Created in COMSOL Multiphysics 6.2

Spherical Cap with Central Point Load

Introduction

Buckling is a phenomenon that can cause sudden failure of a structure. A linear buckling analysis predicts the critical buckling load. Such an analysis, however, does not give any information about what happens at loads higher than the critical load. Tracing the solution after the critical load is called a *postbuckling analysis*.

A spherical cap with a point load at its crown is a common example to study postbuckling analysis of 2D axisymmetric shells. The critical load, snap-through behavior, and softening and stiffening effects are the interesting aspects which are studied in this example.

In order to predict the postbuckling behavior, one need to use the nonlinear solver and ramp up the applied load to compute the structure deformation. The buckling load can then be based on when a certain, not acceptable, deformation is reached.

Once the critical buckling load has been reached, it can happen that the structure undergoes a sudden large deformation into a new stable configuration. This is known as a snap-through phenomenon. A snap-through phenomenon cannot be always simulated using prescribed load in a standard nonlinear static solver because the problem becomes numerically singular. In the current example, the displacement at the crown increases monotonically even if the load decreases after a critical point in the snap-through region. Thus, using displacement control is a useful strategy for this example.

Model Definition

The model studied here is a benchmark for a spherical cap subjected to a point load at its crown; see Ref. 1.

- The radius of the spherical cap is $a = 10$ m and the thickness is $th = 0.20384$ m. The sector angle of the spherical cap is $\pi/4$ radians.
- The edge/point which is not on axis of revolution is fixed.
- In the study the variation of the crown (center) axial displacement with respect to the applied load is of interest.

Due to the axial symmetry, only the part of the cap which is located at positive r-coordinates is modeled. The full geometry of the spherical cap with loading and boundary conditions is shown in Figure 1.

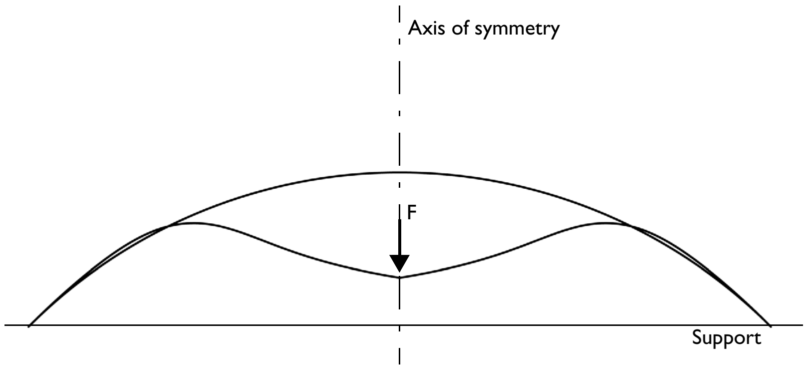


Figure 1: Problem description.

Results

For a spherical cap, the load versus displacement curve exhibits a critical load which is followed by a gradual snap and further increase in stiffness. Figure 2 and Figure 3 show the total displacement using the Solid Mechanics and Shell interfaces, respectively, at three different crown displacements. The annotations in the figures shows the corresponding point loads which closely match the benchmarked numerical solutions given in Ref. 1.

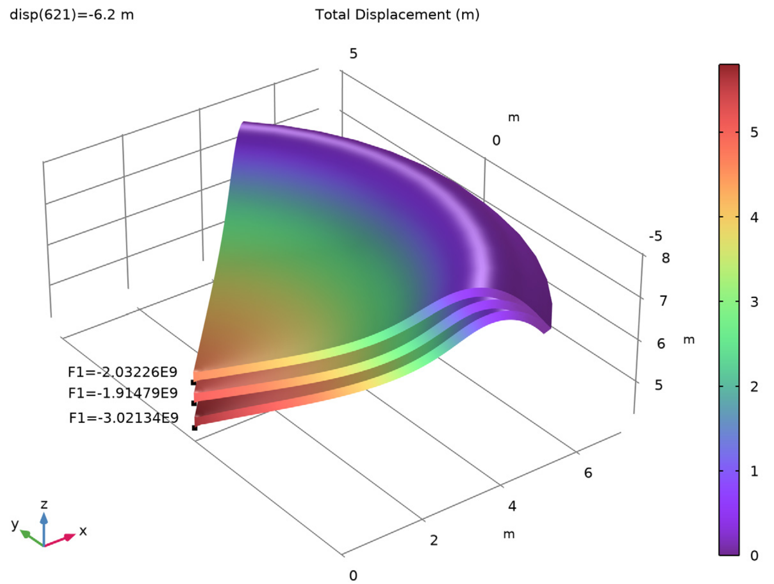


Figure 2: Total displacement computed in the Solid Mechanics Interface using 40 mesh elements.

