

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 Editor

### 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 Calling the Shaft editor, settings

The main shaft calculation screen (Figure 1.1) 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). The "**Basic data**" tab is where you input the basic settings (shaft position, speed, sense i.e. direction of rotation) either directly or by clicking the "Calculation/Settings" menu option (for further information about calculation-specific settings, press "F1" to call the Help/Manual).

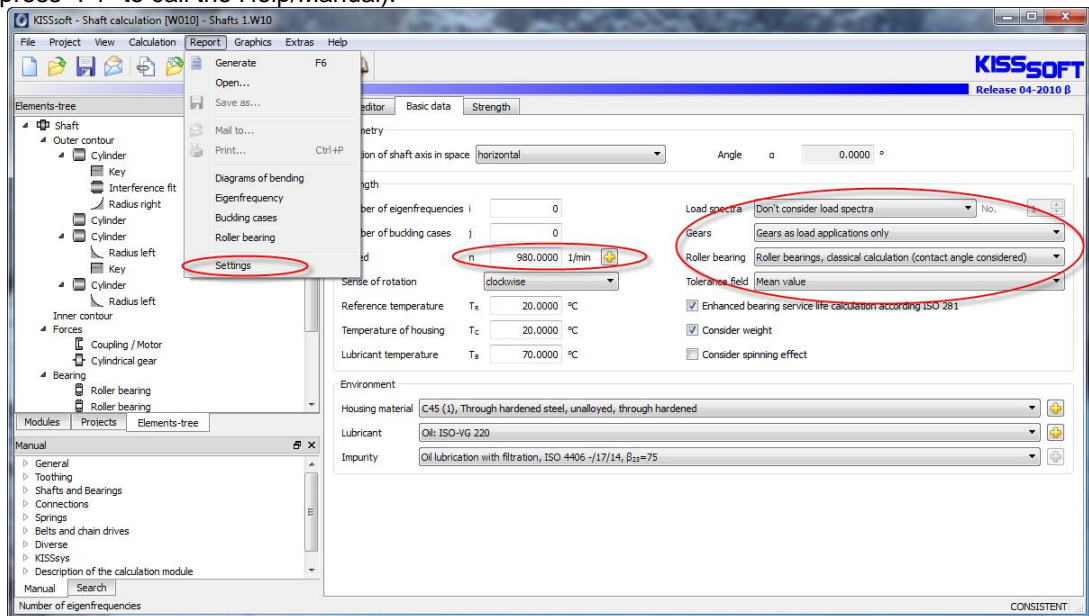


Figure 1.1 Shaft editor, module-specific settings, input speed, materials etc.

### 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 positioning/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 on a list of elements that can be added to the calculation. You can also create sub-elements for axial symmetric contours (cylinder and cone).

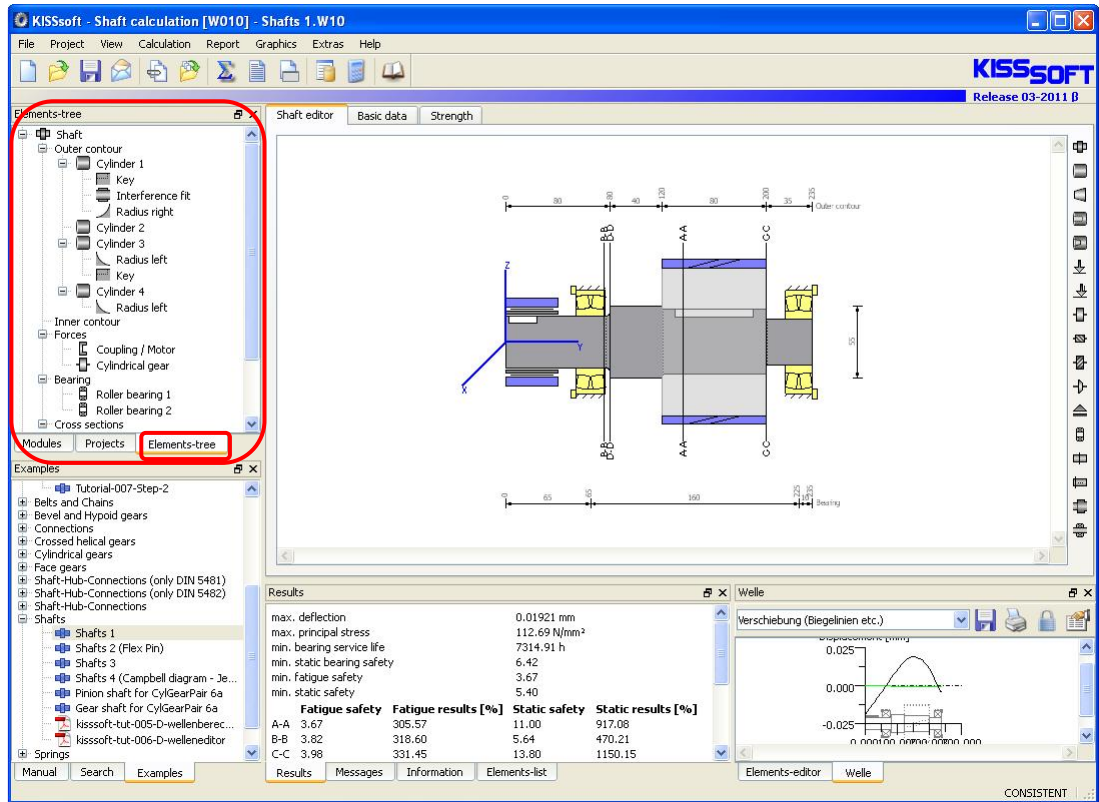
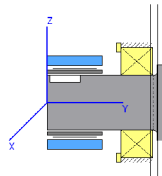


