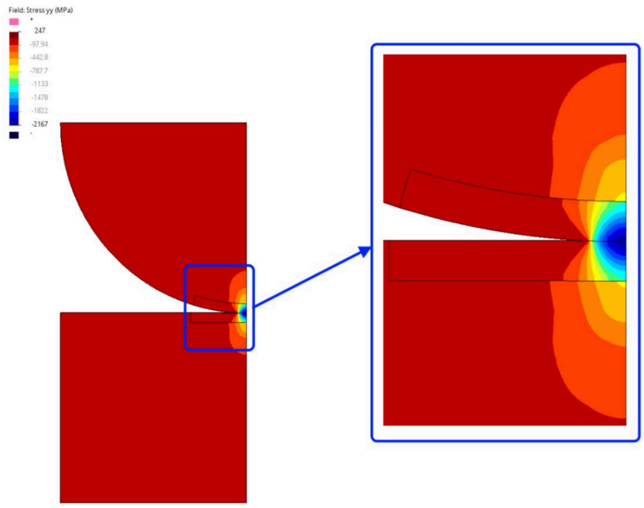



# CONTACT ANALYSIS FOR SMALL DEFORMATIONS

Application note

# Objectives:

- Perform Contact Problem analysis (non-linear) using Akselos Modeler
- Compare analysis results with theoretical results.

# Model Description:

The model below is a finite element representation of a contact problem between a sphere (radius R = 1m) and a half-space. The sphere is subjected to a force F = 1000 kN.

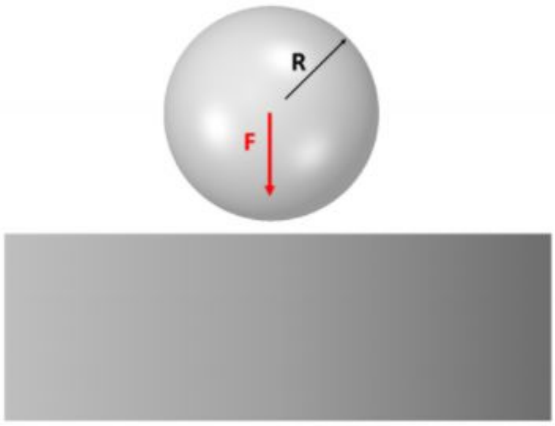



***Figure 1: Model schematic.***

# Model Properties:

| Sphere Properties | |
|---|---|
| Elastic Modulus ($E_1$) | 432 GPa |
| Poisson Ratio ($v_1$) | 0.32 |
| **Half-space Properties** | |
| Elastic Modulus ($E_2$) | 72 GPa |
| Poisson Ratio ($v_2$) | 0.32 |

***Table 1: Model Properties***

## Hand Calculations

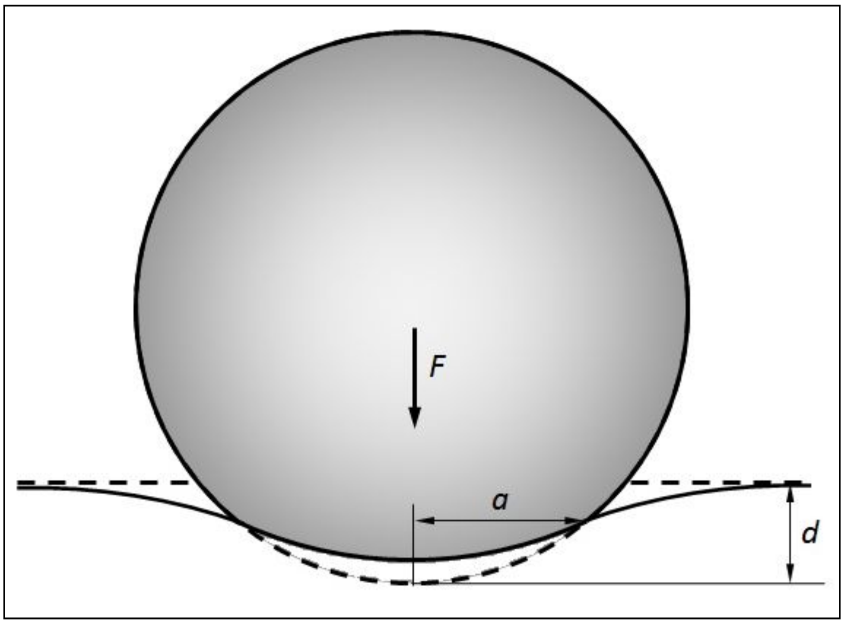



***Figure 2: Contact of an elastic sphere with an elastic half-space***

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

$$a = \sqrt{Rd}$$

d is determined by:

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

$$F = \frac{4}{3} E^* R^{1/2} d^{3/2}$$

and $\nu_1$, $E_1$ (resp. $\nu_2$, $E_2$) are the Poisson ratio and Young's modulus for the sphere (resp. half-space).

