COMSOL MULTIPHYSICS®

Cylinder Roller Contact

SOLVED WITH COMSOL MULTIPHYSICS 3.5a

COMSOL

Cylinder Roller Contact

Introduction

Consider an infinitely long steel cylinder resting on a flat aluminum foundation, where both structures are elastic. The cylinder is subjected to a point load along its top. The objective of this study is to find the contact pressure distribution and the length of contact between the foundation and the cylinder. An analytical solution exists, and this model includes a comparison against the COMSOL Multiphysics solution. This model is based on a NAFEMS benchmark (see Ref. 1).

Model Definition

This is clearly a plane strain problem and the Plane Strain application mode from the Structural Mechanics Module is thus a suitable (two-dimensional) application mode. The geometry is reduced to half of the geometry at the vertical symmetry axis due to symmetry reasons.
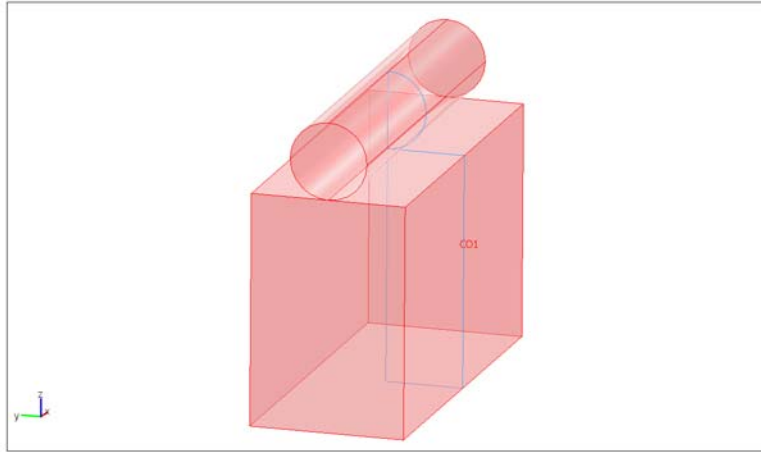

Figure 1: Geometry in model

The cylinder is subjected to a point load along its top with an intensity of 35 kN. Both the cylinder and block material are elastic, homogeneous, and isotropic.

The contact modeling method in this example does not include friction as described in Ref. 1. This model uses a contact pair, which is a straightforward way to implement a contact problem using the Structural Mechanics Module.

Results and Discussion

The deformed shape and the Von Mises stress distribution are depicted in Figure 2.
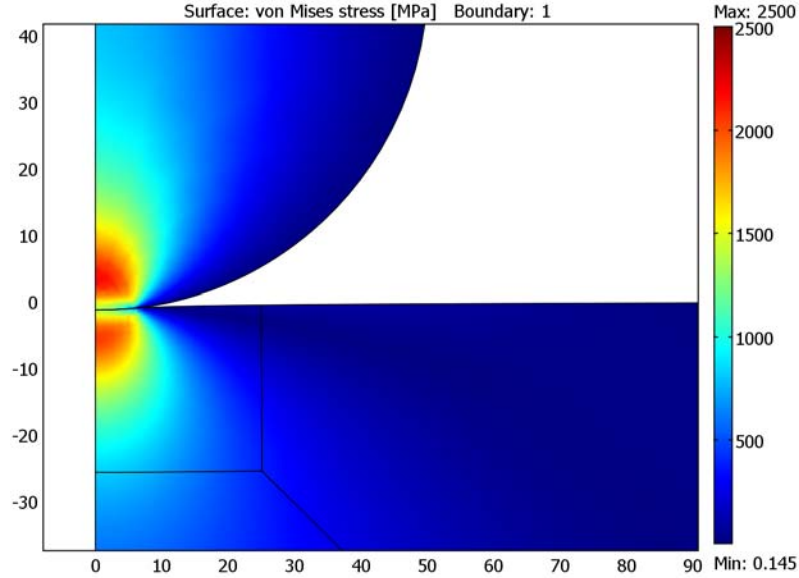
Figure 2: Deformation and von Mises stress at the contact area.

The analytical solution for the contact pressure as a function of the x-coordinate is given by

$$P = \sqrt{\frac{F_n E'}{2\pi R'} \times \left(1 - \left(\frac{x}{a}\right)^2\right)}$$

$$a = \sqrt{\frac{8F_n R'}{\pi E'}}$$

where F_n is the applied load (load/length), E' the combined elasticity modulus, R' the combined radius, and x is the x-axis coordinate.

