Model created in COMSOL Multiphysics 6.4

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.

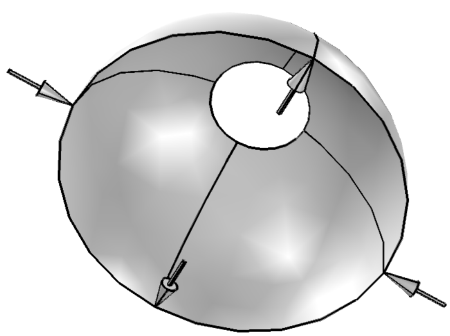


Figure 1: The geometry and loads.

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.

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 equivalent stress.

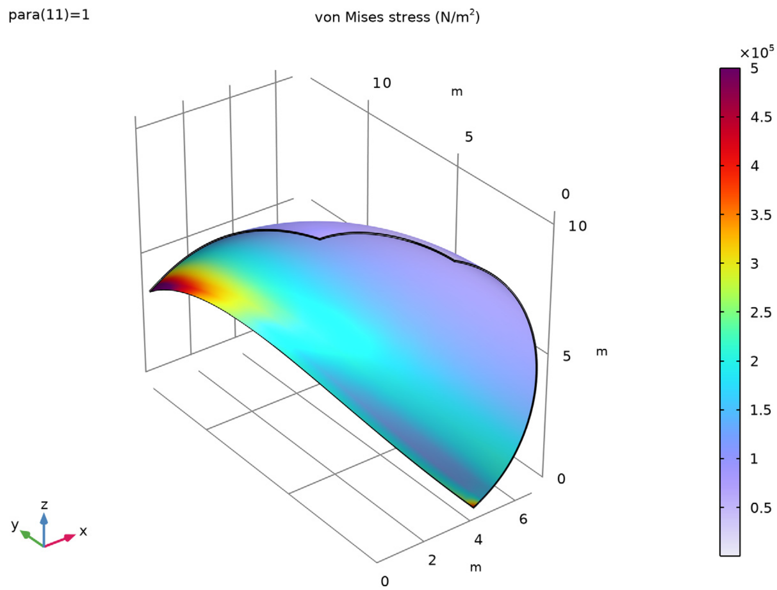


Figure 2: von Mises stress on top surface.

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 by one order of magnitude during the loading.

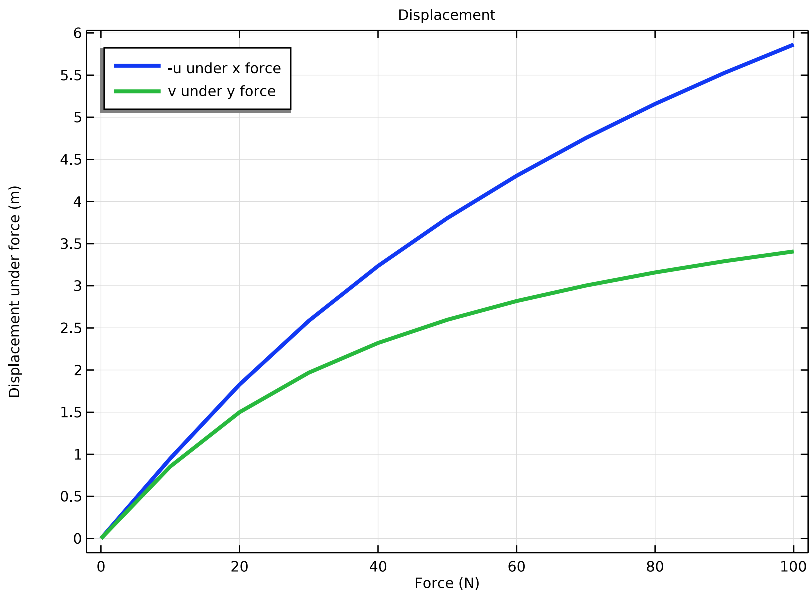


Figure 3: Displacements as functions of applied load.

Notes About the COMSOL Implementation

In a highly nonlinear problem it is a good idea to use the parametric continuation 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, manual control of the damping is used in the Newton method. This will often improve solution speed for problems with severe geometrical nonlinearities.

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.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/pinched_hemispherical_shell`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Structural Mechanics** > **Shell (shell)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies** > **Stationary**.

6 Click **Done**.

GEOMETRY 1

Sphere 1 (sph1)

1 In the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for **Sphere**, locate the **Size** section.

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

4 Click **Build Selected**.

Block 1 (blk1)

1 In the **Geometry** toolbar, click **Block**.

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

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

4 In the **Depth** text field, type `10`.

5 In the **Height** text field, type `10`.

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

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

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

9 Click **Build Selected**.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **sph1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **blk1** only.

6 Click **Build Selected**.

Convert to Surface 1 (csur1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Surface**.

2 Select the object **dif1** only.

3 In the **Settings** window for **Convert to Surface**, click **Build Selected**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, 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 In the **Settings** window for **Delete Entities**, click **Build Selected**.

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

MATERIALS

Steel

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Steel` in the **Label** text field.

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

Property	**Variable**	**Value**	**Unit**	**Property group**
Young's modulus	E	68.25e6	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's ratio
Density	rho	6850	kg/m³	Basic

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

SHELL (SHELL)

Thickness and Offset 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Shell (shell)** click **Thickness and Offset 1**.

