MESYS AG
Technoparkstrasse 1
CH–8005 Zürich
info@mesys.ch
T: +41 44 455 68 00

Tutorial Series — Shaft Calculation - Starter Scope
3D Elastic Parts – Steering Knuckle

Contents

Using a steering knuckle as an example, this tutorial shows how to integrate 3D-elastic components into the MESYS Shaft Calculation. The workflow covers importing the 3D geometry, aligning it in the coordinate system, and defining bearing and contact faces. The components are then assembled into a complete model according to the application requirements, and the wheel-suspension load cases are set up. Along the way, useful simplifications are presented and plausibility checks are specified. The goal is a repeatable process that enables efficient integration of 3D parts and provides robust results for system evaluation.

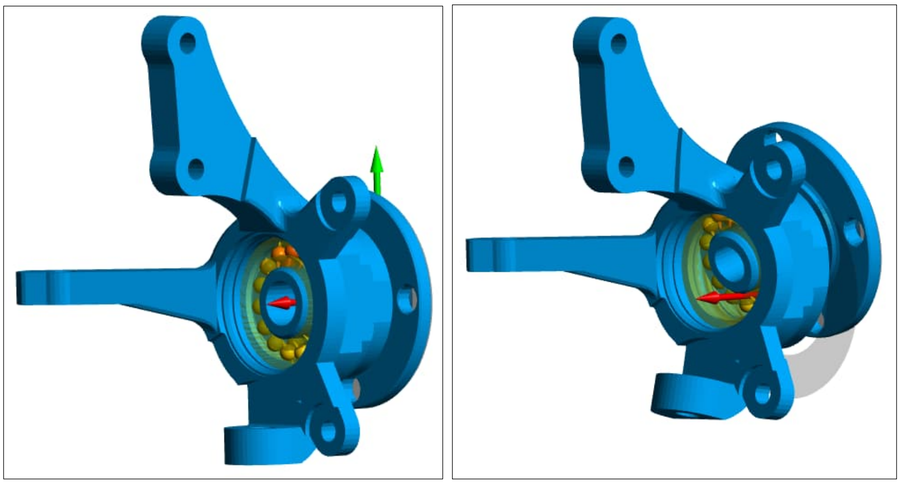


Figure 1

To follow along, you'll need the prepared 3D data and the associated calculation model; both are available in the MESYS [Downloads section](). This tutorial uses MESYS version 12-2025.

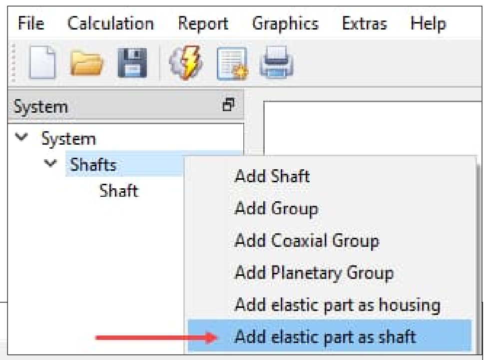
Add elastic part as shaft

Use the context menu of 'Shafts' or 'Groups' to insert a 3D-elastic component as a Shaft or Housing. Once added, the component appears immediately in the Shaft/Group tree, where you can freely adjust its name, orientation, and properties.

➡ In the MESYS Shaft Calculation, create a new model.

➡ Add an 'Elastic shaft' as shown in Figure 2.

Figure 2



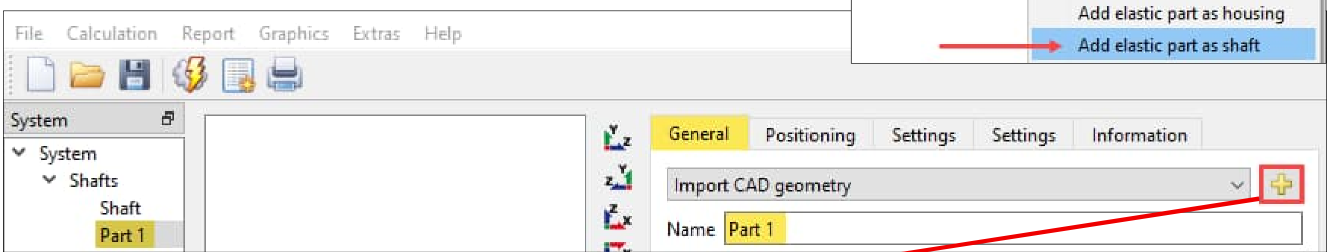


Figure 3

➡ Click the ✚ button to import the STEP file for the wheel-bearing spindle (see Figures 3/4).

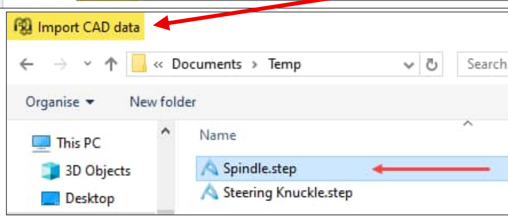
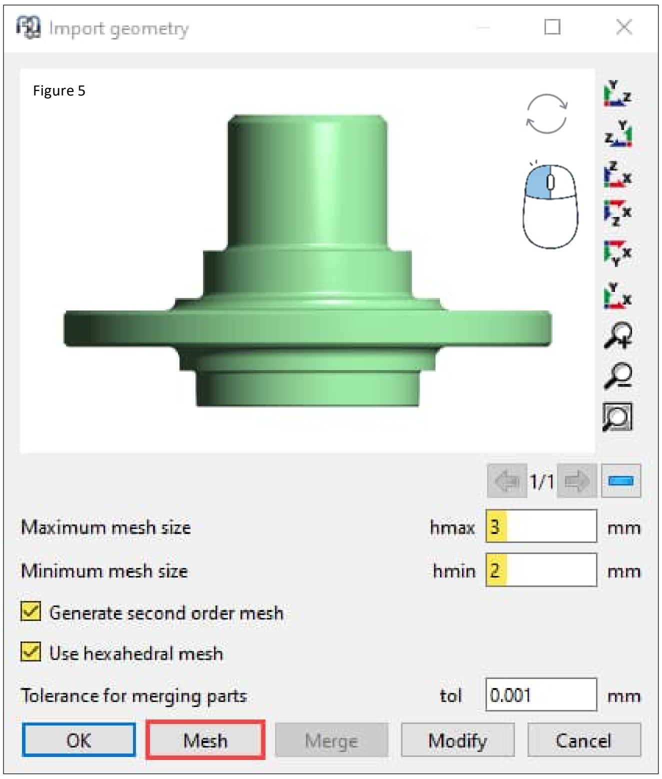
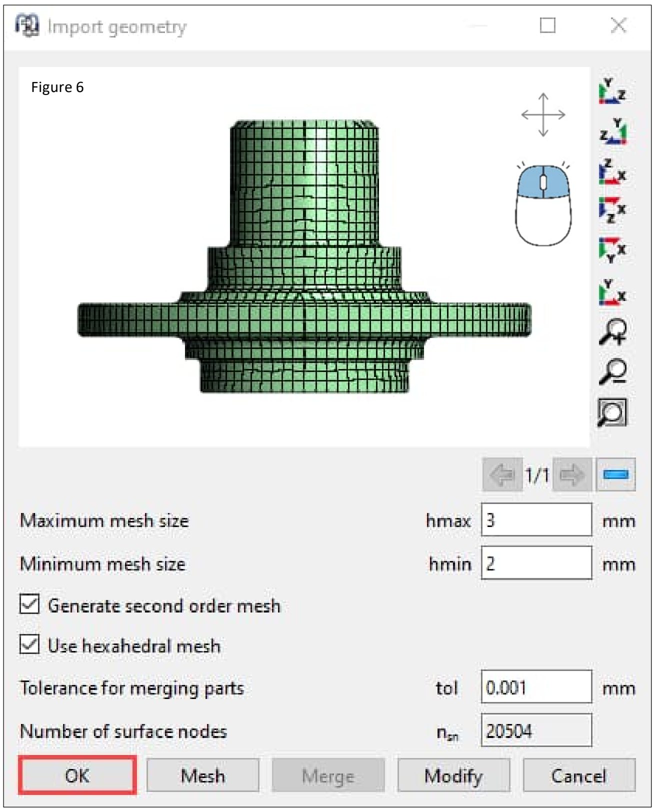