The combined Young's modulus and radius are defined as:

$$E' = \frac{2E_1E_2}{E_2(1 - v_1^2) + E_1(1 - v_2^2)}$$

$$R' = \lim_{R_2 \to \infty} \frac{R_1R_2}{R_1 + R_2} = R_1$$

In these equations, E_1 and E_2 are Young's modulus of the roller and the block, respectively, and R_1 is the radius of the roller.

This gives a contact length of 6.21 mm and a maximum contact pressure of 3585 MPa.

The contact pressure along the contact area for both the analytical and the COMSOL Multiphysics solution are depicted in Figure 3. The COMSOL Multiphysics solution is the solid line, the analytical result is the dashed line.
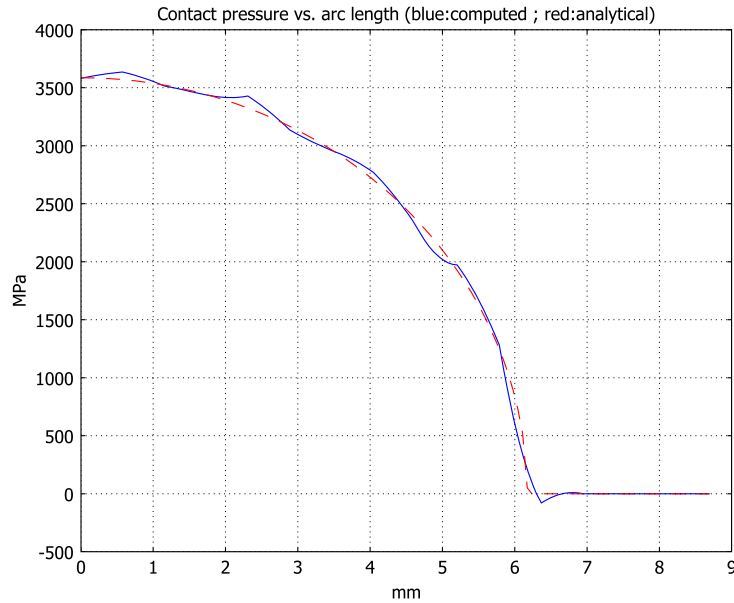

Figure 3: Analytical pressure distribution and COMSOL Multiphysics solution (dashed).

The maximum contact pressure obtained with COMSOL Multiphysics is 3581 MPa, which is in good agreement with the analytical solution.

The maximum contact length (equivalent to a null contact pressure) is evaluated at 6.31 mm.

References

1. A.W.A. Konter, *Advanced finite element contact benchmarks*, NAFEMS, 2006.

2. M. A. Crisfield, *Non-linear Finite Element Analysis of Solids and Structures, volume 2: Advanced Topics*, John Wiley & Sons Ltd., England, 1997.

Modeling Using COMSOL Multiphysics

The Structural Mechanics Module supports contact boundary conditions using contact pairs. The contact pair is defined by a master (contacting) boundary and a slave (contacted) boundary. The contact boundary pair comprises a flat boundary and a curved boundary. The flat boundary is defined as the master boundary and the curved boundary as the slave boundary.

Because contact is only present in a small area, a local mesh refinement is required. Due to the geometry shape, you use a free mesh for the cylinder domain and a mapped mesh for the aluminum block. The block geometry requires some modification in order to set up a refined mesh area.

This example assumes a nominal thickness (**thickness** = 1).

The selected length unit in this model is mm, so the MPa unit system, which is based on mm as the length unit, is a suitable unit system.

Model Library path: `Structural_Mechanics_Module/Benchmark_Models/cylinder_roller_contact`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 On the **New** page, select **2D** from the **Space dimension** list.

