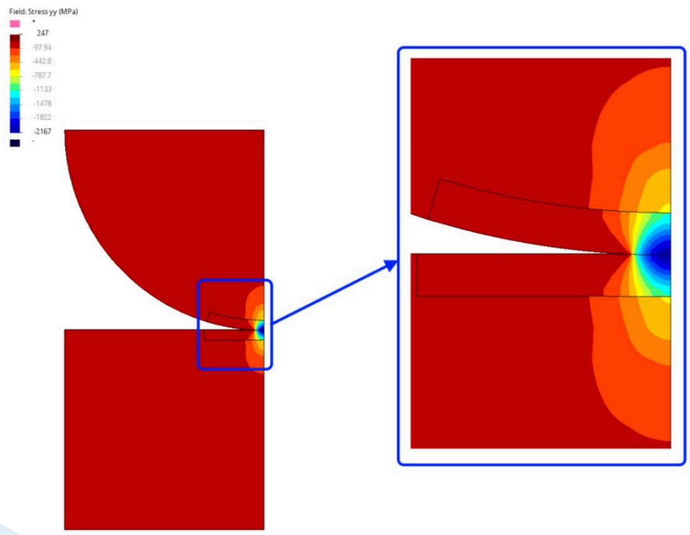
AKSELOS



CONTACT ANALYSIS FOR CYLINDERS WITH PARALLEL AXES

Application note

Objectives:

- Perform Contact Problem analysis (non-linear) to predict Akselos Modeler, then compare with theoretical results.

Model Description:

The model below is a finite element representation of a Contact between two cylinders with parallel axes. The first cylinder has a radius R = 0.8m and the second cylinder has a radius R = 1m with length = 0.04m. The sphere is subjected to a force F = 192 kN.

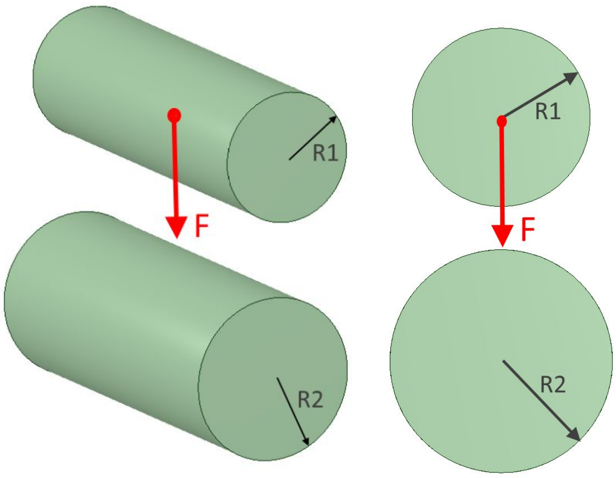


Figure 1: Model schematic.

Model Properties:

Cylinder 1 (R=0.8) Properties	
Elastic Modulus (E_1)	432 GPa
Poisson Ratio (v_1)	0.32
Cylinder 2 (R=1) Properties	
Elastic Modulus (E_2)	432 GPa
Poisson Ratio (v_2)	0.32

Table 1: Model Properties

Hand Calculations

Based on theory, The radius of the contact area is given as:

$$a = \sqrt{Rd}$$

Here d is determined by:

$$\frac{1}{E^*} = \frac{1-{\nu_1}^2}{E_1} + \frac{1-{\nu_2}^2}{E_2}$$

$$\frac{1}{R} = \frac{1}{R_1} + \frac{1}{R_2}$$

$$F = \frac{\pi}{4} E^* L d$$

and ν_1, E_1 (resp. ν_2, E_2) are the Poisson ratio and Young's modulus for the sphere (resp. half-space).

The maximum normal pressure is given by:

$$P_{max} = \left(\frac{E \times F}{\pi \times L \times R} \right)^{1/2}$$

Also, we use the following material and geometric properties:

$E_1 = 432$ GPa, $E_2 = 432$ GPa, $\nu_1 = \nu_2 = 0.32$, $R_1 = 0.8$m, $R_2 = 1$m.