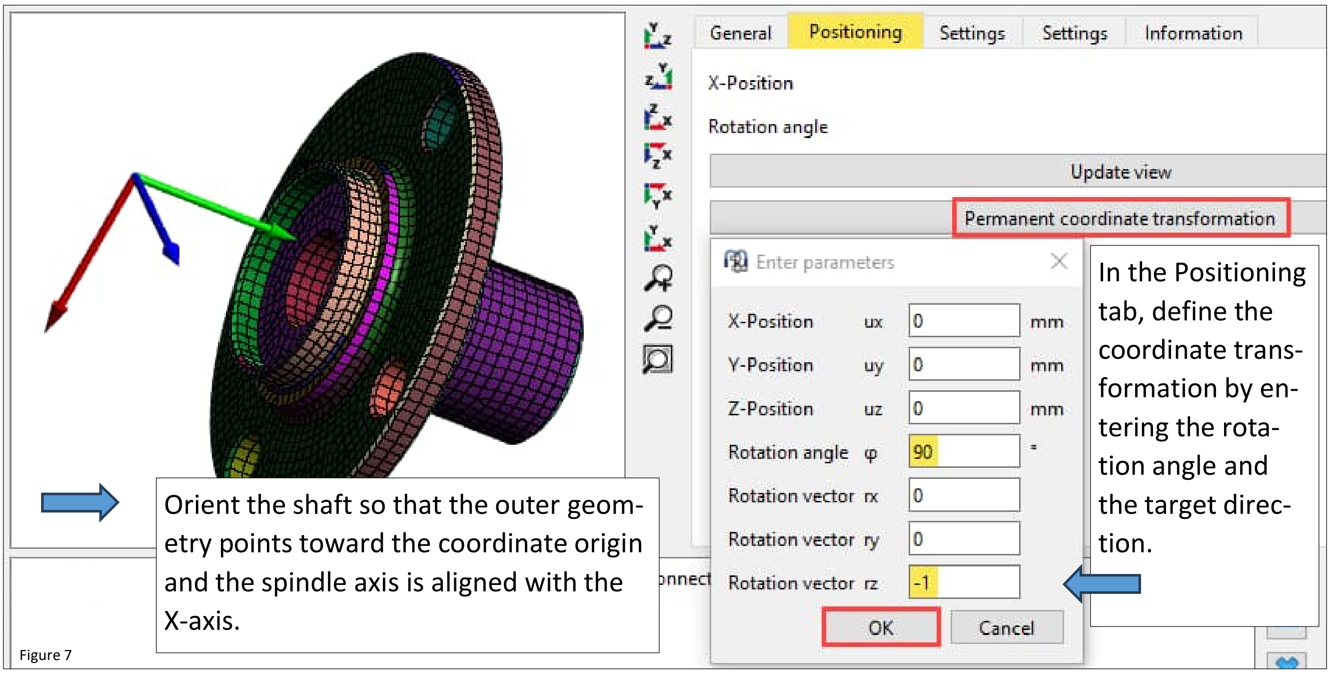
Figure 4





Enable 'Generate second order mesh' to improve geometric fidelity using quadratic shape functions (see Figure 5).

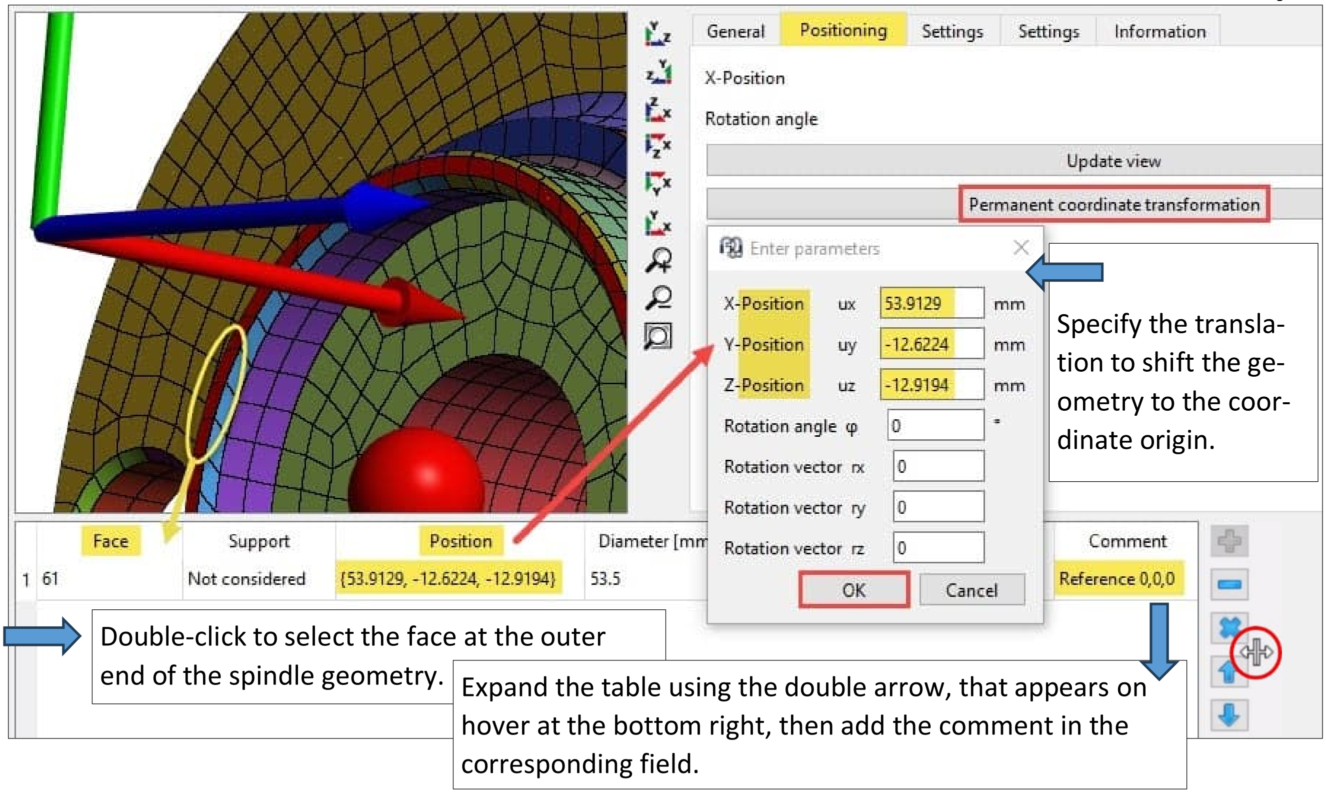
Set reasonable mesh sizes and activate meshing to generate the corresponding reduced stiffness matrix (Figures 5/6).