Figure 2.1 The five most important elements required to create a shaft

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	$\alpha_k$	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	Bearing forces, constraints	Yellow
Cross sections	Critical cross section	Effect of notch, position, geometry, surface roughness	Notch factors, strain	Black

## 2.2 Coordinate system

Note that the coordinate system is right-handed and Cartesian.



Positive x-axis : out of the screen

Positive y-axis: from left to right, in direction of shaft axis

Positive z-axis: from bottom to top

Position of application: angle between positive x-axis and towards positive z-axis

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

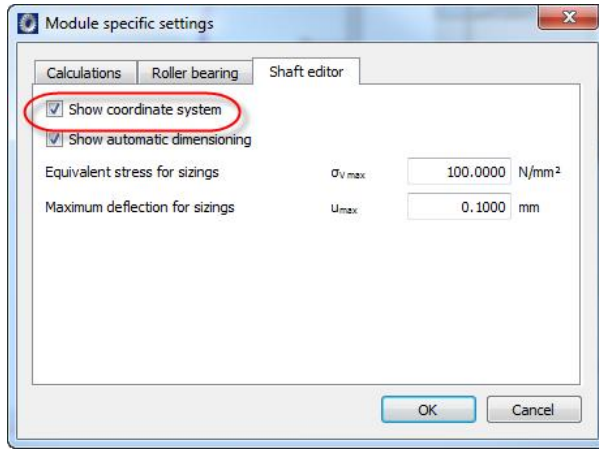


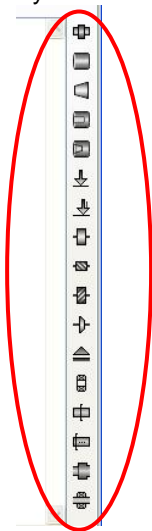
Figure 2.2 Shaft editor dialog in Module specific settings

## 2.3 Editing functions, screen view

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

Function	Description
+ key/- key/Pos1 key	Zoom in/zoom out/full screen
Left-hand mouse button	Select element/move element
Right-hand mouse button	Zoom in/zoom out/full screen
Delete	Delete selected element