The maximum normal pressure is given by:

$$P_{max} = \frac{3F}{2\pi a^2}$$

Also, we use the following material and geometric properties:

***$E_1$ = 432 GPa, $E_2$ = 72 GPa, $\nu_1$ = $\nu_2$ = 0.32, R = 1m.***

Then we have:

***$P_{max}$ = 2091.39 MPa***

# STEP 1: Create components

- Create model including 4 component, where 2 components contain the 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.)***

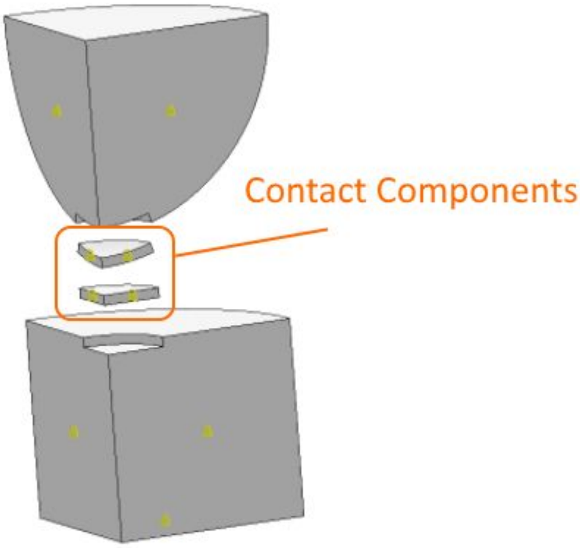



***Figure 3: Components***

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

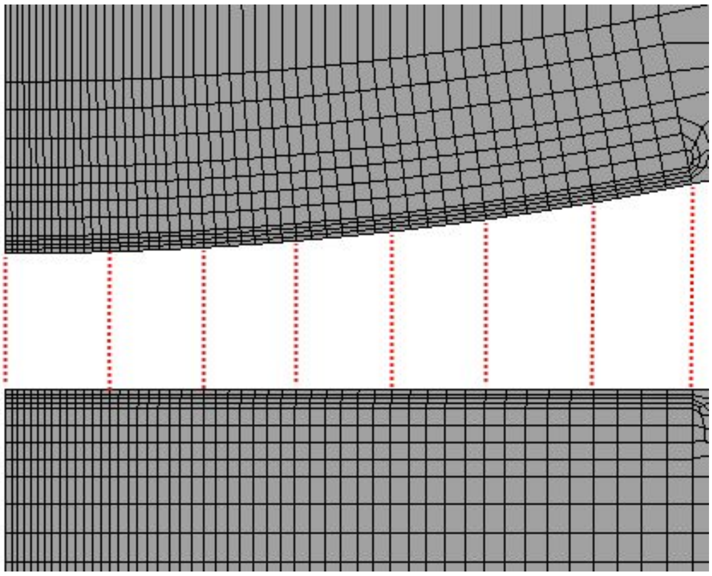


***Figure 4: Conforming mesh at Contact surfaces***

- 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 half-space 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).

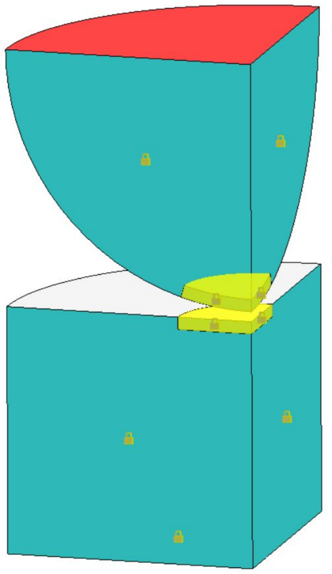


***Figure 5: Boundary Conditions***

## STEP 2: Create Contact surfaces

Add the JSON code below for the 2 Contact components:

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

Contact_surface is the name of Contact Surface in each component.