Then we have:

$P_{max} = 909.54$ MPa

STEP 1: Create components

- Create a model consisting of 4 component, where 2 components contain Contact surfaces. *(Refer to the Jacket Leg Contact Problem Application Note in the Akselos User Manuals for a step-by-step tutorial on how to create Contact components.)*

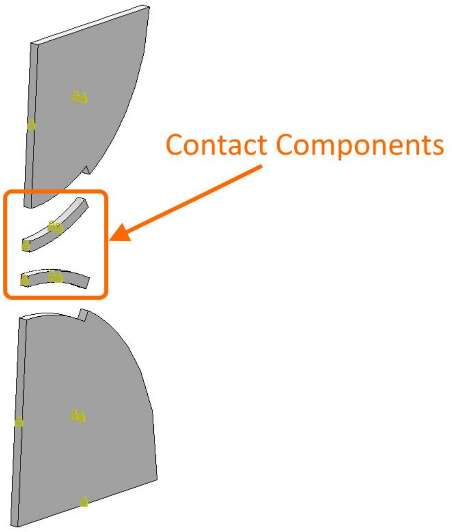


Figure 2: Components

- To obtain and accurate solution, you should make Conforming Meshes at the contact surfaces as shown in the figure below.

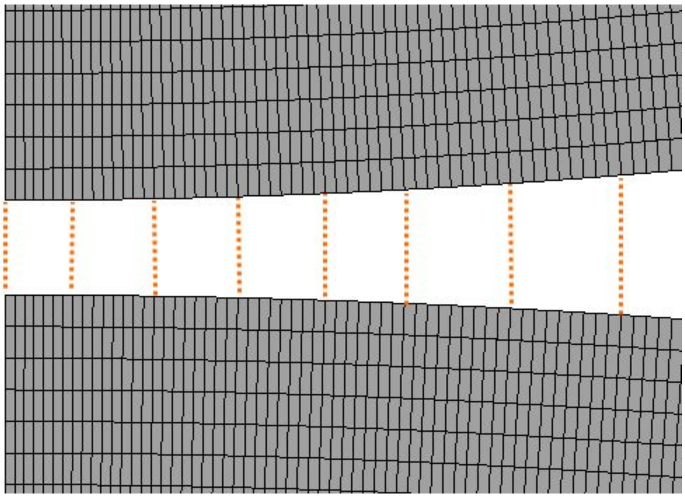


Figure 3: Conforming mesh at Contact surfaces

- The model geometry and FE mesh are shown in Figure 5. It also includes the symmetry boundary conditions on the green surfaces which represent only one quarter of the domain. The base of the cylinder domain is clamped, and we impose the force F as a pressure on the red surface. We impose Dirichlet boundary condition on the round surfaces which appears on the green surfaces so that the model can move in the vertical direction (i.e. the y-direction).

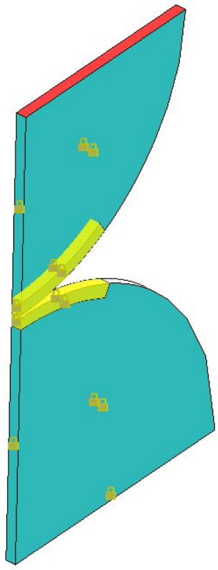


Figure 4: Boundary Conditions

STEP 2: Create Contact surfaces

We add below code in json file of 2 Contact components:

```
"contact_surfaces": [
  {
   "surface_name": "contact_surface"
  }
 ],
```