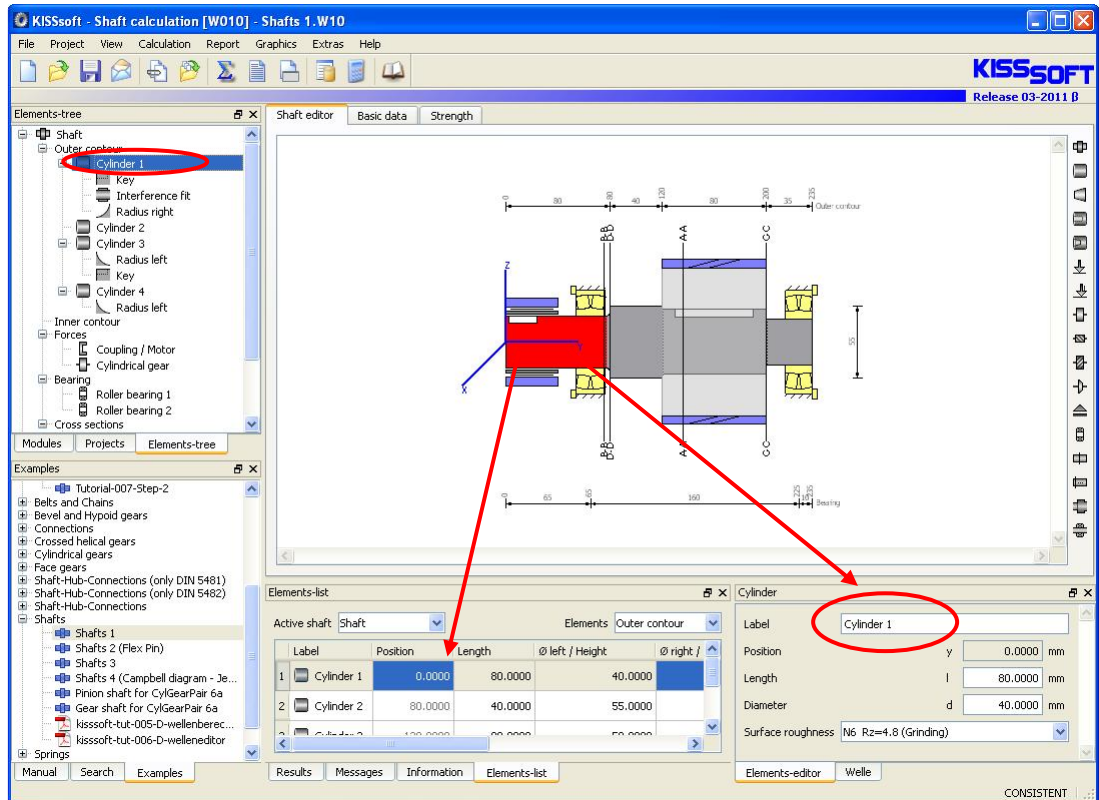
On the right-hand side of the Editor, you see the following icons which you can use to input elements more easily:



- Add shaft
- Add cylinder
- Add bevel
- 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

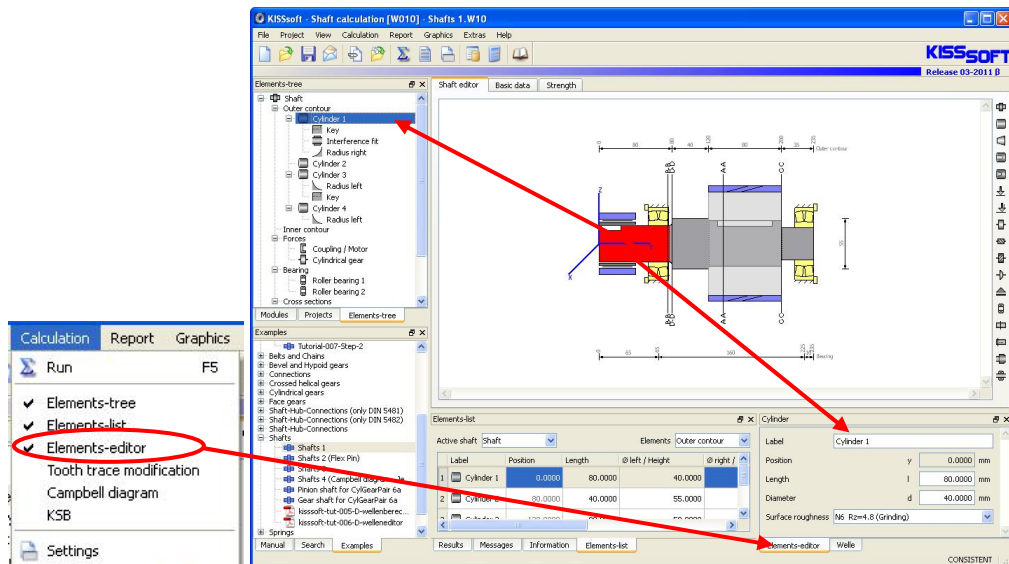
## 2.4 Entering main dimensions

To define a shaft section, select the element in the **"Elements-tree → Outer contour"** or select the corresponding icons from the vertical tool bar, for example, for **"Cylinder"** and the appropriate dialog opens in the Elements-editor (Figure 2.3).



