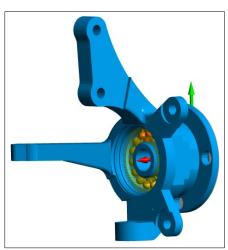


Tutorial Series

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.





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 12-2024.

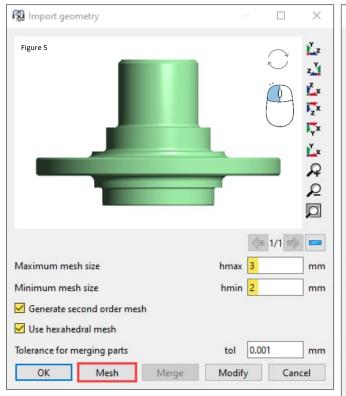
Figure 1

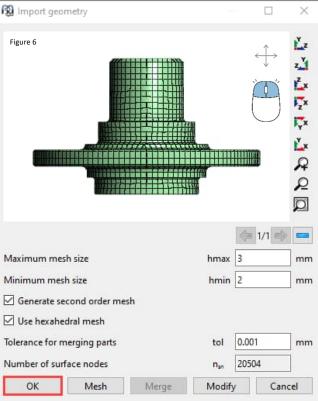
A Steering Knuckle.step

Add elastic part as shaft Figure 2 Use the context menu of 'Shafts' or 'Groups' to insert a 3D-elastic Calculation Graphics component as a Shaft or Housing. Once added, the component appears immediately in the Shaft/Group tree, where you can freely ad-System just its name, orientation, and properties. System Shafts Add Shaft In the MESYS Shaft Calculation, create a new model. Shaft Add Group Add Coaxial Group Add an 'Elastic shaft' as shown in Figure 2. Add Planetary Group Add elastic part as housing File Calculation Report Graphics Extras Add elastic part as shaft System Positioning Ľz Settings Settings Information System z.Y Shafts Import CAD geometry Shaft Z_x Name Part 1 M Import CAD data Figure 3 → ✓ ↑ 🖟 « Documents > Temp Click the 💠 button to import the STEP Organise file for the wheel-bearing spindle (see This PC A Spindle.step 3D Objects

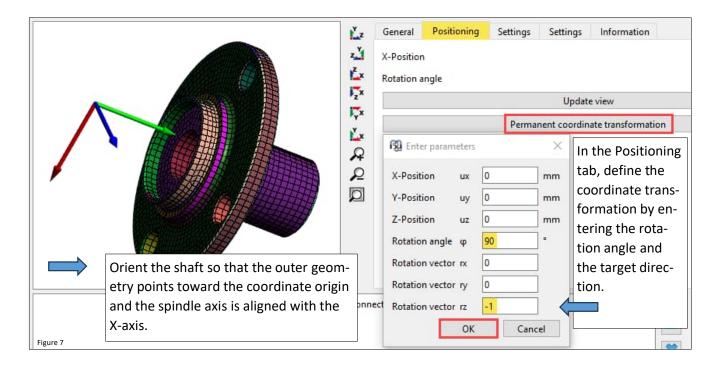
Figures 3/4).

