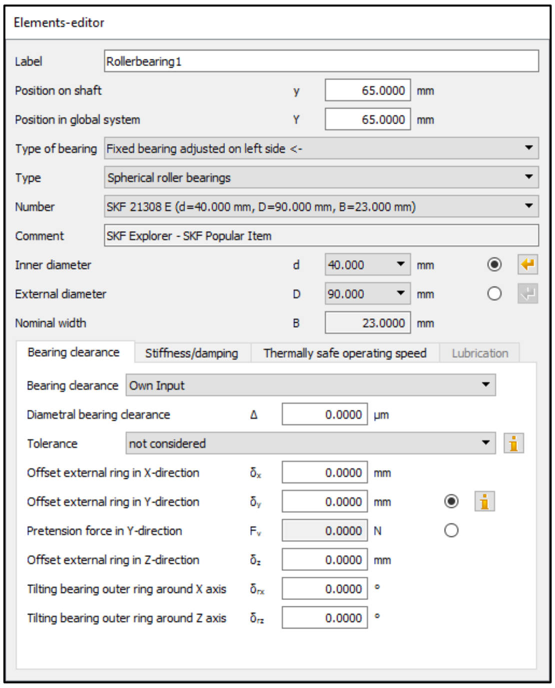


Figure 12. Dialog for positioning and selecting the bearing type

2.7.2 Inputting roller bearings

In the Elements-tree, click on "**Bearings**" and then right-hand mouse click on **"Roller bearing"** to add these elements. Alternatively, you can select the bearing in the graphic. To do this, click the **"Shaft editor"** tab. You can then define the bearing data:



Positioning bearings

Will the bearing take axial forces?
Select bearing construction
Select bearing type
The bearing diameter is determined automatically on the basis of the bearing position

Select radial bearing clearance group
If required, specify bearing offset

If required, specify bearing stiffness

If required, input factors for the calculation of the thermally safe operating speed

Figure 13. Defining a roller bearing

2.7.3 Special bearings

You can also input stiffness values for bearings. These are then taken into account when calculating load, deformation and also when calculating eigenfrequencies. This allows you to simulate the influence of roller bearing or housing stiffness. However, you must determine this using a different method (for example, in an FEM calculation).

You must use several individual bearings to fully model linear bearings. In this case, you must use bearings that have resilience/stiffness values!