**Figure 2.3** 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 **"Add an element in front"** 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 this is not available in the display window, click **"Calculation/Elements-editor"** to display it (Figure 2.4).



**Figure 2.4** Dialog for displaying the Elements-editor

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

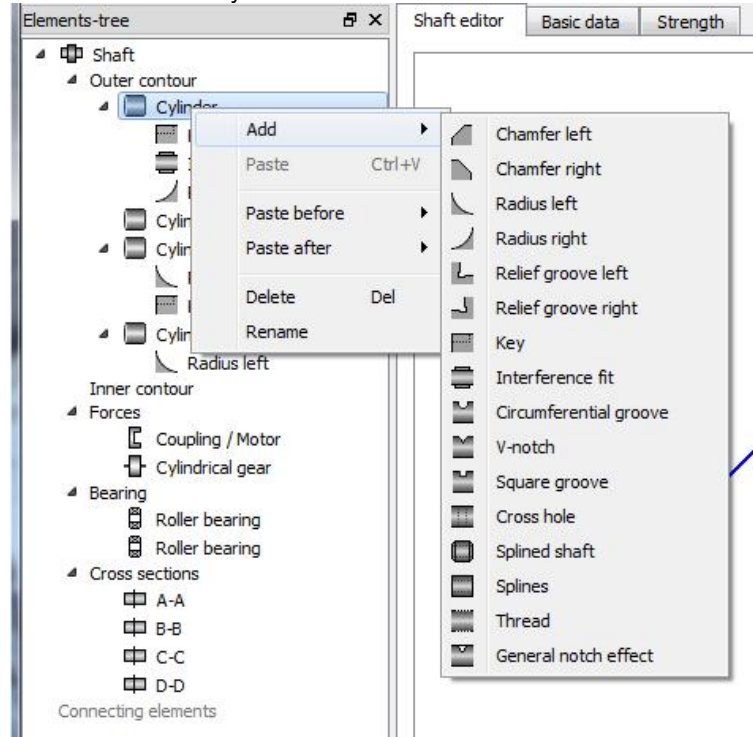


Figure 2.5 Dialog for defining sub-elements

## 2.6 Inputting loads

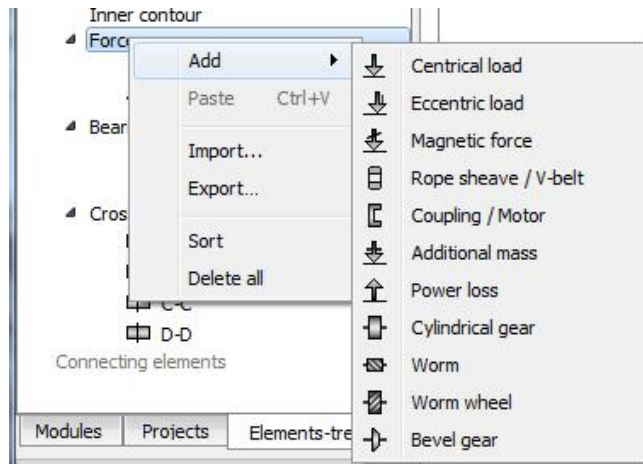
### 2.6.1 General forces

Select **"Forces→/Central 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 midline 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

