Pinched Hemispherical Shell

Introduction

This example studies the deformation of a hemispherical shell, where the loads cause significant geometric nonlinearity. The maximum deflections are more than two magnitudes larger than the thickness of the shell. The problem is a standard benchmark, used for testing shell formulations in a case which contains membrane and bending action, as well as large rigid body rotation. It is described in Ref. 1.

Model Definition

Figure 1 shows the geometry and the applied loads. Due to the double symmetry, the model only includes one quarter of the hemisphere.

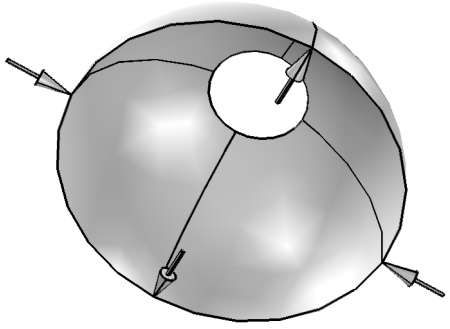

Figure 1: The geometry and loads.

The material is linear elastic with E = 68.25 MPa and ν = 0.3. The radius of the hemisphere is 10 m, and the thickness of the shell is 0.04 m. The hole at the top has a radius of 3.0902 m because 18° in the meridional direction from the top has been removed. The forces all have the value 200 N before taking symmetry into account. In the model, two forces of 100 N are applied in the symmetry planes at the lower edge of the shell.

©2013 COMSOL

Results and Discussion

The target solution in Ref. 1 is $u = -5.952$ m under the inward acting load and $v = 3.427$ m under the outward acting load. Both target values have an error bound of ±2%. The values computed in COMSOL are $u = -5.862$ m and $v = 3.407$ m. Both values are within 2% of the target. Figure 2 shows the deformed shape of the shell together with contours for the effective stress.

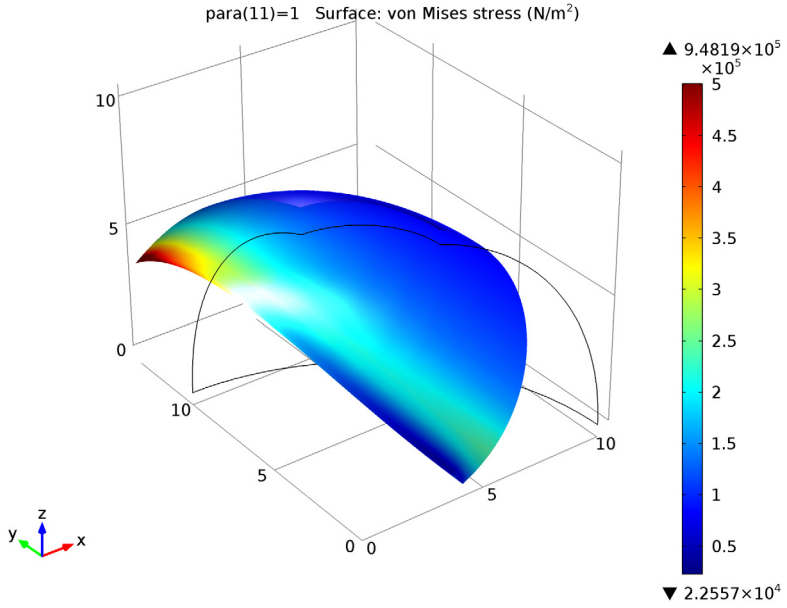

Figure 2: von Mises stress on top surface.

 ©2013 COMSOL

The change in the displacement as the load parameter increases is shown in Figure 3. As can be seen, the nonlinear effects are strong. The incremental stiffness with respect to the *y*-direction force increases with one order of magnitude during the loading.

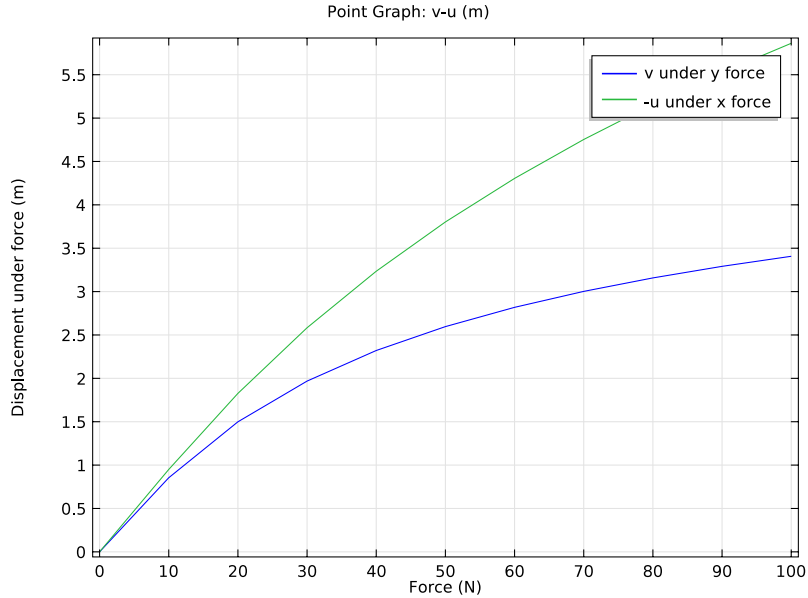

Figure 3: Displacements as functions of applied load.