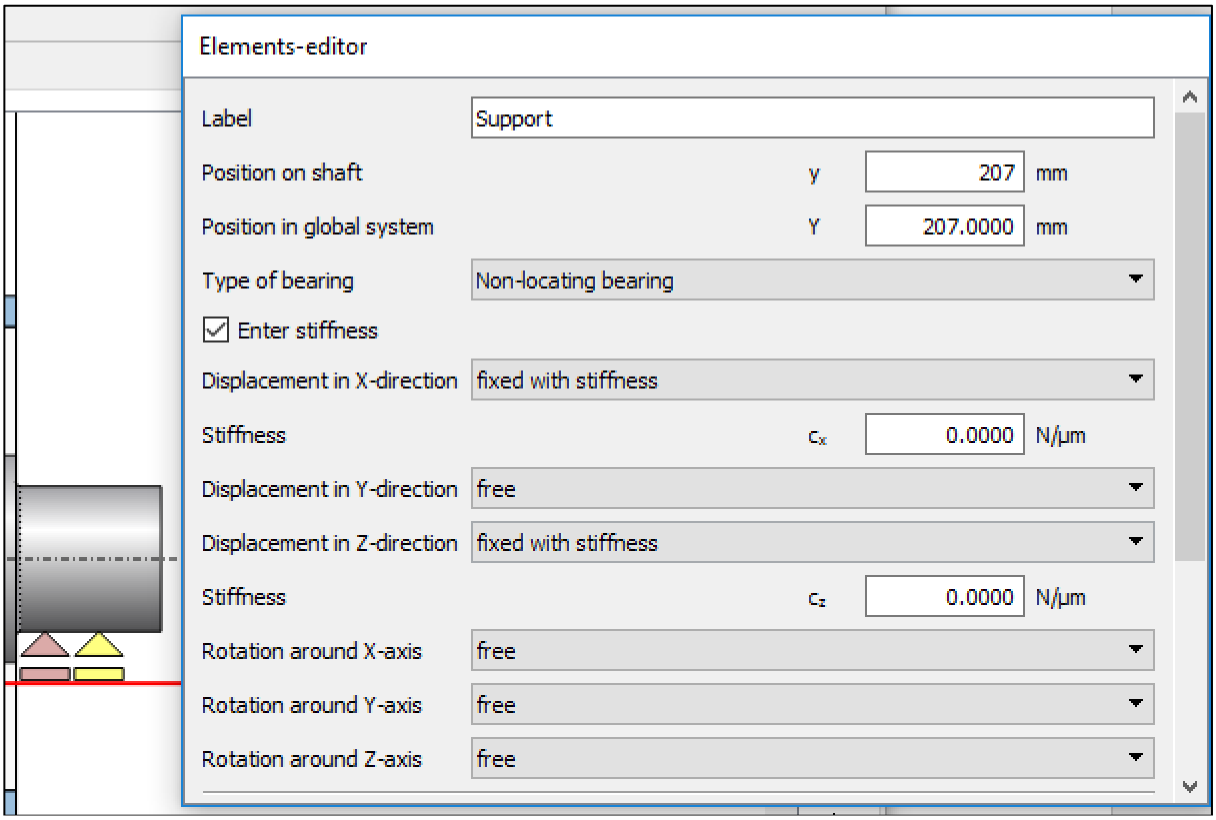


Figure 14. Example showing modeling of a line bearing using several individual bearings

2.8 Sizing for constant equivalent stress

Now optimize the shaft geometry (main dimensions) so that the equivalent stress (nominal stress) is more or less constant along the length of the shaft (lightweight construction). In the module-specific settings, input the data on which this calculation is based (see Figure 15). Here you can input the equivalent stress on which the calculation is based (in this example 100 N/mm^2).

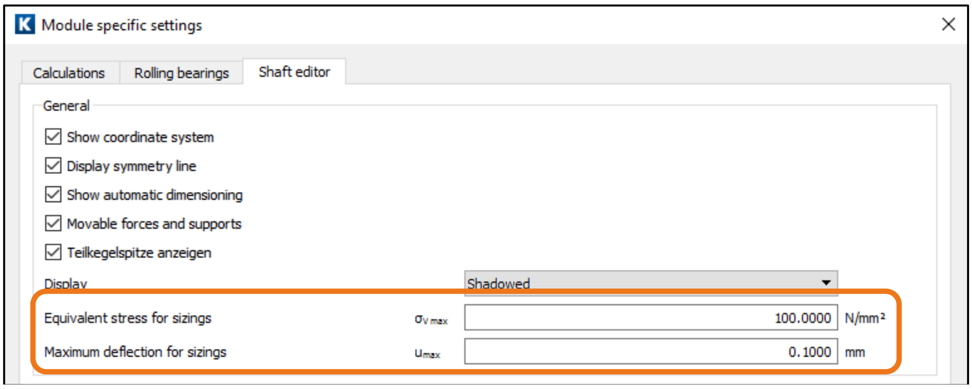


Figure 15. Predefining permitted values for the sizing

In the menu, click **"Sizing" -> "for strength"** to start the calculation to define the outer contour of the shaft for the specified load so that the predefined equivalent stress is achieved in each cross section.

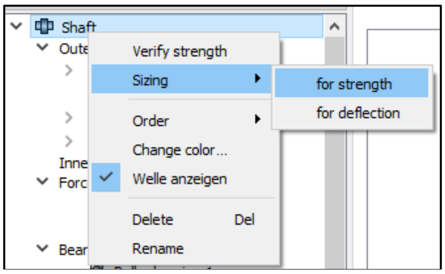


Figure 16. Call the "Sizing"-> "for strength" menu option by clicking with the right-hand mouse button on the shaft

The resulting contour appears as a green line. You can now modify the shaft's dimensions – insofar as is practical to suit the design – to match this green line. This enables you to achieve an even load (excluding the notches) along the entire shaft

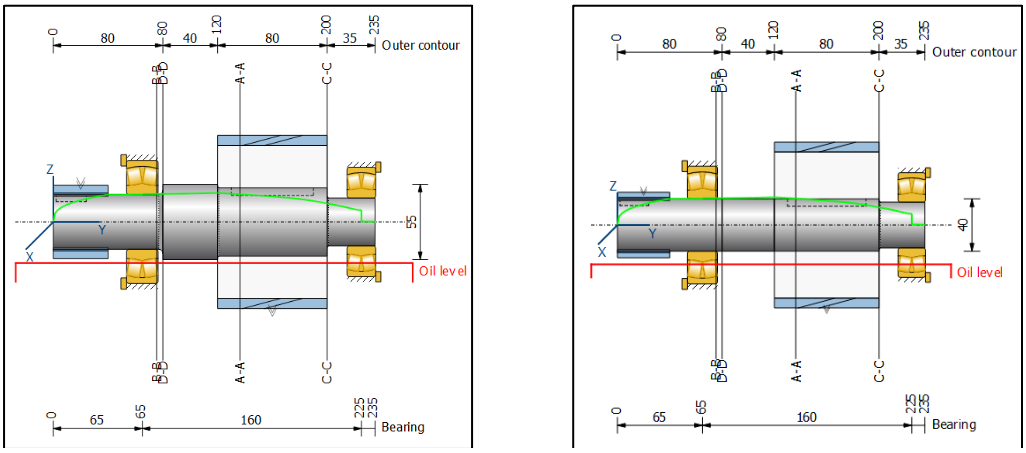


Figure 17. Original geometry with proposal for the diameter (green line), right: Modified shaft geometry, with "Sizing" -> "for strength" menu called