The next step is to activate a reference plane in 3D space and place it at the coordinate origin.

MESYS AG
Technoparkstrasse 1
CH–8005 Zürich
info@mesys.ch
T: +41 44 455 68 00

Figure 8



Enter a name for the component (Figure 9).

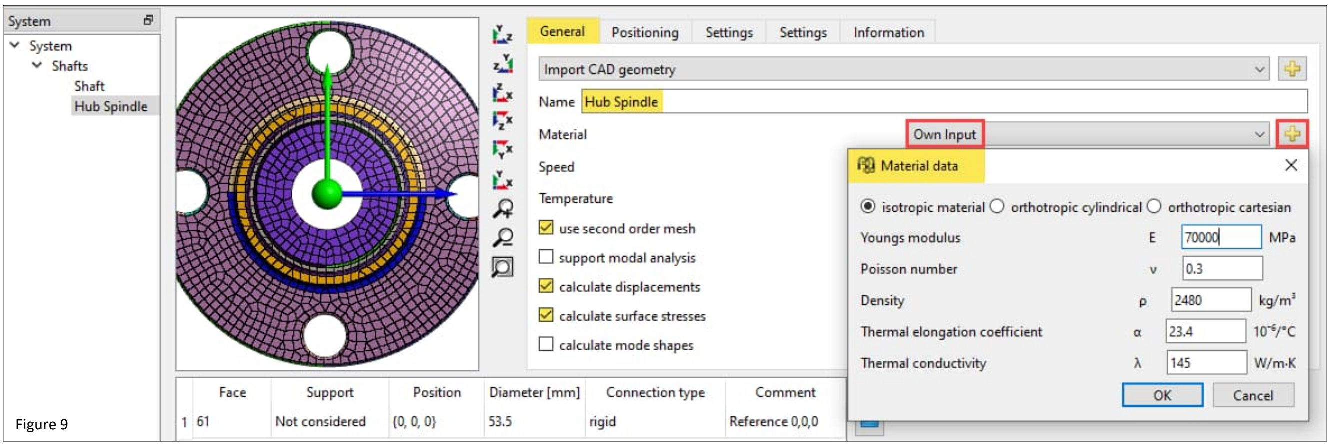


Figure 9

Select an isotropic non-ferrous alloy as the material using a custom material entry to highlight more pronounced deformation under load (Figure 9).

Enable 'second order mesh', 'Calculate displacements' and 'Calculate surface stresses' so the results are available for review under the 'Graphics' menu (Figure 9).

Additionally, in the first 'Settings' tab, enable 'Centrifugal expansion', 'Ring stiffness', and 'Ring contact' (Figure 10).

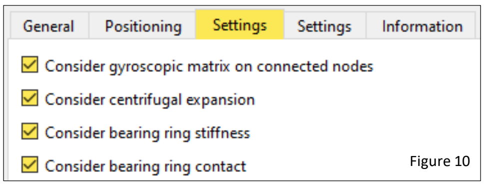


Figure 10

The option 'Consider bearing ring stiffness' adds the stiffness of the bearing rings to the elastic bearing support.

With 'Consider bearing ring contact', a contact model between the bearing ring and the component is activated. The fitting (interference) calculation will then not set the operating clearance directly; instead, the clearance results from expansion or contraction of the elastic parts. If the component stiffness is non-uniform, the bearing ring is already deformed after assembly.

Engineering Consulting Software AG
MESYS AG
Technoparkstrasse 1
CH–8005 Zürich
info@mesys.ch
T: +41 44 455 68 00

Add elastic part as housing

Use the context menu of Shafts or Groups to insert the 3D-elastic component as Elastic housing (Figure 11). After selection, the new component appears immediately in the Shaft/Group tree, where you can freely adjust its name, orientation, and properties.

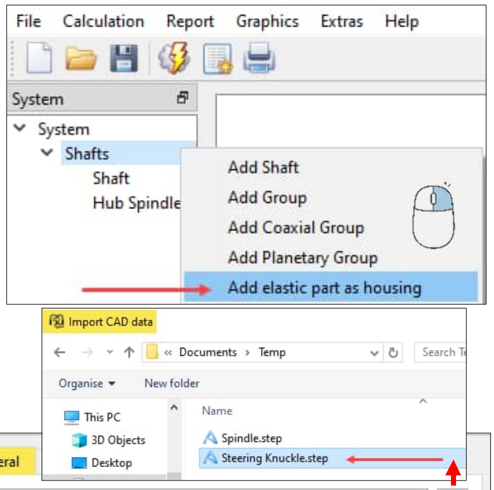


Figure 11

- Repeat the import procedure as described under [Add elastic part as shaft](), including assigning the material for the steering knuckle.
- Assign a name to the component (Figure 12).

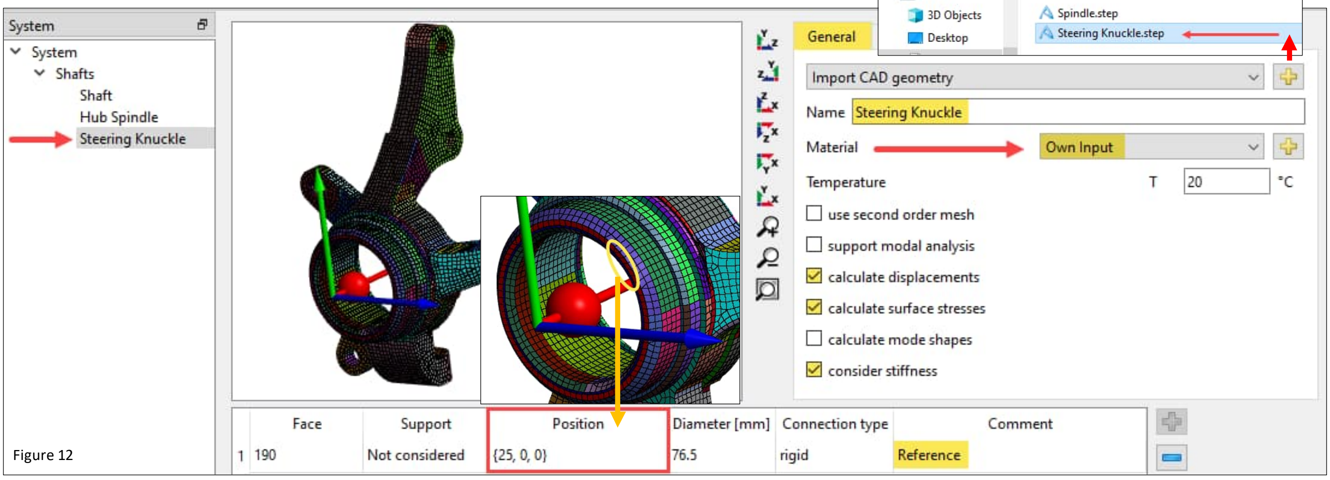


Figure 12

- Assign material properties to this component as shown in [Figure 9]().
- Enable 'calculate displacements' and 'calculate surface stresses' so results will be available under the Graphics menu (Figure 12).
- Align the component according to the coordinates shown in Figure 12 by selecting a face at the outermost end of the geometry as the positioning reference.
- Additionally, in the first Settings tab, enable 'Ring stiffness' and 'Ring contact' (Figure 13).