Notes About the COMSOL Implementation

In a highly nonlinear problem it is a good idea to use the parametric solver to track the solution instead of trying to solve at the full load. Several solver settings can be tuned to improve the convergence. Due to the large difference between the bending and the membrane stiffnesses in a thin shell, a small error in the approximated displacements during the iterations can cause large residual forces. For this reason, the model uses manual control of the damping in the Newton method. This will often improve solution speed for problems with severe geometrical nonlinearity.

Because the model uses point loads, the gradients are steep close to the locations where the loads are applied. For this reason you modify the distribution of the elements so that finer elements are generated toward the corners of the model. From a computational point of view, this is more effective than using a uniform refinement of the mesh.

Reference

1. N.K. Prinja and R.A. Clegg, "A Review of Benchmark Problems for Geometric Non-linear Behaviour of 3-D Beams and Shells (SUMMARY)," *NAFEMS Ref: R0024*, pp. F9A–F9B, 1993.

Model Library path: `Structural_Mechanics_Module/Verification_Models/pinched_hemispherical_shell`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

3 In the **Add physics** tree, select **Structural Mechanics>Shell (shell)**.

4 Click **Next**.

5 Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.

6 Click **Finish**.

GEOMETRY 1

Sphere 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Sphere**.

2 In the **Sphere** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type 10.

4 Click the **Build Selected** button.

Block 1

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Block**.

2 In the **Block** settings window, locate the **Size and Shape** section.

3 In the **Width** edit field, type 10.

4 In the **Depth** edit field, type 10.

5 In the **Height** edit field, type 10.

 ©2013 COMSOL

6 Locate the **Position** section. In the **x** edit field, type `-5`.

7 In the **y** edit field, type `-5`.

8 In the **z** edit field, type `10*cos(18*pi/180)[m]`.

9 Click the **Build Selected** button.

Difference 1

1 Right-click **Geometry 1** and choose **Boolean Operations>Difference**.

2 In the **Difference** settings window, locate the **Difference** section.

3 Under **Objects to add**, click **Activate Selection**.

4 Select the object **sph1** only.

5 Under **Objects to subtract**, click **Activate Selection**.

6 Select the object **blk1** only.

7 Click the **Build Selected** button.

Convert to Surface 1

1 Right-click **Geometry 1** and choose **Conversions>Convert to Surface**.

2 Select the object **dif1** only.

3 Click the **Build Selected** button.

Delete Entities 1

1 Right-click **Geometry 1** and choose **Delete Entities**.

2 On the object **csur1**, select Boundaries 1–8 only.

You can do this by first selecting all boundaries and then removing Boundary 9.

3 Click the **Build Selected** button.

4 Click the **Zoom Extents** button on the Graphics toolbar.

MATERIALS

Material 1

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Material**.

2 Right-click **Material 1** and choose **Rename**.

3 Go to the **Rename Material** dialog box and type `Steel` in the **New name** edit field.

4 Click **OK**.

5 In the **Material** settings window, locate the **Geometric Entity Selection** section.

6 From the **Geometric entity level** list, choose **Boundary**.

7 From the **Selection** list, choose **All boundaries**.

8 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value
Young's modulus	E	68.25e6
Poisson's ratio	nu	0.3
Density	rho	6850

Note that the density is not used for a static analysis so the value you enter has no effect on the solution.

SHELL

1 In the **Shell** settings window, locate the **Thickness** section.

2 In the *d* edit field, type `0.04`.

Symmetry 1 1

1 Right-click **Model 1>Shell** and choose **Symmetry**.

2 Select Edges 1 and 4 only.

Prescribed Displacement/Rotation 1

1 In the **Model Builder** window, right-click **Shell** and choose **Points>Prescribed Displacement/Rotation**.

2 Select Point 4 only.

3 In the **Prescribed Displacement/Rotation** settings window, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in z direction** check box.