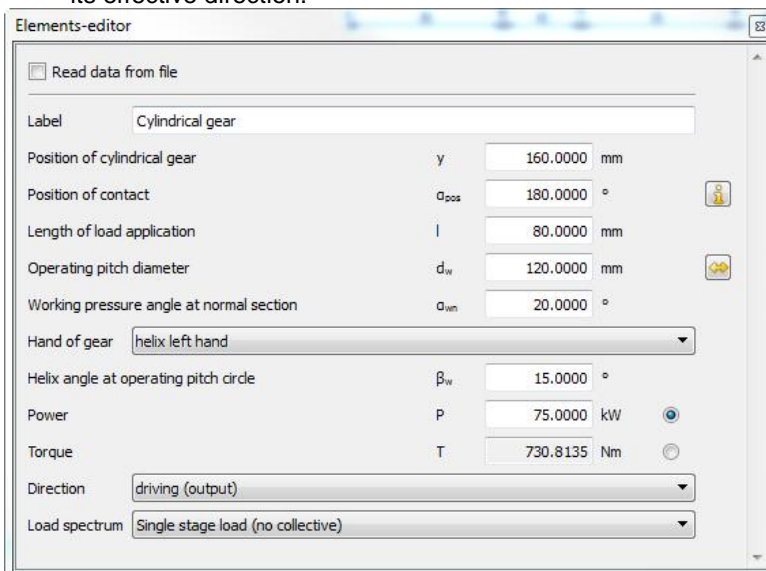
In addition to allowing the input of general force vectors, KISSsoft also provides a range of predefined machine elements. Here, the resulting forces on the shaft are determined from the geometry of the machine elements and the predefined power. As you no longer need to convert, for example, the helix angle and reference diameter manually, the load data you input is significantly less prone to errors.

In **"Forces"**, select a machine element (Figure 2.6). The dialog shown in Figure 2.7 then appears.



**Figure 2.6** 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 machine element

### "Cylindrical gear"

Direction: (power flow)

Driving: the shaft drives a system. Power is drawn from the machine element.

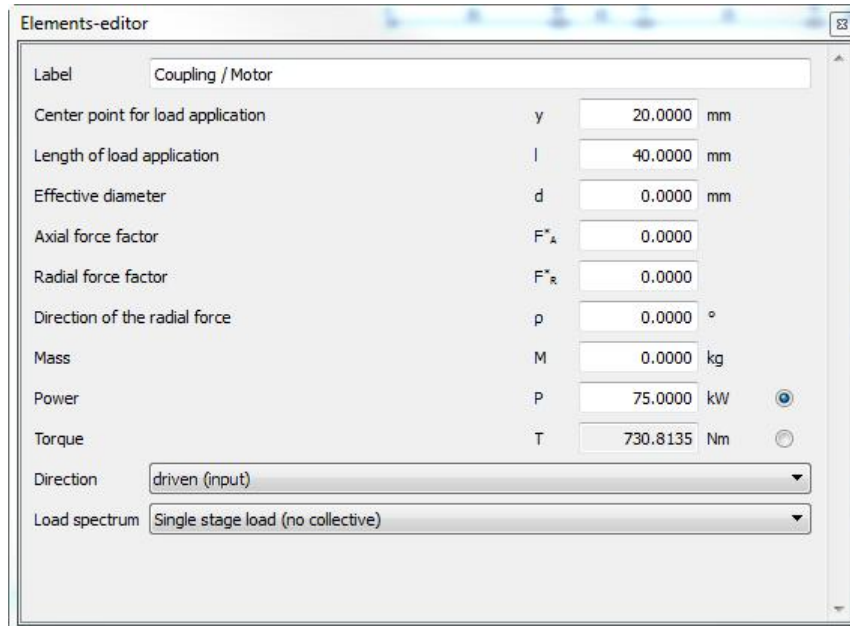
Driven: the shaft is driven from outside. Power is given to the machine element.

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 2.7** Definition of a machine element, using the example of a cylindrical gear

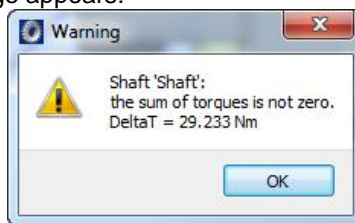
Two machine elements are used in this example: one coupling/motor (Figure 2.8) and a cylindrical gear (Figure 2.7).



**Figure 2.8** Defining 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 2.9** Warning message to tell you the power balance in the system is not equal

## 2.7 Adding bearings

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

### 2.7.1 General bearings

You define a bearing in the Elements-tree. In the Elements-tree, click on "Bearings" and then right-hand mouse click on **"Bearing (in general)"** 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 end of the shaft). In the next step, enter the bearing type.