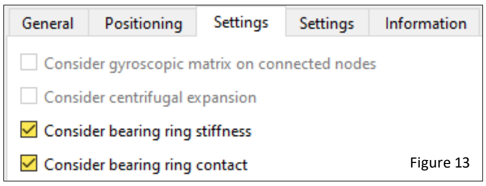


Figure 13

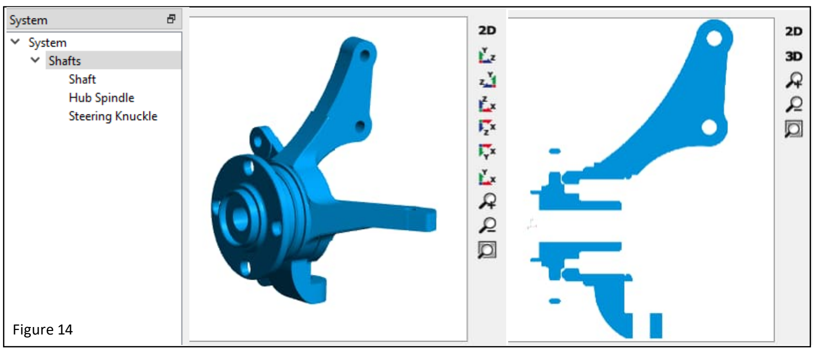


Figure 14

- This completes the import and positioning of the elastic components.
- Both components now appear in the 2D and 3D views.

MESYS AG
Technoparkstrasse 1
CH–8005 Zürich
info@mesys.ch
T: +41 44 455 68 00

Wheel bearing setup

Concept

The wheel bearing will consist of a pair of angular contact ball bearings, each 40 × 74 × 15 mm (Figure 15). In the next step, we will define the inner and outer contact faces.

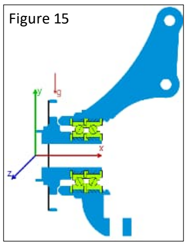


Figure 15

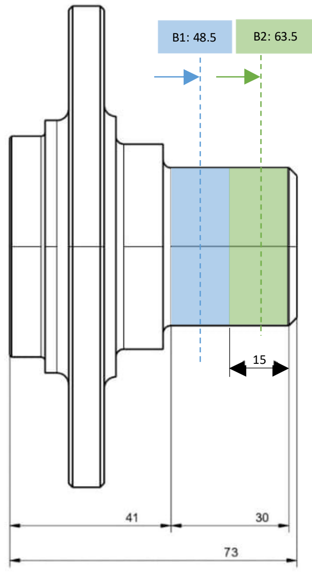


Figure 16

Spindle

The axial positions of the bearings can be taken from the dimensioned view of the wheel-bearing spindle on the right.

- In the lower pane that lists the faces for the wheel-bearing spindle, open the context menu and select 'Add face'(Figure 17).

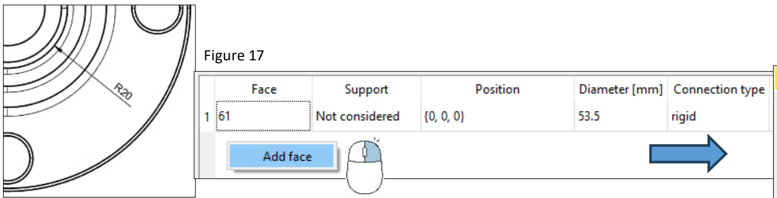


Figure 17

- Enter Position according to the axial location for the first bearing (Figure 18); set the bearing width as L and the bearing inner diameter as D.

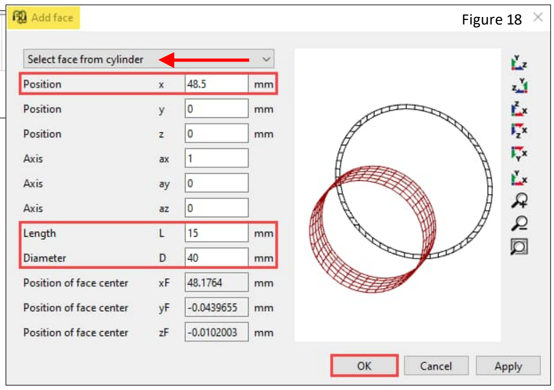


Figure 18

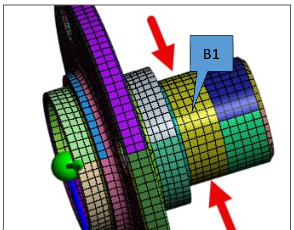


Figure 19

- This creates an integral, single face for bearing seat B1 (Figure 19).
- Repeat the face-addition step for bearing seat B2 (Figure 20).

- If the mesh density causes the two faces to have slightly different widths, select both and merge them via the context menu (Figure 20).