2 Expand to the next submenu by clicking on **Structural Mechanics Module> Plane Strain>Static analysis**.

3 Click **OK**.

1 From the **Options** menu select **Constants** and enter the following constant names, expressions, and descriptions (optional); when done, click **OK**.

| NAME	EXPRESSION	DESCRIPTION
E1	70e3[MPa]	Block Young's modulus
E2	210e3[MPa]	Cylinder Young's modulus
nu	0.3	Poisson's ratio
Fn	35[kN]	External load
E_star	2*E1*E2/((E1+E2)*(1-nu^2))	Combined Young's modulus
R	50[mm]	Combined radius
th	1[mm]	Thickness
lc	10[mm]	Estimated contact length
a	sqrt(8*Fn*R/(pi*E_star*th))	Contact length
pmax	sqrt(Fn*E_star/(2*pi*R*th))	Maximum contact pressure

GEOMETRY MODELING

1 In the **Draw** menu select **Use Assembly**.

1 Shift-click the **Ellipse/Circle (Centered)** button on the Draw toolbar, then set the **Radius** to 50 and the position of the center to (0, 50). Click **OK**.

2 Shift-click the **Rectangle/Square** button on the Draw toolbar and set both **Width** and **Height** to 200. Set the corner position to (−100, −200). Click **OK**.

3 Click the **Zoom Extents** button on the Main toolbar.

4 Draw a second rectangle, this one of width 100 and height 300 with the lower left corner at (−100, −200).

5 Click the **Create Composite Object** button on the Draw toolbar and type the expression (C1+R1)-R2 in the **Set formula** edit field. Click **OK**.

6 Choose **Specify Objects>Square** from the **Draw** menu to create two squares with the following specifications:

PROPERTY	SQUARE 1	SQUARE 2
Width	100	25
Base	Corner	Corner
x	0	0
y	-100	-25

7 Choose **Specify Objects>Line** from the **Draw** menu and create a line between grid point (25, –25) and grid point (100, –100).

The mesh needs to be refined in the contact area. To do this in an efficient way, divide the curved boundary into two separate boundaries. To achieve this, add a point along the curved boundary.

8 Draw a **Point** at the origin (0, 0).

9 Click the **Rotate** button on the Draw toolbar, then set the **Rotation angle** to 10 degrees and the *y*-coordinate of the **Center point** to 50. Click **OK**.

10 Select all domains (press Ctrl+A), then click the **Split Object** button on the Draw toolbar.

11 Select all the geometry objects of the top part (CO3 and PT2), then click the **Coerce to Solid** button on the Draw toolbar.

12 Select all the geometry objects of the bottom part (CO2, CO4, CO5, and B2), then click the **Coerce to Solid** button.
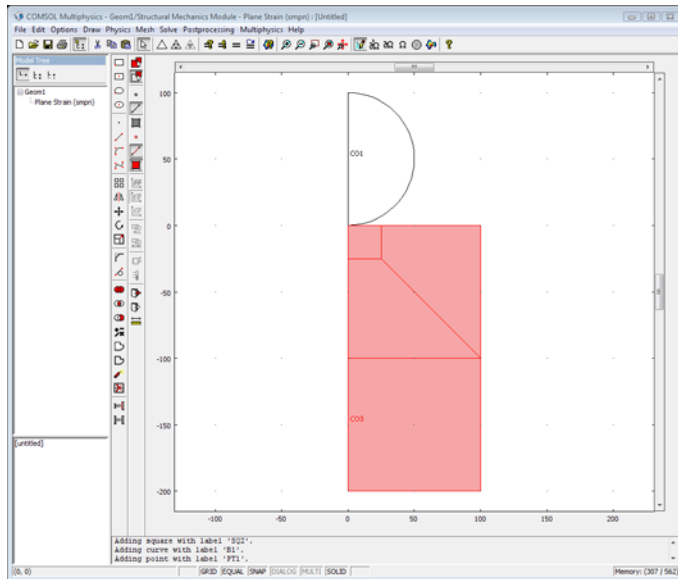
PHYSICS SETTINGS

Model Settings

1 In the **Physics** menu, go to **Model Settings** and select **MPa** from the **Base unit system** list.

2 Click **OK**.

1 Choose **Physics>Subdomain Settings** to specify the subdomain settings for the Plane Strain application mode according to the following table:

SETTINGS	SUBDOMAINS 1–4	SUBDOMAIN 5
E	E1	E2
ν	nu	nu

The nominal thickness is used; make sure that **thickness** is set to 1 for all subdomains.