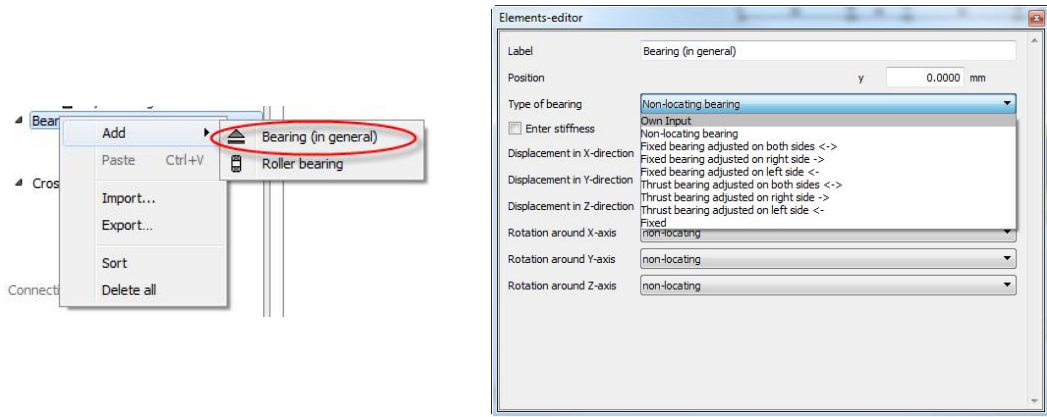


Figure 2.10 Dialog for positioning and selecting the bearing type

## 2.7.2 Input roller bearings

If you selected a roller bearing in the **"Elements-tree**, the dialog shown in Figure 2.11 appears. Alternatively, you can select the bearing in the graphic. To do this, click the **"Shaft editor"** tab. You can then define the bearing data:

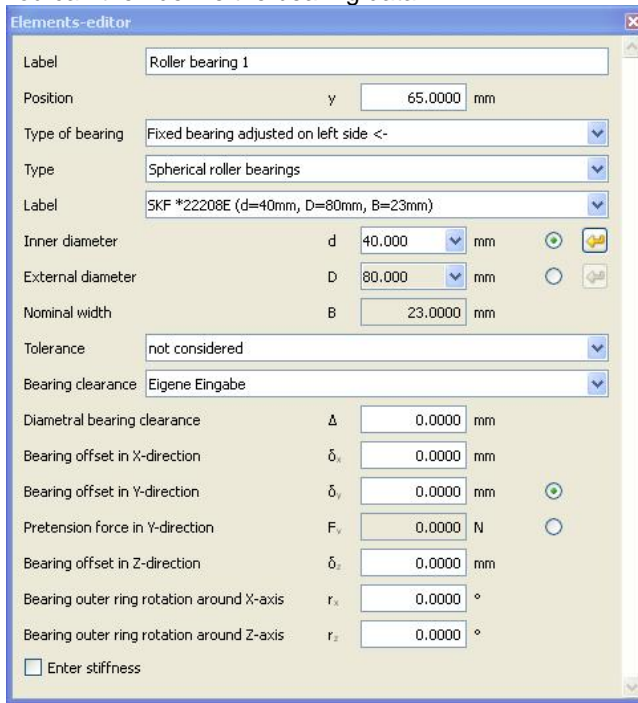


Figure 2.11 Defining a roller bearing

- Positioning bearings
- Record axial forces on bearing
- Select bearing construction
- Select bearing type
- The bearing diameter is determined automatically on the basis of the bearing position
- Select dimensional and running accuracy
- Select radial bearing clearance group
- If required, specify bearing offset
- If required, specify bearing stiffness

## 2.7.3 Special bearings

You can also input stiffness values for bearings. These are then taken into account when calculating the load and deformation, and also when calculating eigen frequencies. 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 a 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!

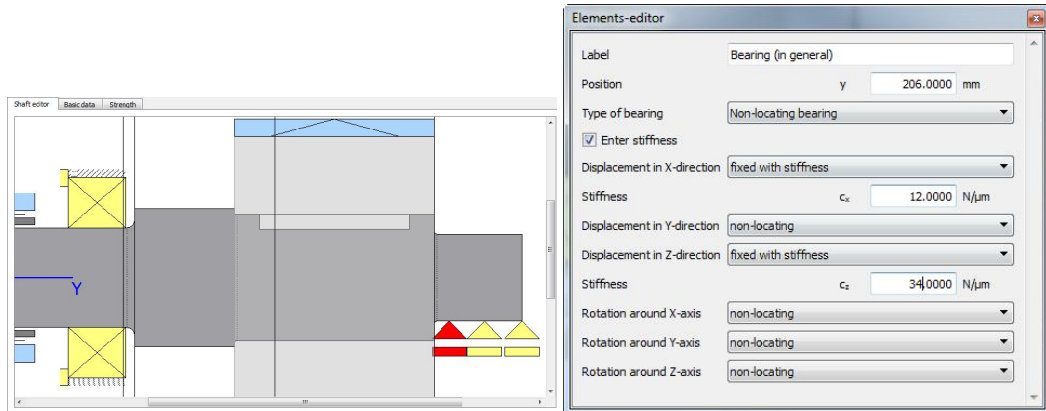


Figure 2.12 Example showing modeling of line bearings using several individual bearings