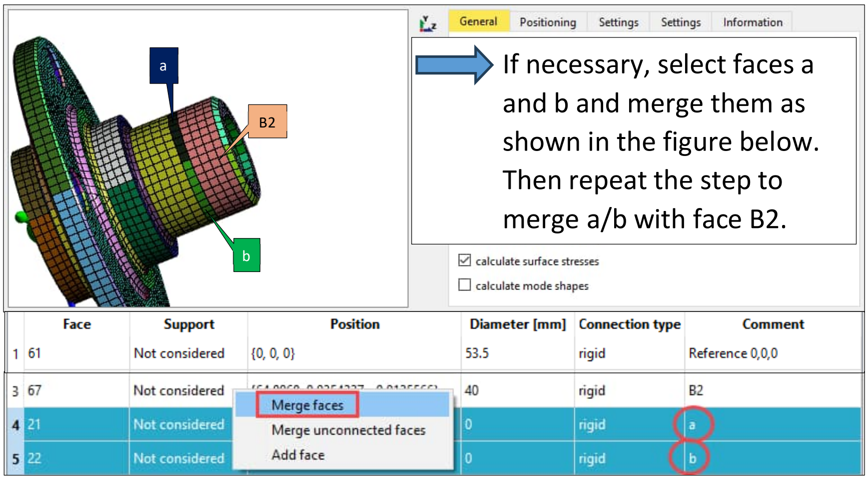


Figure 20

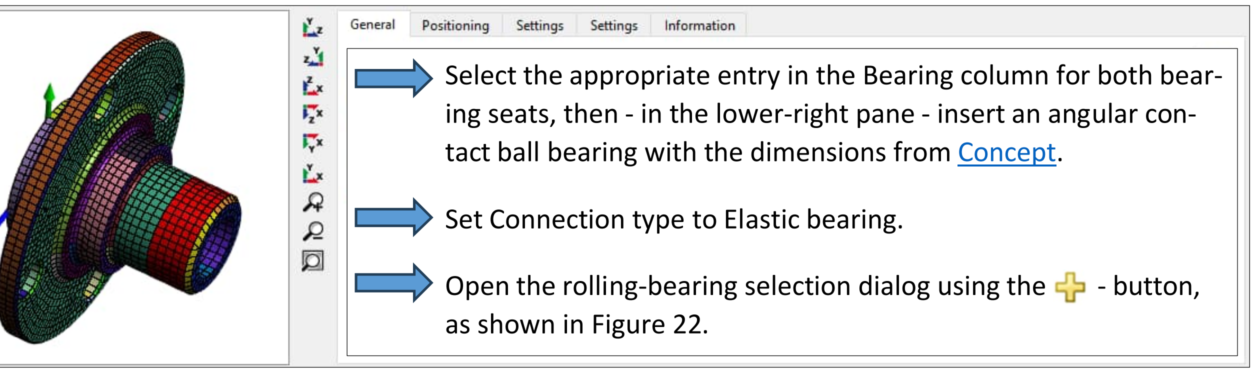
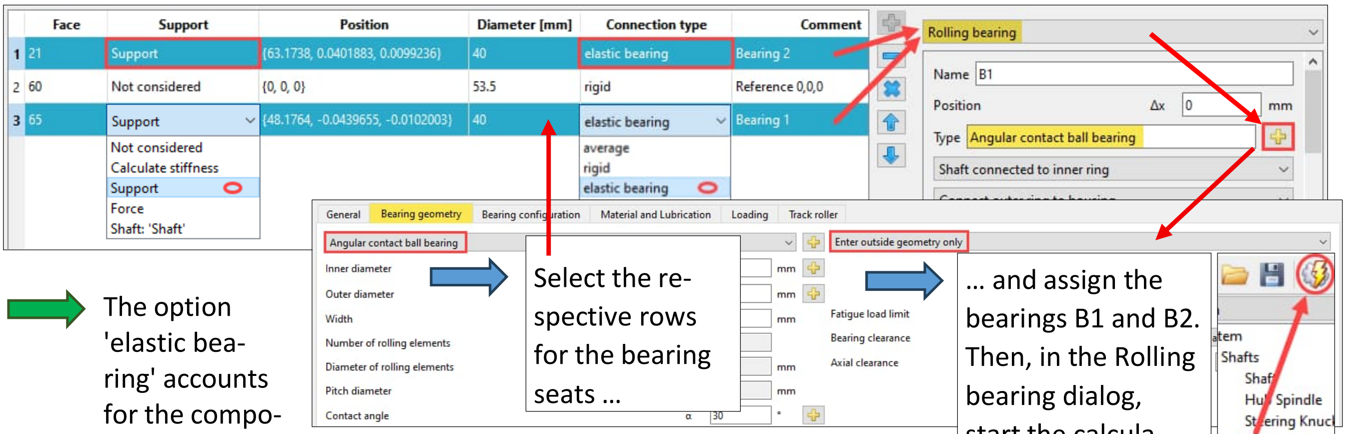


Figure 21

Figure 22



The option 'elastic bearing' accounts for the component's elastic deformations when computing the bearing's load distribution.

Steering Knuckle

In the lower pane with the tabular list of faces, open the context menu for the steering knuckle and choose 'Add face' (Figure 24). For bearing seat B1, enter x = 25 + 15.023 + (50.011 - 15.023)/4 = 48.77 mm (Figure 23).

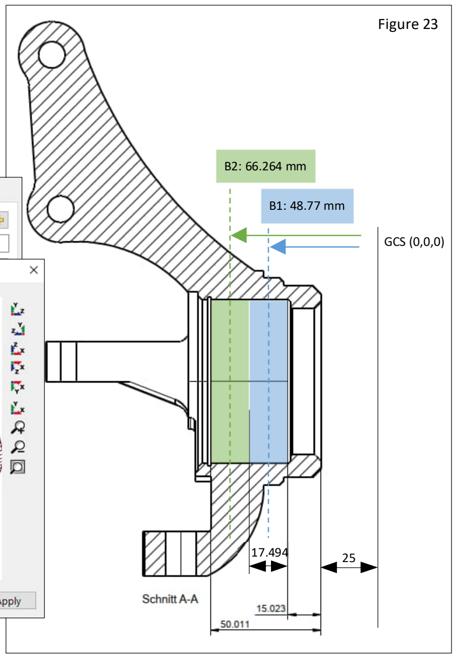


Figure 23

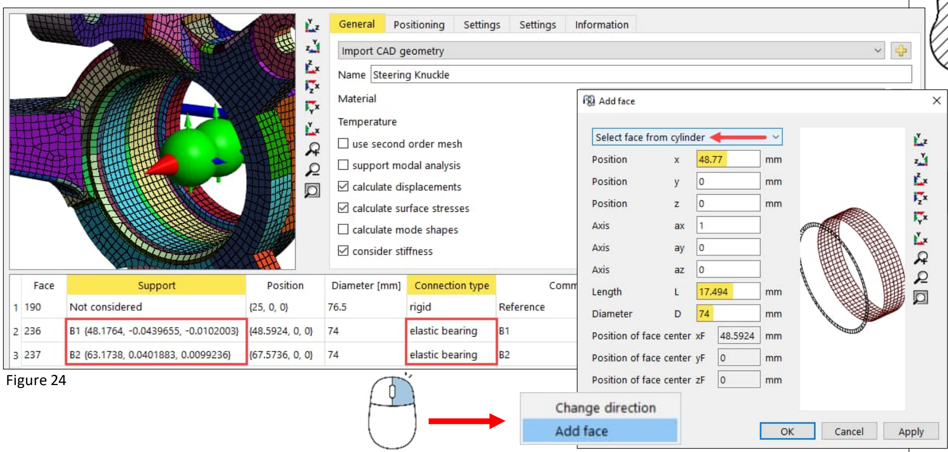


Figure 24

In the lower pane with the tabular list of faces, open the context menu for the wheel-bearing spindle and choose 'Add face' (Figure 24).
For bearing seat B2, enter *Position x = 25 + 15.023 + (50.011 − 15.023)3/4 = 66.264 mm (Figure 24).

The individual bearing-seat width of 17.494 mm is obtained as half of the total bearing-seat width: (50.011 − 15.023) / 2.

In the 'Bearing' column, assign B1 and B2 to their respective bearing-seat faces (Figure 24).

In the 'Connection type' column, set 'elastic bearing' for the bearing-seat faces of both bearings B1 and B2 (Figure 24).

In the 2D view, set the orientation of B2 to O arrangement (back-to-back) (Figur 25).

Figure 25

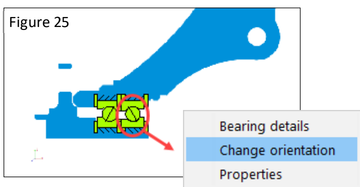
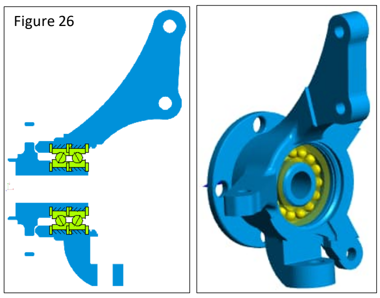


Figure 26



The rolling bearings now appear in the 2D and 3D views (Figure 26).