2 Click **OK**.

Boundary Conditions

1 Choose **Physics>Boundary Settings** to open the **Boundary Settings** dialog box; then specify the following constraint conditions:

SETTINGS	BOUNDARIES 1, 3, 5, 13	BOUNDARY 2
Constraint condition	Symmetry plane	Fixed

The other boundaries are free, which is the default boundary condition.

2 Click **OK**.

Point Settings

The external load is a point load. Due to the symmetry, apply one half of the total force on the studied geometry:

1 From the **Physics** menu select **Point Settings**.

2 In the **Point Settings** dialog box select Point 11.

3 Click the **Load** tab, and then type -Fn/2 in the **F$_y$** edit field.

4 Click **OK**.

Contact Pairs

Define a contact pair for Boundaries 7 and 14. A general advice is to set the curved surface as slave (contacting) boundary and the straight surface as master (contacted) boundary. The top boundary (number 7) is set to slave boundary and the bottom boundary (number 14) as master boundary in this model. The steel block is also the correct master surface because it has a high stiffness.

1 From the **Physics** menu select **Contact Pairs**.

2 In the **Contact Pairs** dialog box, click **New**.

3 In the **Master boundaries** list, select Boundary 7.

4 Click the **Check Selected** button below the **Master boundaries** list.

5 In the **Slave boundaries** list select Boundary 14.

6 Click the **Check Selected** button below the **Slave boundaries** list.

7 Click **OK**.

The next step is to specify the contact parameters in the **Boundary Settings** dialog box.

8 In the **Boundary Settings** dialog box, click the **Pairs** tab, and select **Pair 1 (contact)**.

Note: To reduce the number of iteration steps and improve convergence, it is good practice to set an initial contact pressure as close to the solution as possible. A good approximation is to use the value of the external pressure, in this case the external point load divided by an estimated contact length and the thickness. In this model you need to set an initial contact pressure to make the model stable for the initial conditions.

---

9 On the **Contact, Initial** page, type (Fn/2)/(lc*th) in the **Contact pressure** edit field (the division by 2 is to account for the symmetry).

10 Click **OK**.

MESH GENERATION

You can achieve an accurate contact pressure by refining the mesh around the contact zone. A general recommendation is to have at least 10 contacting nodes along the slave contact boundary. Apply an element size of 0.59 mm for the cylinder and 1.5 mm for the block. Due to the geometry shape, use a free mesh (triangular elements) for the cylinder and a mapped mesh (quadrilateral elements) for the block.

1 Open the **Free Mesh Parameters** dialog box by selecting **Free Mesh Parameters** from the **Mesh** menu.

2 Click the **Boundary** tab. Select Boundary 14, then type 0.59 in the **Maximum element size** edit field.

3 Click the **Subdomain** tab, then select Subdomain 5. Click **OK**.

4 Click the **Mesh Selected (Free)** button on the Mesh toolbar.

In the message log you can see that the free mesh is composed of around 360 elements.

5 Open the **Mapped Mesh Parameters** dialog box by selecting **Mapped Mesh Parameters** from the **Mesh** menu.

6 On the **Boundary** page, select Boundaries 3, 5, and 7. Then select **Constrained edge element distribution** and type 20 in the **Number of edge elements** edit field.

7 On the **Boundary** page, select Boundary 1. Then select the **Constrained edge element distribution** and type 10 in the **Number of edge elements** edit field. Click **Apply**.

8 Go to **Subdomain** page and select Subdomains 1, 2, 3, and 4.

9 Click the **Mesh Selected** button. Click **OK**.

The information in the message log shows that the total mesh consists of approximately 1750 elements (roughly 360 elements for the free mesh and 1400 elements for the mapped mesh).
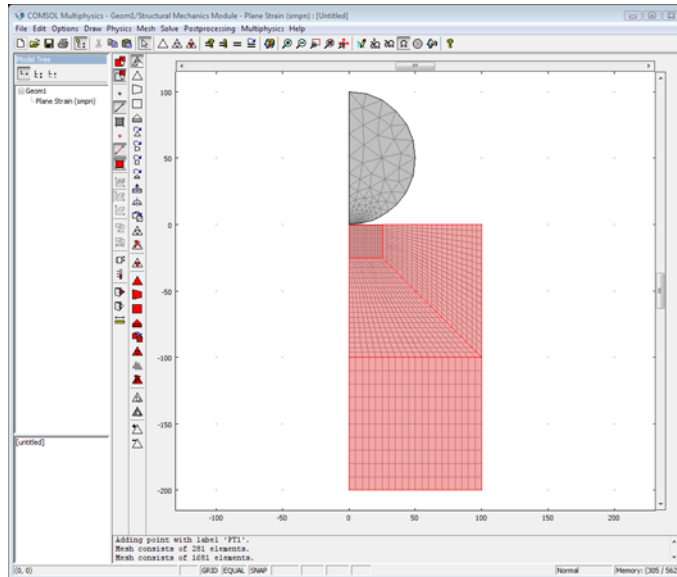