## 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 to be based (see Figure 2.13). Here you can now input the equivalent stress on which the calculation is to be based (in this example 100N/mm<sup>2</sup>).

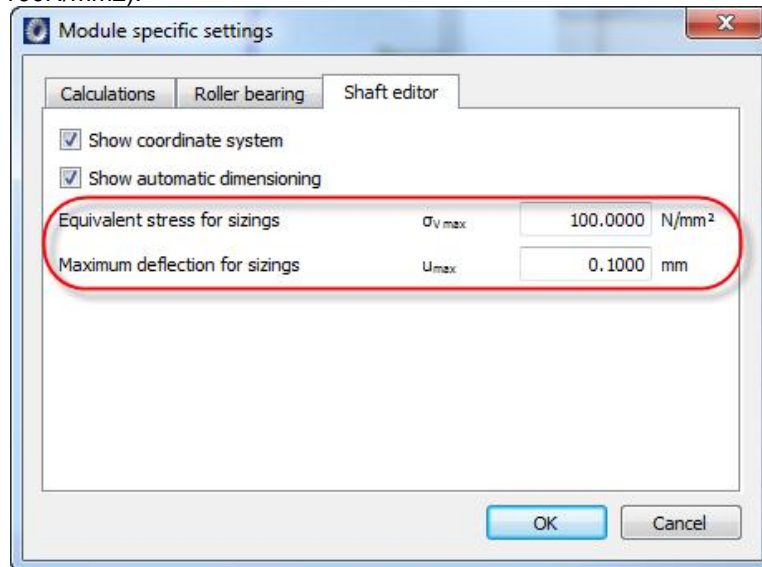
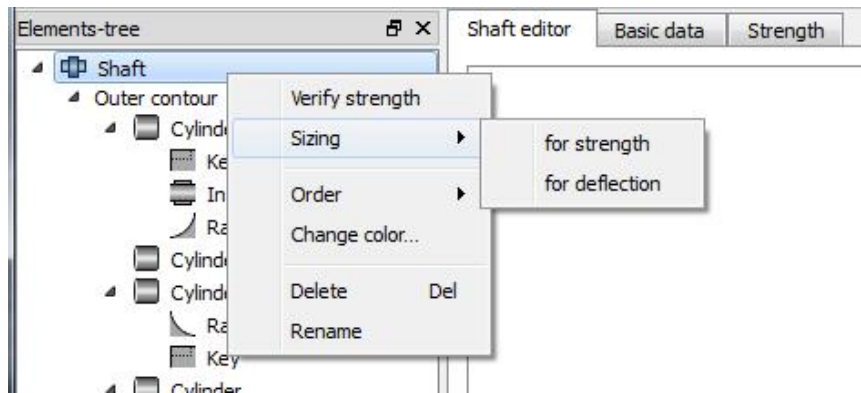


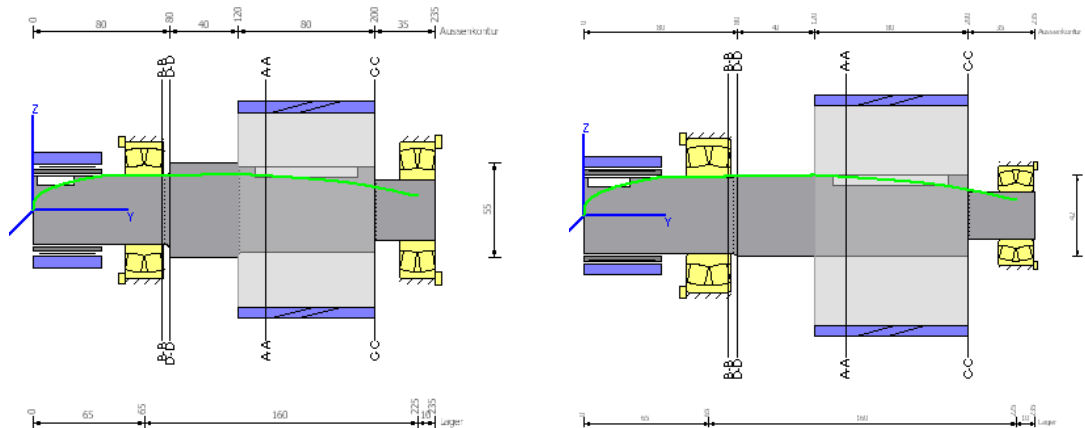
Figure 2.13 Predefining permitted values for the sizing