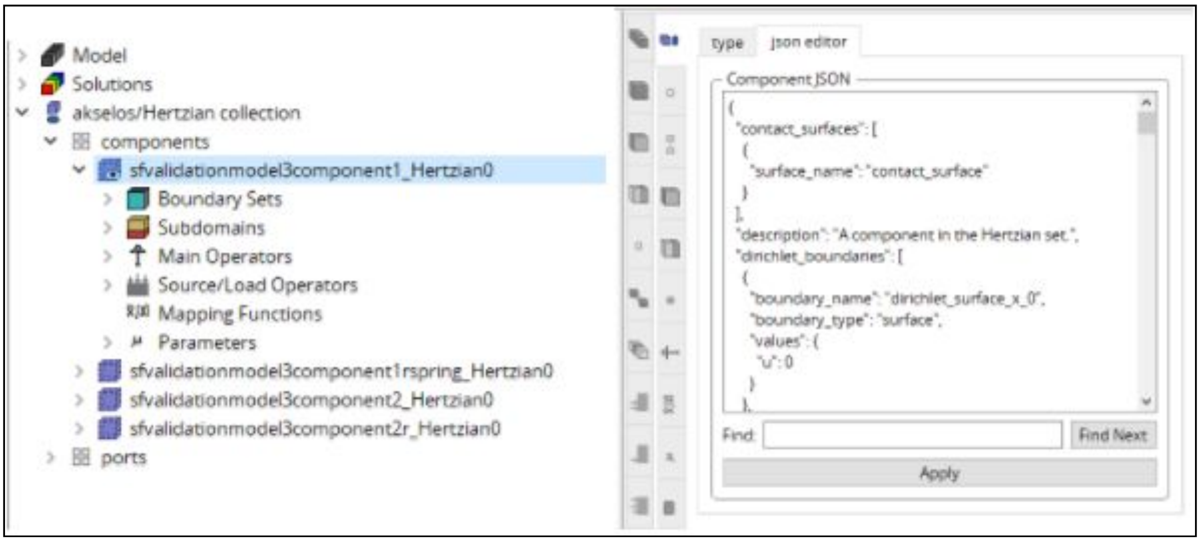



***Figure 6: Create Contact surfaces***

# STEP 3: Create Model with Contact

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

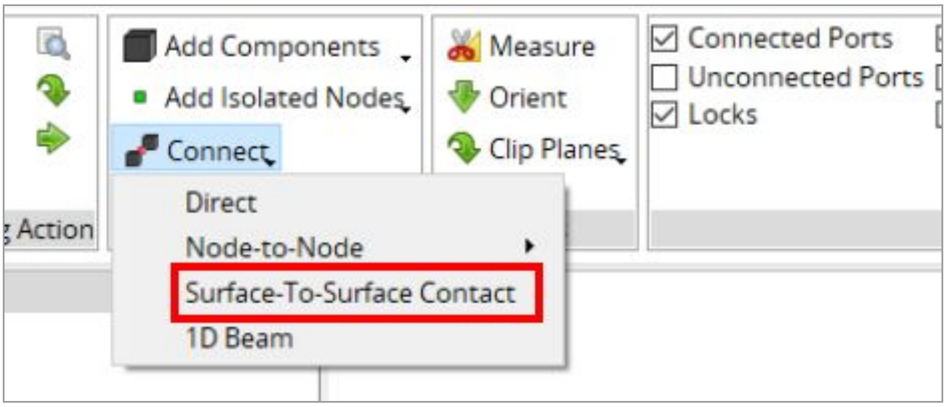



***Figure 6: Surface-to-Surface Contact***

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

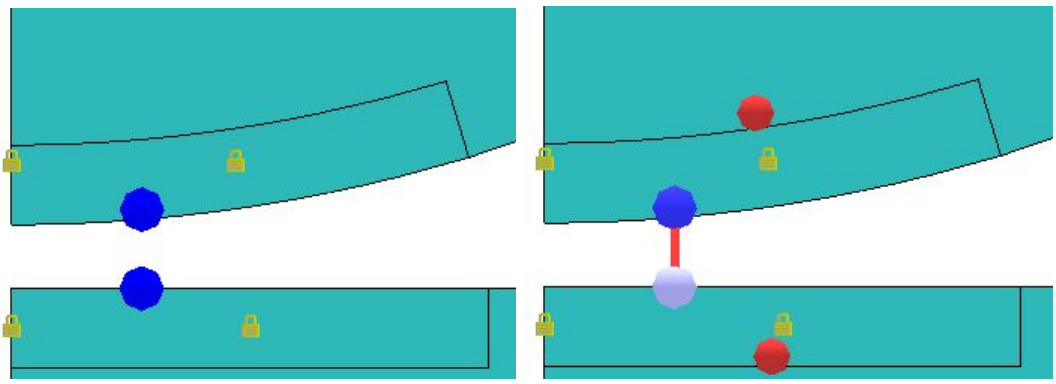
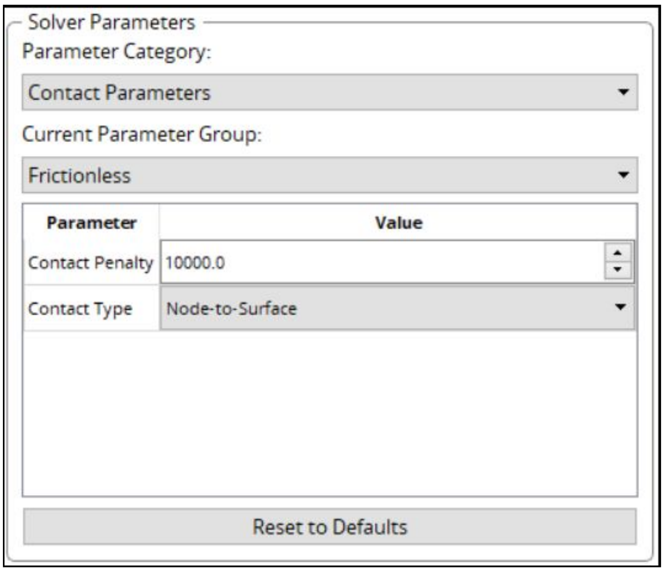