2.9 Sizing for maximum bending

In a similar way, you can also optimize shaft geometry to ensure that the permitted bending (deflection) is not exceeded. This permitted deflection is predefined here as shown in Figure 14 (here 0.1 mm). After you click **"Sizing"→ "for deflection"** to call the calculation, a factor is calculated by which all the shaft diameters are increased/reduced to ensure that the degree of deflection matches the predefined value. The resulting shaft geometry is then displayed as a magenta line. You can now modify the shaft's dimensions – insofar as is practical to suit the design – to match this magenta line.

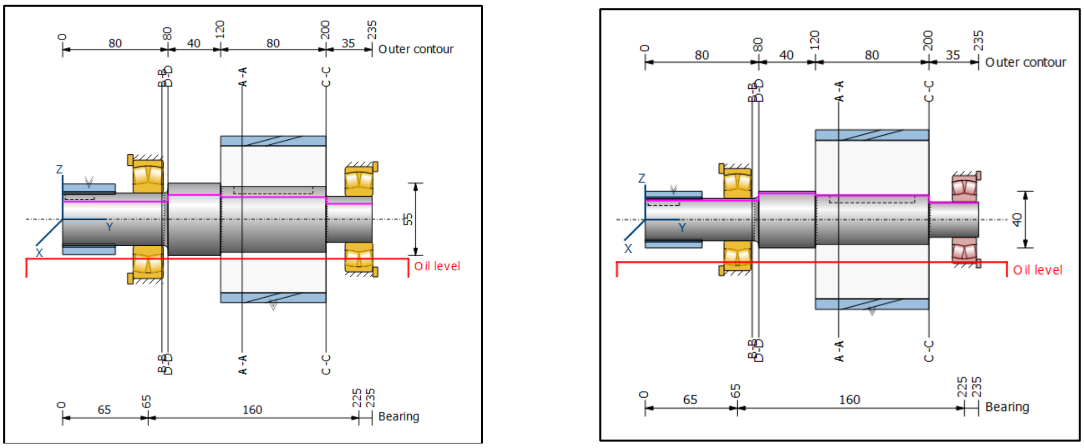


Figure 18. Original geometry with proposal for the diameter (magenta line), right: Modified shaft geometry, call "Sizing"-> "for deflection"

2.10 Several shafts

You can now use the shaft system's calculation module to input and analyze several coaxial shafts at the same time in the same shaft calculation. This type of modeling is very useful, for example, when analyzing the shaft in a planetary gear.

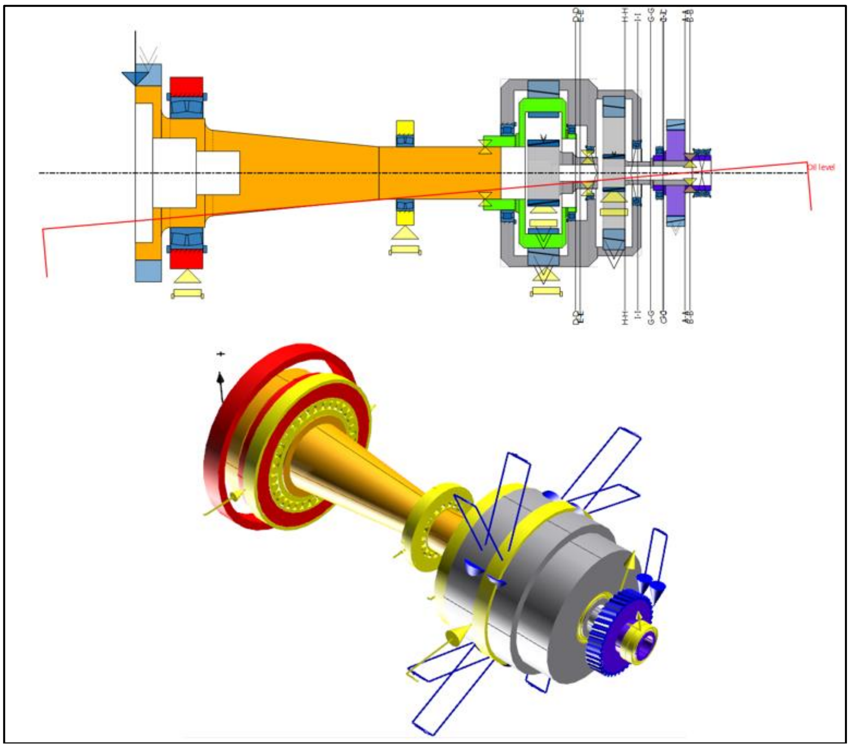


Figure 19. Example Shafts 6 (wind turbine main shaft)