What is important to note in the figures is the snap-through behavior and softening effect after the critical load. The top surface in both figures corresponds to the critical load, while the middle surface is corresponding to the load after the critical point. This shows that although deformation increases, the load decreases due to softening after the critical load. The third surface in both figures shows an increase in displacement with an increase in load, indicating an increase in stiffness after the snap through phase.

Figure 4 shows the variation of axial displacement at the crown of the spherical cap versus the applied load. For the Shell interface, three different discretizations (4, 8, 16 mesh elements) are used. For the Solid Mechanics interface 40 mesh elements are used. These discretizations are the same as in Ref. 1.

The results match the values in the reference quite closely. Note however, that these results are reported for certain discretizations and element formulations. There is no target value as such.

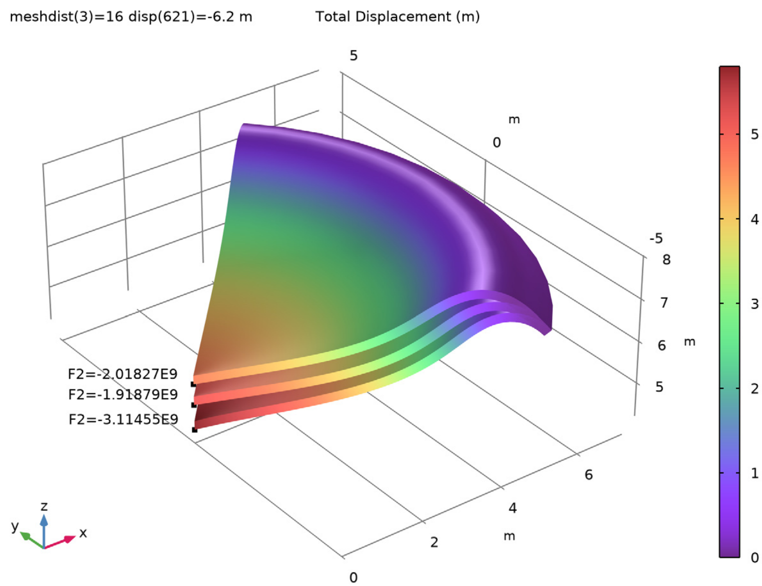


Figure 3: Total displacement computing in the Shell Interface using 16 mesh elements.

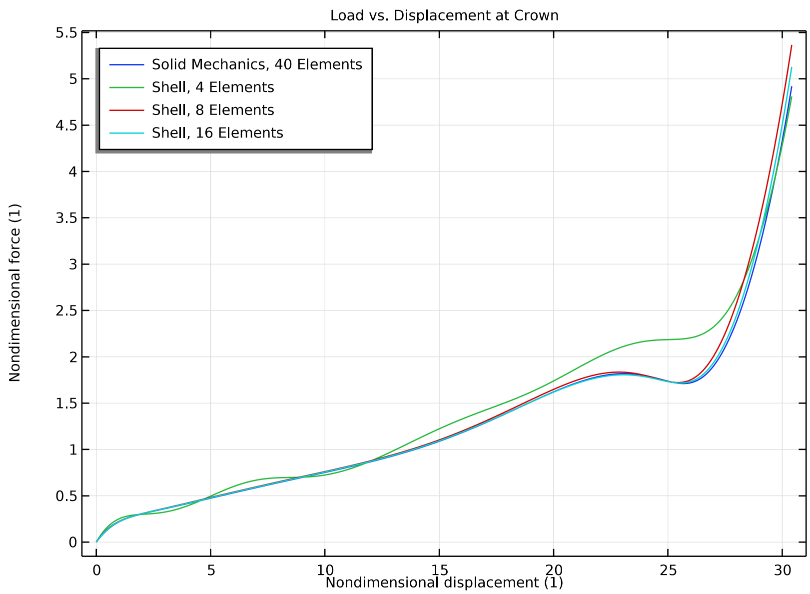


Figure 4: Applied load versus center displacement.

In Table 1, the results from the Solid Mechanics interface with 40 mesh elements are compared with the reference.

TABLE 1: SOLID MECHANICS IN NONDIMENSIONAL FORMAT.