Point Load 1

1 Right-click **Shell** and choose **Points>Point Load**.

2 Select Point 4 only.

3 In the **Point Load** settings window, locate the **Force** section.

4 In the $\mathbf{F}_p$ table, enter the following settings:

-100*para	x
0	y
0	z

 ©2013 COMSOL

Point Load 2

1 Right-click **Shell** and choose **Points>Point Load**.

2 Select Point 2 only.

3 In the **Point Load** settings window, locate the **Force** section.

4 In the $\mathbf{F}_p$ table, enter the following settings:

0	x
100*para	y
0	z

MESH 1

Mapped 1

1 In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **More Operations>Mapped**.

2 In the **Mapped** settings window, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

Distribution 1

1 Right-click **Model 1>Mesh 1>Mapped 1** and choose **Distribution**.

2 Select Edges 1 and 4 only.

3 In the **Distribution** settings window, locate the **Distribution** section.

4 From the **Distribution properties** list, choose **Predefined distribution type**.

5 In the **Number of elements** edit field, type 16.

6 In the **Element ratio** edit field, type 3.

7 From the **Distribution method** list, choose **Geometric sequence**.

Distribution 2

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 2 and 3 only.

3 In the **Distribution** settings window, locate the **Distribution** section.

4 From the **Distribution properties** list, choose **Predefined distribution type**.

5 In the **Number of elements** edit field, type 16.

6 In the **Element ratio** edit field, type 3.

7 Select the **Symmetric distribution** check box.

8 From the **Distribution method** list, choose **Geometric sequence**.

9 Click the **Build All** button.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Description
para	0	Solver parameter

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.

2 In the **Stationary** settings window, locate the **Study Settings** section.

3 Select the **Include geometric nonlinearity** check box.

4 Click to expand the **Study Extensions** section. Select the **Continuation** check box.

5 Click **Add**.

6 In the table, enter the following settings:

Continuation parameter	Parameter value list
para	range(0,0.1,1)

7 In the **Model Builder** window, right-click **Study 1** and choose **Show Default Solver**.

8 Expand the **Study 1>Solver Configurations** node.

Solver 1

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solver 1** node, then click **Stationary Solver 1**.

2 In the **Stationary Solver** settings window, locate the **General** section.

3 In the **Relative tolerance** edit field, type 0.00010.

4 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solver 1>Stationary Solver 1** node, then click **Parametric 1**.

5 In the **Parametric** settings window, click to expand the **Tuning** section.

 ©2013 COMSOL

6 Select the **Tuning of step size** check box.

7 In the **Initial step size** edit field, type `0.1`.

8 In the **Minimum step size** edit field, type `0.1`.

9 Locate the **General** section. From the **Predictor** list, choose **Constant**.

10 In the **Model Builder** window, under **Study 1>Solver Configurations>Solver 1>Stationary Solver 1** click **Fully Coupled 1**.

11 In the **Fully Coupled** settings window, click to expand the **Method and Termination** section.

12 From the **Nonlinear method** list, choose **Constant (Newton)**.

13 In the **Damping factor** edit field, type `1`.

14 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

Stress Top (shell)

1 In the **Model Builder** window, expand the **Results>Stress Top (shell)** node, then click **Surface 1**.

2 In the **Surface** settings window, click to expand the **Range** section.

3 Select the **Manual color range** check box.

4 In the **Maximum** edit field, type `500000`.

5 Click the **Plot** button.

1D Plot Group 3

1 In the **Model Builder** window, right-click **Results** and choose **1D Plot Group**.

2 Right-click **1D Plot Group 3** and choose **Point Graph**.

3 Select Points 2 and 4 only.

4 In the **Point Graph** settings window, locate the **y-Axis Data** section.

5 In the **Expression** edit field, type `v-u`.

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** edit field, type `para*100[N]`.

8 Click to expand the **Legends** section. Select the **Show legends** check box.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
`v under y force`
`-u under x force`

11 In the **Model Builder** window, click **1D Plot Group 3**.

12 In the **1D Plot Group** settings window, locate the **Plot Settings** section.

13 Select the **x-axis label** check box.

14 In the associated edit field, type `Force (N)`.

15 Select the **y-axis label** check box.

16 In the associated edit field, type `Displacement under force (m)`.

17 Click the **Plot** button.

Evaluate the displacements in the points where a comparison should be made with the target.

Derived Values

1 In the **Model Builder** window, under **Results** right-click **Derived Values** and choose **Point Evaluation**.

2 Select Points 2 and 4 only.

3 In the **Point Evaluation** settings window, locate the **Data** section.

4 From the **Parameter selection (para)** list, choose **Last**.

5 Locate the **Expression** section. In the **Expression** edit field, type `u`.

6 Click the **Evaluate** button.

7 In the **Expression** edit field, type `v`.

8 Click the **Evaluate** button.

 ©2013 COMSOL