Figure 4

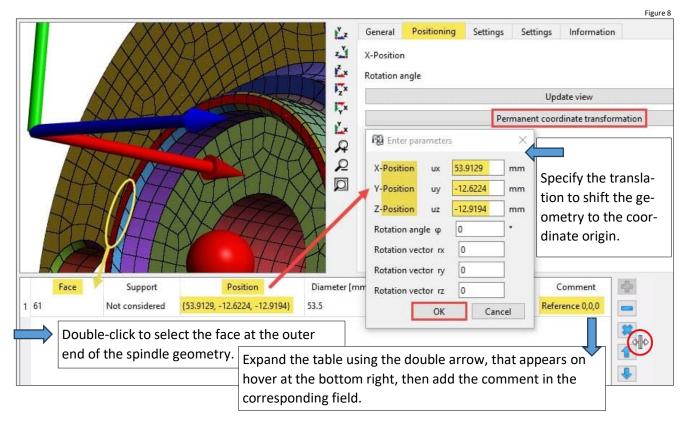




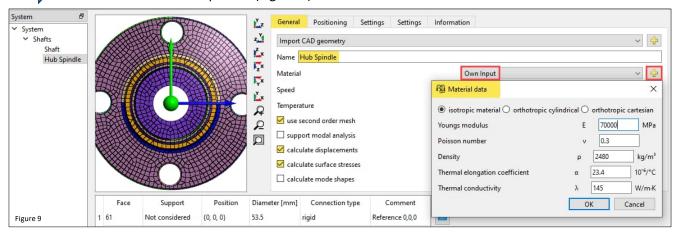
- Enable 'Generate second order mesh' to improve geometric fidelity using quadratic shape functions (see Figure 5).
- Set reasonable mesh sizes and activate meshing to generate the corresponding reduced stiffness matrix (Figures 5/6).



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

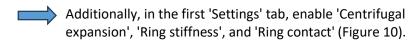


Enter a name for the component (Figure 9).



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).





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.

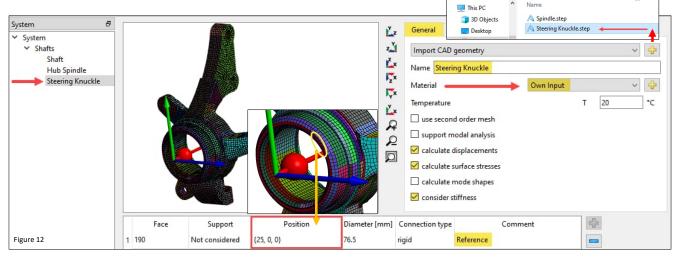


Add elastic part as housing

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.

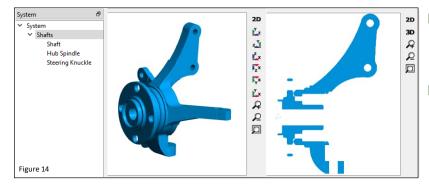
Repeat the import procedure as described under Add elastic part as shaft, including assigning the material for the steering knuckle.

Assign a name to the component (Figure 12).



- 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).





This completes the import and positioning of the elastic components.

Both components now appear in the 2D and 3D views.

Figure 11

Help

Report

Graphics

Add Shaft

Add Group

Add Coaxial Group

Add Planetary Group

Add elastic part as housing

Extras

Calculation

Shaft

Hub Spindle

~ ↑ 🔠 « Do

System

System

Shafts



Wheel bearing setup Concept

The wheel bearing will consist of a pair of angular contact ball bearings, each $40 \times 74 \times 15$ mm (Figure 15). In the next step, we

will define the inner and outer contact faces. **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).

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.