Contact_surface is the name of Contact Surface in each component

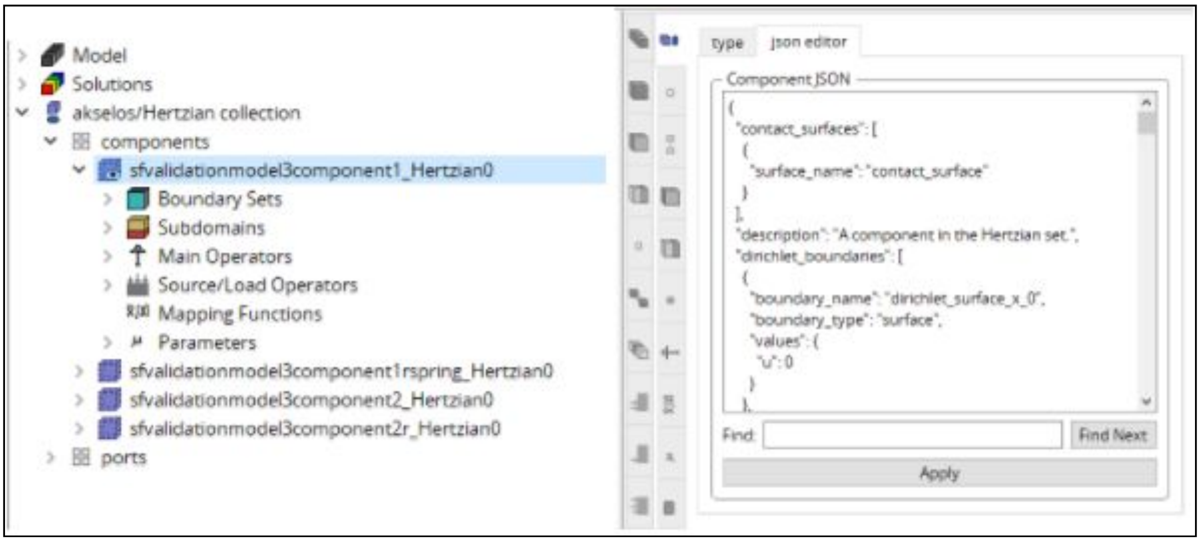


Figure 5: Create Contact surfaces

STEP 3: Create Model with Contact

- Press *Connect/Surface-To-Surface Contact*

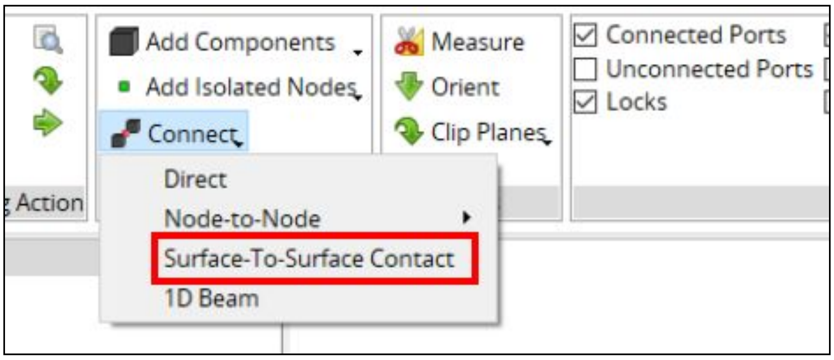


Figure 6: Surface-to-Surface Contact

- Click on the 2 ports on the model to connect them as shown in the figure below:

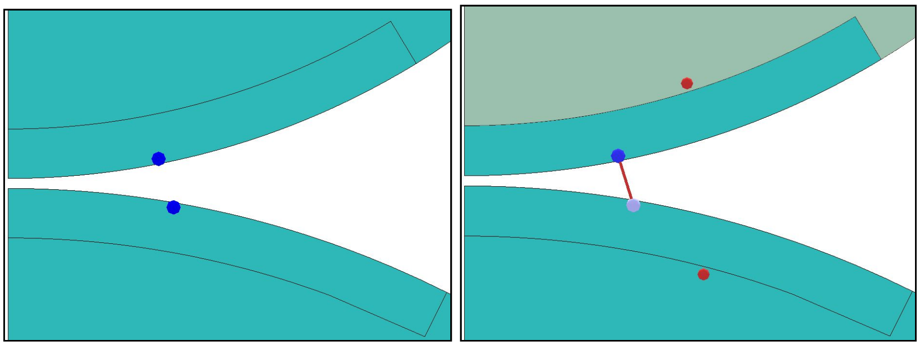


Figure 7: Contact connection

After creating the Contact connections, click on “Connected Ports” in the “Assemble” tab. A settings box will appear. The settings box includes the “Parameters” tab for Contact as shown in the figure below:

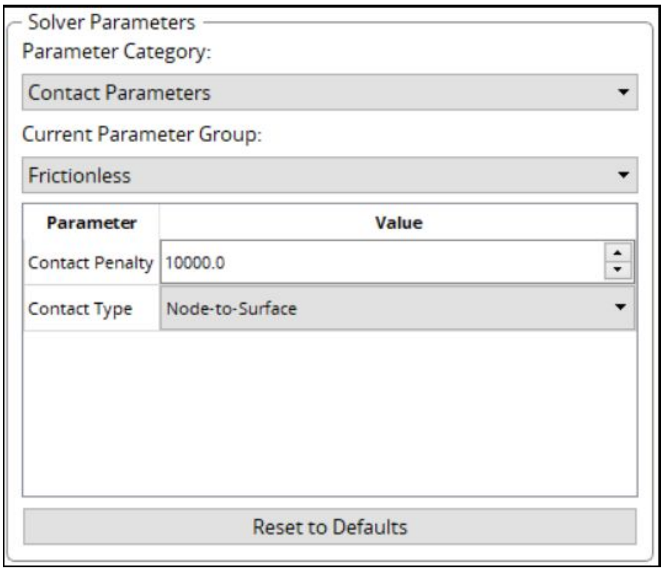


Figure 8: Contact settings box

Parameters at Contact setting box:

- Contact Penalty (kPa/m): Penalty Coefficient of Contact Analysis.
- Master/Slave Location Tolerance: Tolerance for Akselos solver.
- Use Nodal Normal:
- Contact Type: Akselos Modeler supports 2 types of Contact: Node-to-Surface and Surface-to-Surface.