***Figure 7: Contact connection***

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:




The parameters for Contact modeling are:

- 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

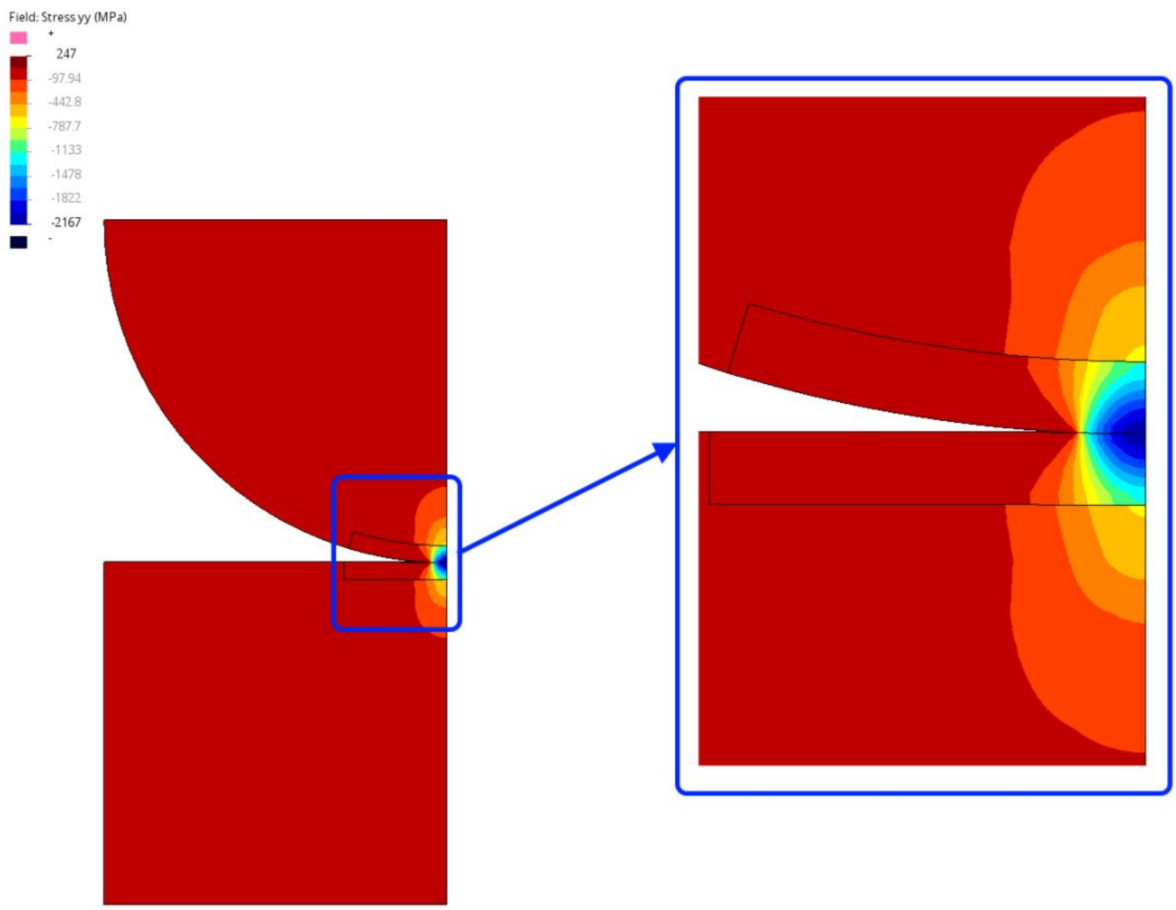
- Upload the model to Akselos server. ***(Refer to the Component Editor tutorials in the Akselos User Manuals for a step-by-step tutorial.)***

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 5)
- 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 (Figure 6) as “FEA components”.

# STEP 3: Results




| Quantity | Hertzian Theory | Hybrid Solver Value | % Difference |
|---|---|---|---|
| $P_{max}$ | 2091.39 MPa | 2167 MPa | 3.6 |