Bearing settings

Set Axial clearance Pa (unmounted) to 'User input', and for both bearings specify Tolerance class, Fits, Roughness, Shaft inner diameter, and Housing outer diameter (Figure 27).

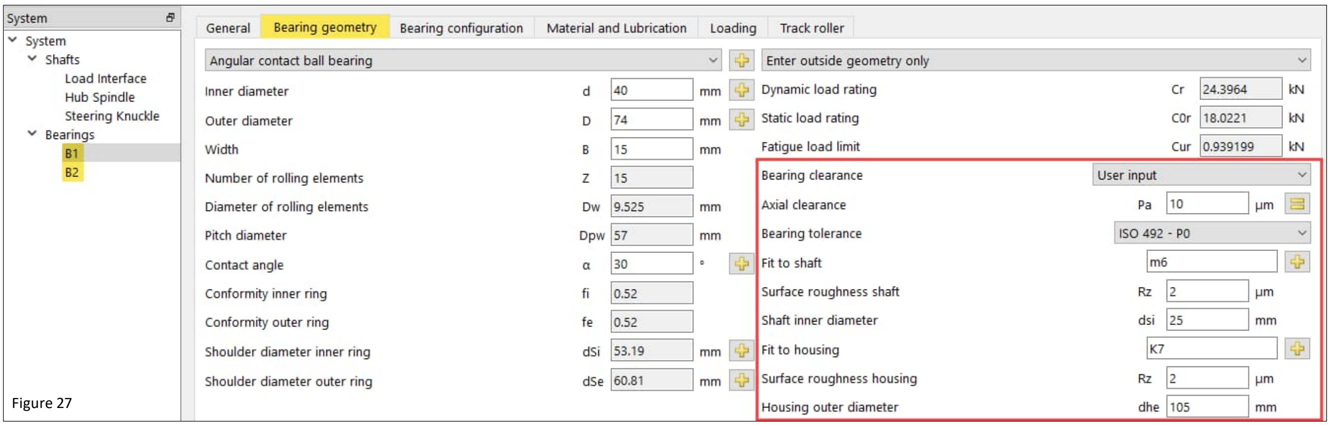


Figure 27

1D shaft coupling

A 1D shaft segment will be coupled to the wheel-bearing spindle. It is used to define an eccentric force. With the appropriate setup, the software will transmit the loads through this 6-DOF interface into the bearings and the steering knuckle in a physically consistent manner.

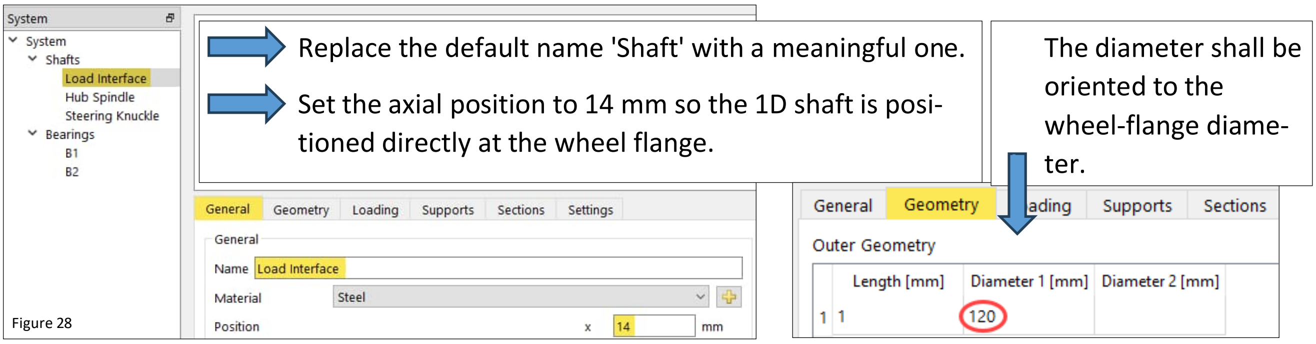


Figure 28

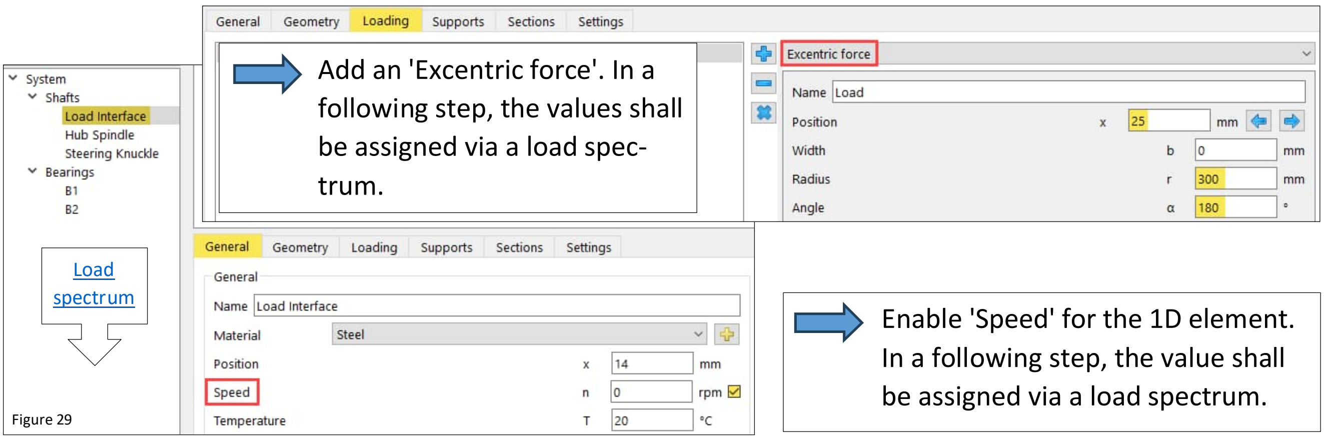
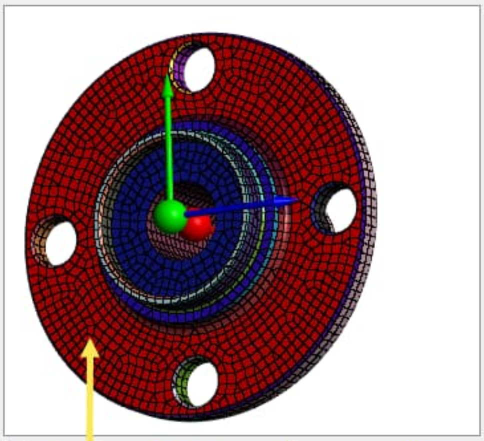


Figure 29

Figure 30

System
- System
  - Shafts
    - Load Interface
    - Hub Spindle
    - Steering Knuckle
  - Bearings
    - B1
    - B2

At this point, define via which face the connection of the 1D element (load interface) to the spindle shall be established.



General | Positioning | Settings | Settings | Information

Select '1D element' as the bearing type in the 'Support' column. Set 'average' for 'Connection type'.