Axis

Length

Diamete Position of face cente

Position of face cente

Figure 15

Figure 16

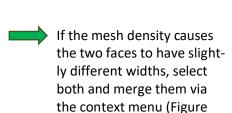
xF 48.1764 -0.0439655

mm

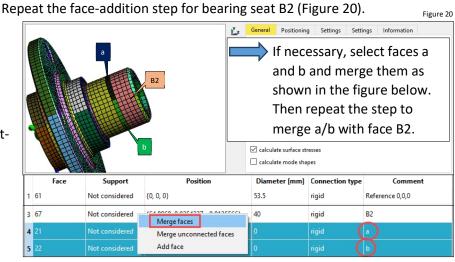
Figure 18

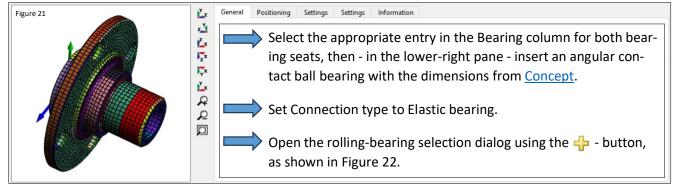
Y,z

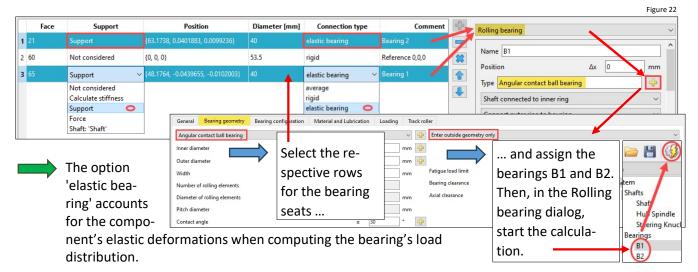
いるでなるでい

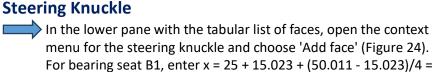


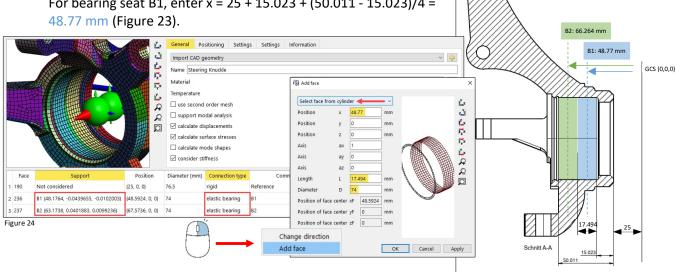
20).











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).

Figure 25

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

Bearing details

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

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

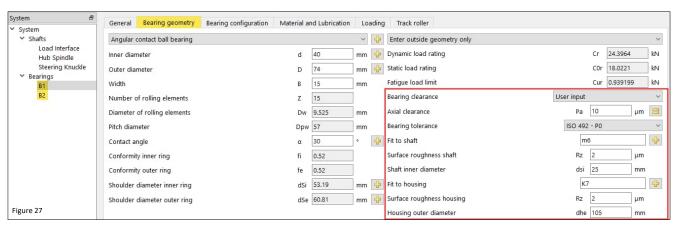
Figure 23



Bearing settings

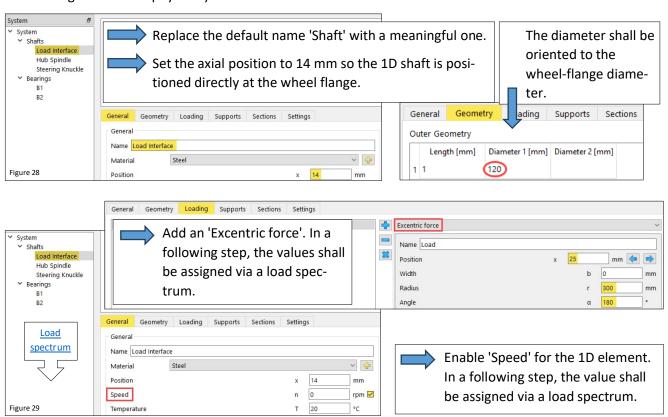
Set Axial clearance Pa (unmounted) to 'User input', and for both bearings specify Tolerance class, Fits, Roughness, Shaft inner diameter, and Housing outer diameter (Figure 27).

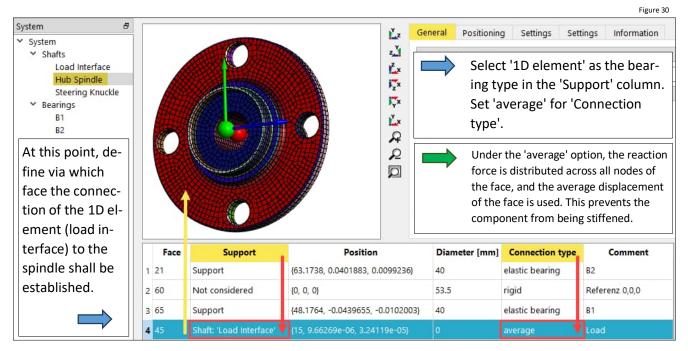




1D shaft coupling

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.