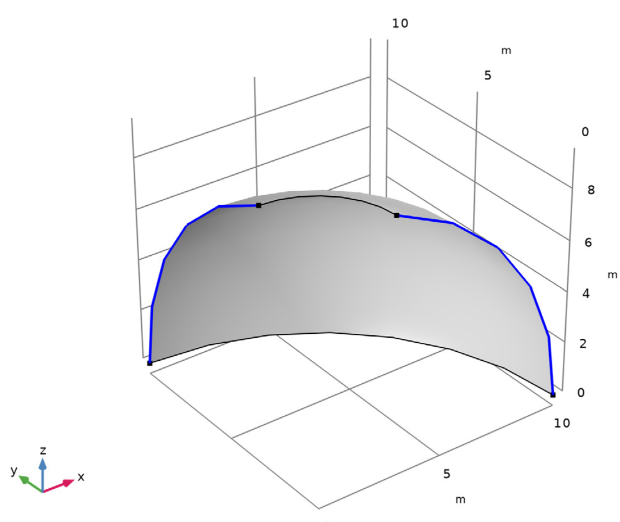
2 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.

3 In the d_0 text field, type `0.04`.

Symmetry 1

1 In the **Physics** toolbar, click **Edges** and choose **Symmetry**.

2 Select Edges 1 and 4 only.



Prescribed Displacement/Rotation 1

1 In the **Physics** toolbar, click **Points** and choose **Prescribed Displacement/Rotation**.

2 Select Point 4 only.

It might be easier to select the correct point by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)

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

4 From the **Displacement in z direction** list, choose **Prescribed**.

Point Load, X

1 In the **Physics** toolbar, click **Points** and choose **Point Load**.

2 In the **Settings** window for **Point Load**, type `Point Load, X` in the **Label** text field.

3 Select Point 4 only.

4 Locate the **Force** section. Specify the $\mathbf{F}_\mathrm{P}$ vector as

-100*para	x
0	y
0	z

Point Load, Y

1 In the **Physics** toolbar, click **Points** and choose **Point Load**.

2 In the **Settings** window for **Point Load**, type `Point Load, Y` in the **Label** text field.

3 Select Point 2 only.

4 Locate the **Force** section. Specify the $\mathbf{F}_\mathrm{P}$ vector as

0	x
100*para	y
0	z

MESH 1

Mapped 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.

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

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

Distribution 1

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

2 Select Edges 1 and 4 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

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

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

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

7 From the **Growth rate** list, choose **Exponential**.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 2 and 3 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

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

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

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

7 Select the **Symmetric distribution** checkbox.

8 From the **Growth rate** list, choose **Exponential**.

9 Click **Build All**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
para	0	0	Solver parameter

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

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

3 Select the **Include geometric nonlinearity** checkbox.

Set up an auxiliary continuation sweep for the **para** parameter.

4 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list
para (Solver parameter)	`range(0,0.1,1)`

7 Locate the **Study Settings** section. From the **Tolerance** list, choose **User controlled**.

8 In the **Relative tolerance** text field, type `0.0001`.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** node, then click **Fully Coupled 1**.

4 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

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

6 In the **Study** toolbar, click **Compute**.

RESULTS

Surface 1

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

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

3 Select the **Manual color range** checkbox.

4 In the **Maximum** text field, type `5e5`.

5 In the **Stress (shell)** toolbar, click **Plot**.

1D Plot Group 2

In the **Results** toolbar, click **1D Plot Group**.

Point Graph 1

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

2 Select Point 4 only.

3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `-u`.

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

6 In the **Expression** text field, type `para*100[N]`.

7 Click to expand the **Coloring and Style** section. From the **Width** list, choose **3**.

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

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

10 In the table, enter the following settings:

Legends
-u under x force

Point Graph 2

1 Right-click **Point Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, locate the **Selection** section.

3 Click to select the **Activate Selection** toggle button.

4 In the list box, select **4**.

5 Click **Remove from Selection**.

6 Select Point 2 only.

7 Locate the **y-Axis Data** section. In the **Expression** text field, type `v`.

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

Legends
v under y force

Displacement

1 In the **Model Builder** window, under **Results** click **1D Plot Group 2**.

2 In the **Settings** window for **1D Plot Group**, type `Displacement` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** checkbox. In the associated text field, type `Force (N)`.

6 Select the **y-axis label** checkbox. In the associated text field, type `Displacement under force (m)`.

7 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

8 In the **Displacement** toolbar, click **Plot**.

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

Evaluation Group 1

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, locate the **Data** section.

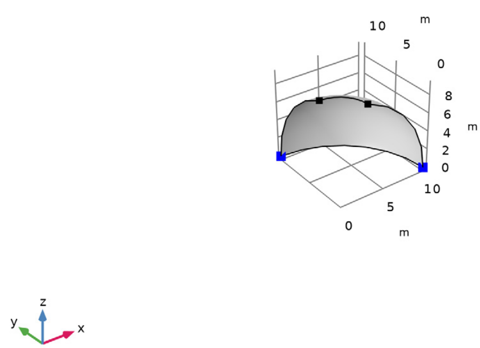
3 From the **Parameter selection (para)** list, choose **Last**.

4 Locate the **Transformation** section. Select the **Transpose** checkbox.

Point Evaluation 1

1 Right-click **Evaluation Group 1** and choose **Point Evaluation**.

2 Select Points 2 and 4 only.



3 In the **Settings** window for **Point Evaluation**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
u	m	Displacement field, X component
v	m	Displacement field, Y component

5 In the **Evaluation Group 1** toolbar, click **Evaluate**.