Now, 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 2.14** Call the "Sizing" -> "for strength" menu option by clicking on the shaft with the right mouse button

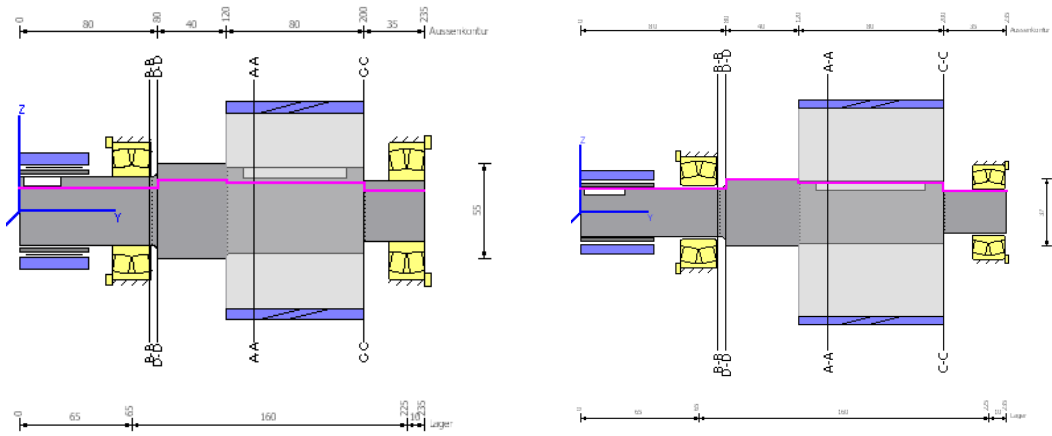
The resulting contour appears as a green line. You can now modify the shaft's dimensions insofar as is practical to suit the design to match this green line. This enables you to achieve an even load (excluding the notches) along the entire shaft.



**Figure 2.15** Original geometry with proposal for the diameter (green line), on the right: modified shaft geometry. "Sizing" -> "for strength" menu option selected

## 2.9 Sizing for maximum deflection

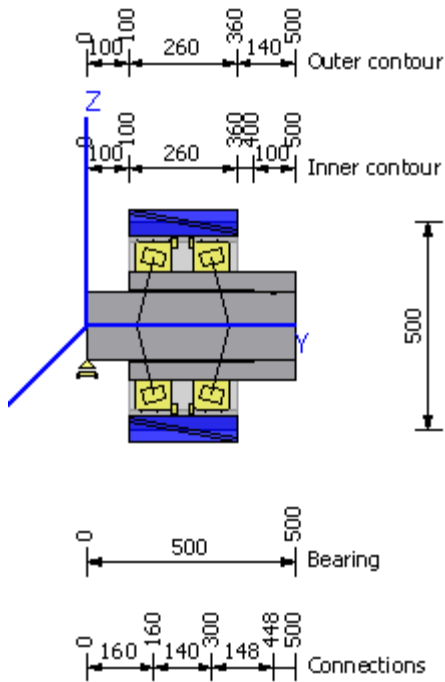
In a similar way, you can also optimize shaft geometry to ensure that the permitted deflection is not exceeded. This permitted deflection is predefined here as shown in Figure 2.13 (here 0.1 mm). After you click **"Sizing" → "for deflection"** to call the calculation, a factor is calculated by which 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 2.16** Original geometry with proposal for the diameter (magenta line), on the right: modified shaft geometry. "Sizing" -> "for deflection" menu option selected

## 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 2.17** Example Shafts 2 (Flex Pin)