COMPUTING THE SOLUTION
Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION
The default plot shows the von Mises stress. Because of the point-load constraint, a stress concentration develops on the top of the cylinder. Due to the automatic plot-range settings, the stress around the contact surface looks almost identical.

1 Click the **Plot Parameters** button on the Main toolbar.

2 On the **General** page, select the **Surface** and **Boundary** check boxes and clear the **Geometry edges** check box in the **Plot type** area.

3 On the **Surface** page, click the **Range** button. In the **Color Range** dialog box clear the **Auto** check box, then type 2500 in the **Max** edit field. Click **OK**.

4 On the **Boundary** page, type 1 in the **Expression** edit field on the **Boundary Data** tab. Click the **Uniform color** option button in the **Boundary color** area, then click the **Color** button and select black. Click **OK**.

5 Click **OK**.

Zoom in on the contact area to get the following plot:
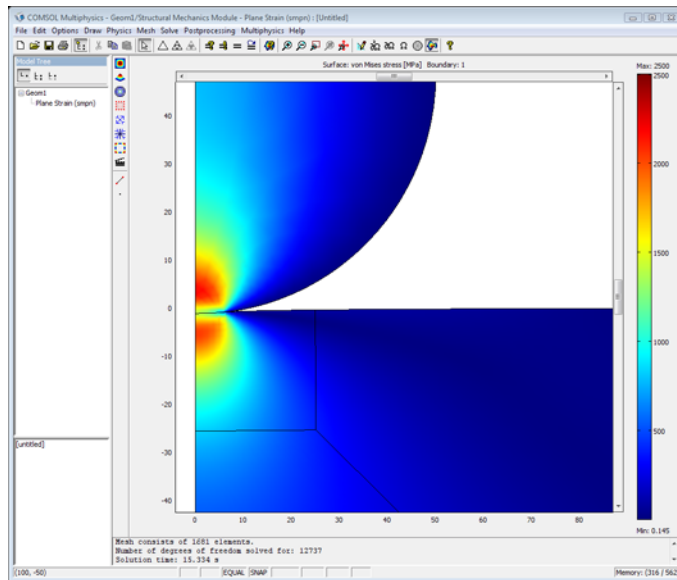
The instructions below show how to create a line plot to compare the analytical contact pressure distribution with the contact pressure calculated by the model.

1 Choose **Options>Expressions>Boundary Expressions** and select Boundary 14. Enter the following constant names and expressions:

NAME	EXPRESSION
theta	atan2(y-50,x)+pi/2

NAME	EXPRESSION
x_local	sqrt(x^2+(y-50)^2)*theta
p_analytical	pmax*sqrt(1-(x_local/a)^2)

2 Click **OK**.

3 In the **Solve** menu, select **Update Model**.

You can now plot the analytical expression of the contact pressure.

4 Choose **Postprocessing>Domain Plot Parameters**.

5 Click the **Line/Extrusion** tab, then from the **Predefined quantities** list select **Contact pressure, contact Pair 1**.

6 Select Boundary 14 (the slave boundary), then click **Apply** to generate a plot of the contact pressure calculated in the model.

7 In the **Expression** edit field type p_analytical.

8 Click the **Line Settings** button, then select **Color** in the **Line color** list, and **Dashed line** in the **Line style** list. Click **OK**.

9 Click the **General** tab, then select **Keep current plot**.

10 Click the **Title/Axis** button and type Contact pressure vs. arc length (blue: calculated, red: analytical) in the **Title** edit field. In the **First axis label** edit field type mm and in the **Second axis label** edit field type MPa. Click **OK**.

11 Click **OK** in **Domain Plot Parameters** dialog box to generate the plot of the analytical expression for the contact pressure.