STEP 3: Solve the model

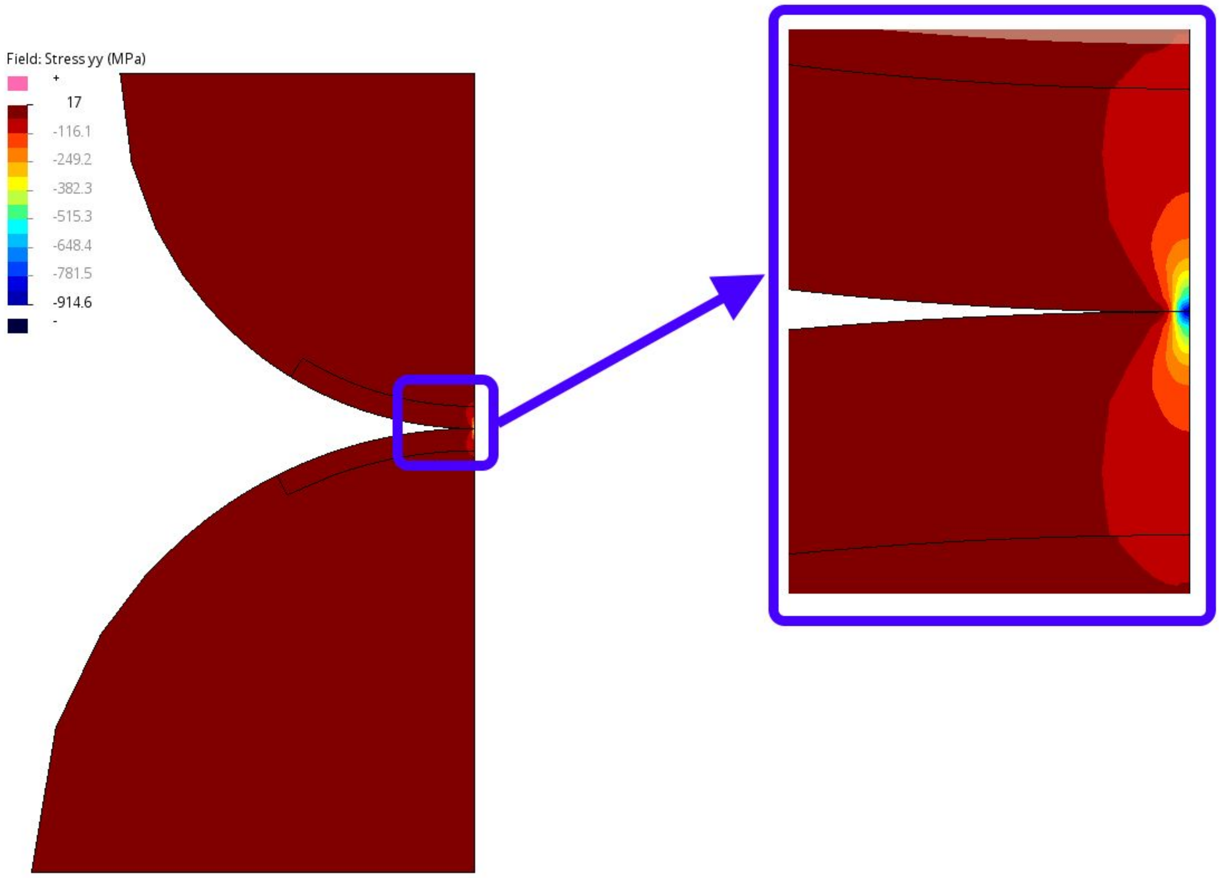
- Upload the model to Akselos server. *(Refer to the Component Editor tutorials in the Akselos User Manuals for a step-by-step tutorial.)*

You can then solve the model with the FEA solver. If you want to solve using the Hybrid solver which solves the model using RB-FEA in linear regions and FEA in the nonlinear regions, you have to train the model first using the **Akselos Web Dashboard**.

To perform a Hybrid Solve:

- Select all components that contain contact connections (the components that are highlighted in yellow in figure 4)
- Choose “Save Current Section” in the Actions tab in the properties panel.
- Select “Hybrid” from the menu in the Solve tab. In the “Solver Options”, select the the contact components that you stored before as “FEA components”.

STEP 3: Results



Quantity	Theory	Hybrid Solver Value	% Difference
P_{max}	909.54 MPa	914.6 MPa	0.56