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

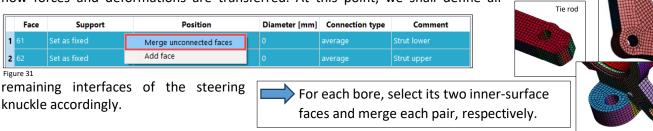
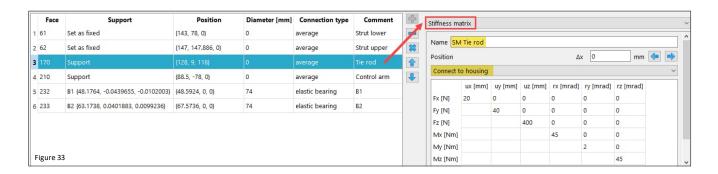
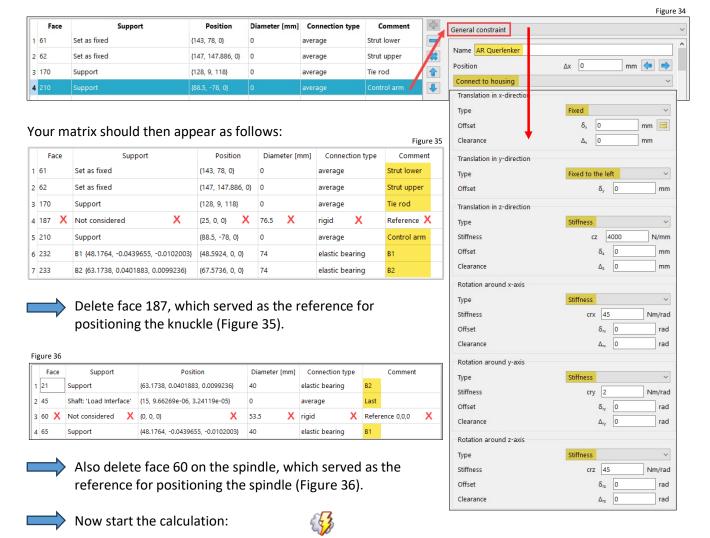


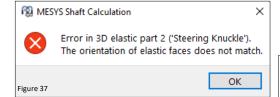
Figure 3

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



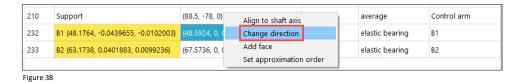
In the right-hand pane, for the connection to the control arm, activate 'General constraint' as shown in Figure 34:





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.



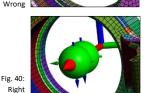


Fig. 39:

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.



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

Optionally, use 'Results element'.

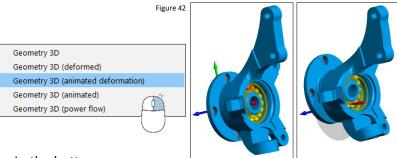
						Figure 41
	Comment	Frequency	n [rpm]	Fx [N]	Fr [N]	Ft [N]
Shaft			Load Interface	Load Interface	Load Interface	Load Interface
Element			General	Belastung	Belastung	Belastung
1	Driving	0.75	815	0	-4500	0
2	Cornering	0.25	815	200	-5130	0
☑ Run calculation for result element only Result element 2						

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).

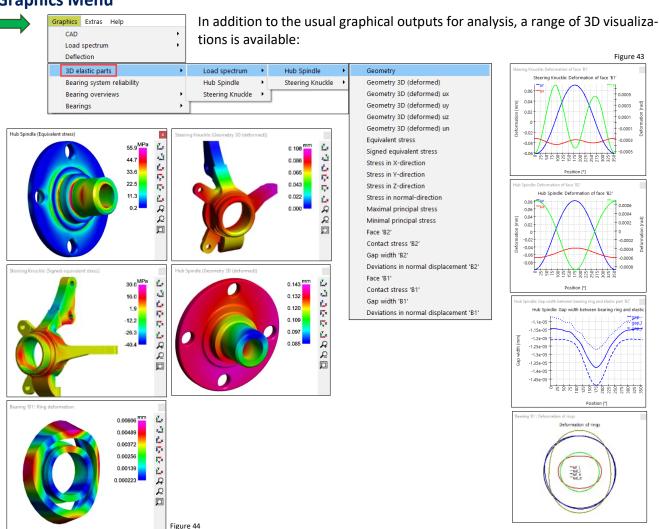


Results Overview



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

Graphics Menu



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.