Under the 'average' option, the reaction force is distributed across all nodes of the face, and the average displacement of the face is used. This prevents the component from being stiffened.

	Face	Support	Position	Diameter [mm]	Connection type	Comment
1	21	Support	{63.1738, 0.0401883, 0.0099236}	40	elastic bearing	B2
2	60	Not considered	{0, 0, 0}	53.5	rigid	Referenz 0,0,0
3	65	Support	{48.1764, -0.0439655, -0.0102003}	40	elastic bearing	B1
4	45	Shaft: 'Load Interface'	{15, 9.66269e-06, 3.24119e-05}	0	average	Load

Interfaces for coupling to the environment

As shown above, in the Bearing column you define the physical connection of the selected face to the environment, while Connection type specifies how this face is mapped to a reduced node and how forces and deformations are transferred. At this point, we shall define all remaining interfaces of the steering knuckle accordingly.

	Face	Support	Position	Diameter [mm]	Connection type	Comment
1	61	Set as fixed	Merge unconnected faces	0	average	Strut lower
2	62	Set as fixed	Add face	0	average	Strut upper

Figure 31

For each bore, select its two inner-surface faces and merge each pair, respectively.

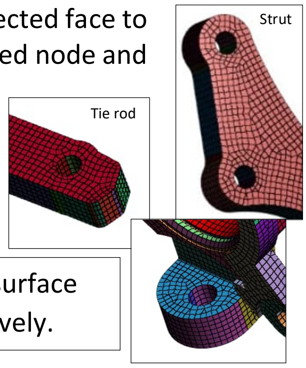


Figure 32

In the right-hand pane, for the connection to the Tie rod, activate a 'Stiffness matrix' with hypothetical content, as shown in Figure 33:

	Face	Support	Position	Diameter [mm]	Connection type	Comment
1	61	Set as fixed	{143, 78, 0}	0	average	Strut lower
2	62	Set as fixed	{147, 147.886, 0}	0	average	Strut upper
3	170	Support	{128, 9, 118}	0	average	Tie rod
4	210	Support	{88.5, -78, 0}	0	average	Control arm
5	232	B1 {48.1764, -0.0439655, -0.0102003}	{48.5924, 0, 0}	74	elastic bearing	B1
6	233	B2 {63.1738, 0.0401883, 0.0099236}	{67.5736, 0, 0}	74	elastic bearing	B2

Stiffness matrix

Name: SM Tie rod

Position Δx 0 mm

Connect to housing

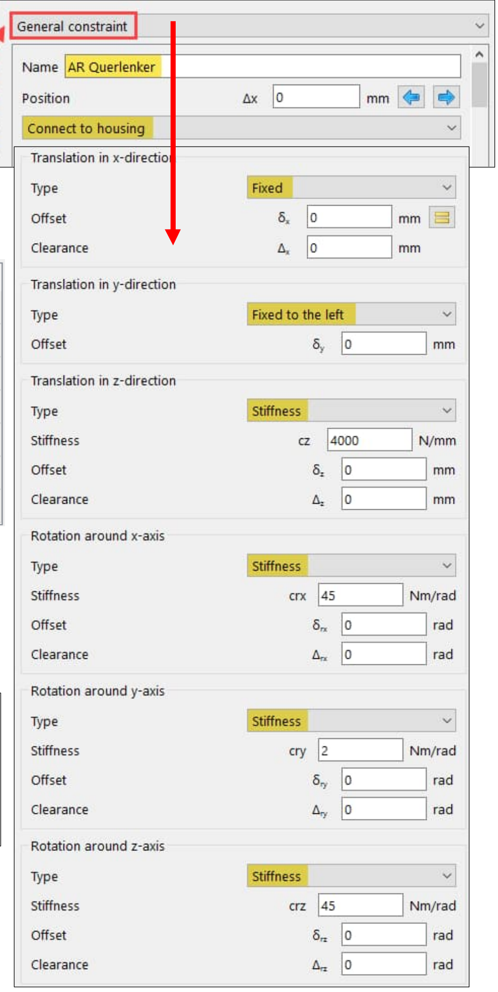
	ux [mm]	uy [mm]	uz [mm]	rx [mrad]	ry [mrad]	rz [mrad]
Fx [N]	20	0	0	0	0	0
Fy [N]		40	0	0	0	0
Fz [N]			400	0	0	0
Mx [Nm]				45	0	0
My [Nm]					2	0
Mz [Nm]						45

Figure 33

In the right-hand pane, for the connection to the control arm, activate 'General constraint' as shown in Figure 34:

Figure 34

	Face	Support	Position	Diameter [mm]	Connection type	Comment
1	61	Set as fixed	{143, 78, 0}	0	average	Strut lower
2	62	Set as fixed	{147, 147.886, 0}	0	average	Strut upper
3	170	Support	{128, 9, 118}	0	average	Tie rod
4	210	Support	{88.5, -78, 0}	0	average	Control arm



Your matrix should then appear as follows:

Figure 35

	Face	Support	Position	Diameter [mm]	Connection type	Comment
1	61	Set as fixed	{143, 78, 0}	0	average	Strut lower
2	62	Set as fixed	{147, 147.886, 0}	0	average	Strut upper
3	170	Support	{128, 9, 118}	0	average	Tie rod
4	187 X	Not considered X	{25, 0, 0} X	76.5 X	rigid X	Reference X
5	210	Support	{88.5, -78, 0}	0	average	Control arm
6	232	B1 {48.1764, -0.0439655, -0.0102003}	{48.5924, 0, 0}	74	elastic bearing	B1
7	233	B2 {63.1738, 0.0401883, 0.0099236}	{67.5736, 0, 0}	74	elastic bearing	B2

Delete face 187, which served as the reference for positioning the knuckle (Figure 35).

Figure 36

	Face	Support	Position	Diameter [mm]	Connection type	Comment
1	21	Support	{63.1738, 0.0401883, 0.0099236}	40	elastic bearing	B2
2	45	Shaft: 'Load Interface'	{15, 9.66269e-06, 3.24119e-05}	0	average	Last
3	60 X	Not considered X	{0, 0, 0} X	53.5 X	rigid X	Reference 0,0,0 X
4	65	Support	{48.1764, -0.0439655, -0.0102003}	40	elastic bearing	B1

Also delete face 60 on the spindle, which served as the reference for positioning the spindle (Figure 36).

Now start the calculation:

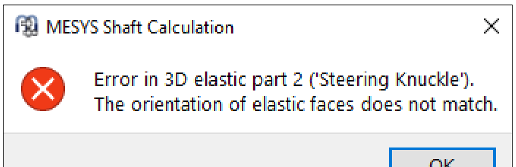


Figure 37

If in the following step you receive the error message shown on the left, the local coordinates for the outer bearing seats must be corrected.

Reverse the direction as shown in Figure 38.

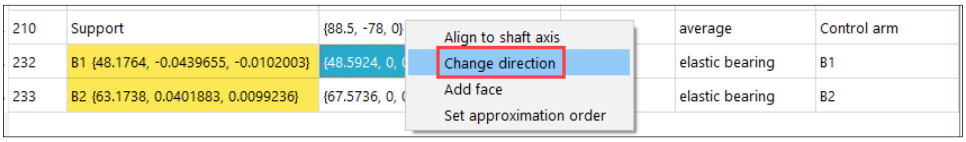


Figure 38

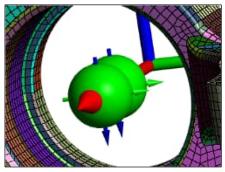


Fig. 39: Wrong

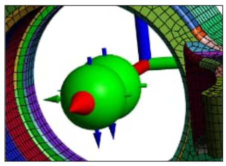


Fig. 40: Right

The calculation step should now complete without any error message, and as confirmation the corresponding icon indicating a successful calculation should appear in the lower-right corner.

Load Spectrum

Enable the option Consider load spectrum.

☑ Consider load spectrum

The simple, hypothetical load spectrum shown alongside shall be set up; in a following step, the results shall be analyzed.

Optionally, use 'Results element'.

Figure 41

	Comment	Frequency	n [rpm]	Fx [N]	Fr [N]	Ft [N]
Shaft			Load Interface	Load Interface	Load Interface	Load Interface
Element			General	Belastung	Belastung	Belastung
1	Driving	0.75	815	0	-4500	0
2	Cornering	0.25	815	200	-5130	0

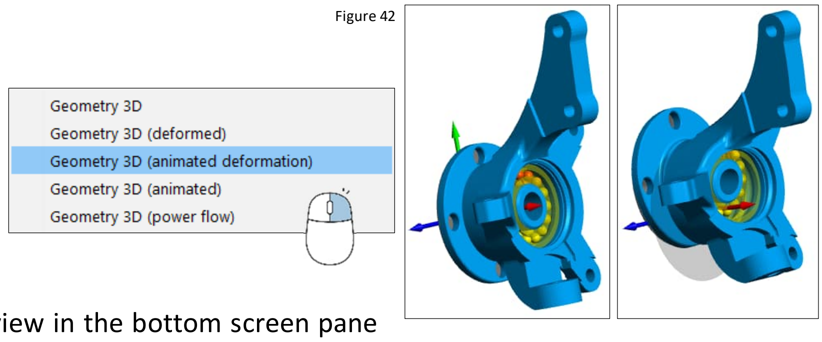
☑ Run calculation for result element only — Result element: 2

Results

Main Window

In the 3D view of the main window, you can display an animated deformation via the context menu (see right-hand figure).

Figure 42



Results Overview

The contents of the Results overview in the bottom screen pane can be edited via Extras => Results overview.

Graphics Menu

In addition to the usual graphical outputs for analysis, a range of 3D visualizations is available:

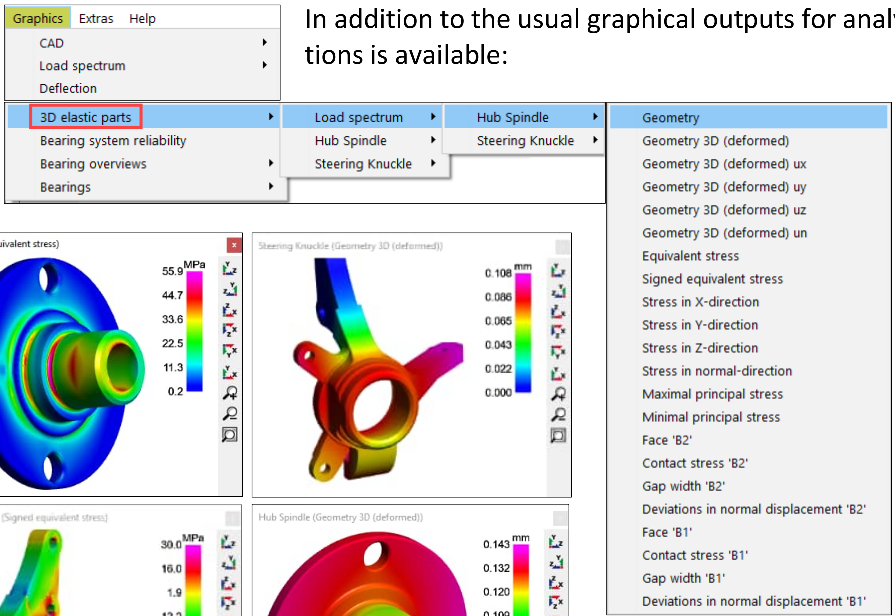
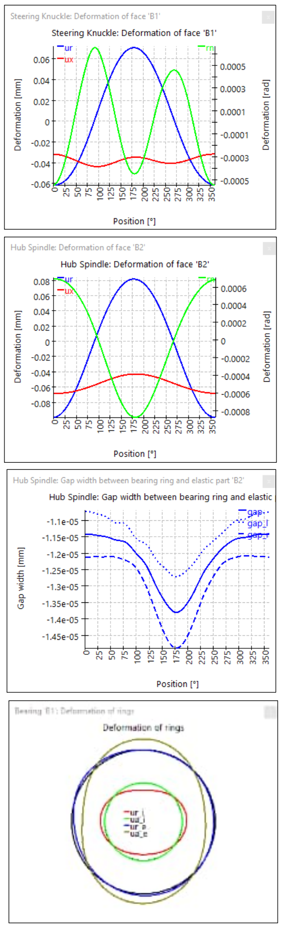


Figure 43



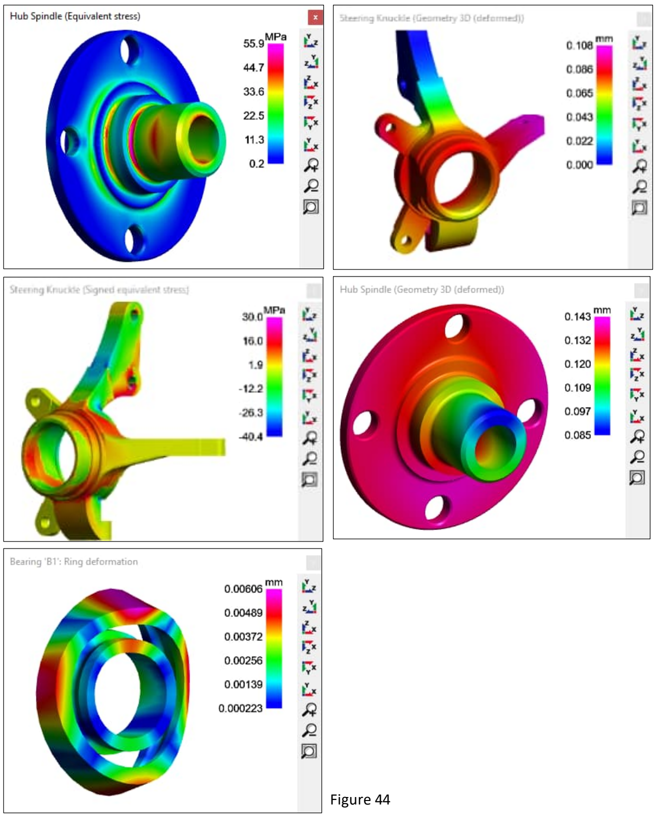


Figure 44

MESYS wishes you an instructive and profitable experience with our tutorials. If you have any queries, suggestions or questions, please do not hesitate to contact info@mesys.ch .