Applied Load	**Displacement in reference**	**Displacement computed**
0.320	2.165	2.250
0.584	6.769	6.920
0.975	13.335	13.600
1.624	19.706	20.025
1.808	22.073	22.450
1.758	24.398	24.665
1.962	26.788	27.170
4.699	29.851	30.265

In Table 2, Table 3, and Table 4, the results from the Shell interface with 4, 8, and 16 mesh elements, respectively, are compared with the reference. Note that with only four elements, there is no snap through behavior, indicating that the mesh is much too coarse. This is experienced also in the reference, even though different types of shell element formulations are used.

TABLE 2: SHELL RESULTS WITH 4 ELEMENTS IN NONDIMENSIONAL FORMAT.

Applied Load	**Displacement target**	**Displacement computed**
0.335	2.367	3.100
0.579	6.921	5.940
0.920	11.614	12.665
1.176	16.423	14.850
1.705	18.964	20.300
2.488	21.393	27.850
2.540	23.659	28.050
3.765	28.541	29.870

TABLE 3: SHELL RESULTS WITH 8 ELEMENTS IN NONDIMENSIONAL FORMAT.

Applied Load	**Displacement target**	**Displacement computed**
0.332	2.326	2.440
0.580	6.720	6.775
0.994	13.642	13.760
1.502	18.487	18.815
1.757	20.887	21.240
1.678(1.722)	25.668	25.500
3.705	28.680	29.330

TABLE 4: SHELL RESULTS WITH 16 ELEMENTS IN NONDIMENSIONAL FORMAT.

Applied Load	**Displacement target**	**Displacement computed**
0.332	2.326	2.445
0.580	6.720	6.800
0.994	13.642	13.800
1.502	18.487	18.945
1.757	20.887	21.640
1.678(1.717)	25.668	25.500
3.705	28.680	29.410

Note that the lowest load after the critical load when using a shell formulation is 1.678 in the reference. This value is not reached in the solutions, where the lowest load is predicted as 1.722 and 1.717 with 8 and 16 elements, respectively. A refined Solid Mechanics model actually indicates that the current values computed here are more accurate than those reported in the reference.

Notes About the COMSOL Implementation

The main feature of this model is that a limit point instability occurs at the buckling load. Load control would not able to track the unstable solution paths after the limit point, so a displacement control is used since the displacement at the crown increases monotonically.

In this case, where the only load is a point load, it would be possible to directly prescribe the displacement in that point, and then measure the reaction force. If the load was more complex, for example a pressure load, that would not be possible. For this reason, a more general approach is shown here.

To employ a displacement control strategy, a point load at the crown is considered as a global degree of freedom and a global equation in terms of axial displacement at the crown is solved to get the point load value.

For a nonlinear problem experiencing a snap-through behavior, there is no general way to determine which controlling parameter to use, so it is necessary to use some physical insight. You need to find a quantity which is monotonically increasing to use as a controlling parameter.

Reference

1. P. Lyons and S. Holsgrove, *Finite Element Benchmarks For 2D Beams And Axisymmetric Shells Involving Geometric Non-Linearity,* NAFEMS, 2005.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/spherical_cap_with_central_point_load`

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 **2D Axisymmetric**.
2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Structural Mechanics>Shell (shell)**.
5 Click **Add**.
6 Click **Done**.

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
a	10[m]	10 m	Radius of cap
th	0.203840[m]	0.20384 m	Thickness of cap
EE	210e9[Pa]	2.1E11 Pa	Young's modulus
Nu	0.3	0.3	Poisson's ratio
Rho	7800	7800	Density
disp	0[m]	0 m	Displacement parameter
meshdist	4	4	Mesh distribution parameter

Define a set of nondimensional variables that will be useful in the postprocessing plots and evaluations.

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.
2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Fn1	-F1*a/(EE*th^3*2*pi)		Nondimensional force
wn1	-w/th		Nondimensional displacement
Fn2	-F2*a/(EE*th^3*2*pi)		Nondimensional force
wn2	-w2/th		Nondimensional displacement

GEOMETRY 1

Circle 1 (c1)

1. In the **Geometry** toolbar, click **Circle**.
2. In the **Settings** window for **Circle**, locate the **Object Type** section.
3. From the **Type** list, choose **Curve**.
4. Locate the **Size and Shape** section. In the **Sector angle** text field, type 45.
5. In the **Radius** text field, type a+th.
6. Click **Build Selected**.
7. Locate the **Rotation Angle** section. In the **Rotation** text field, type 45.
8. Click to expand the **Layers** section. In the table, enter the following settings:

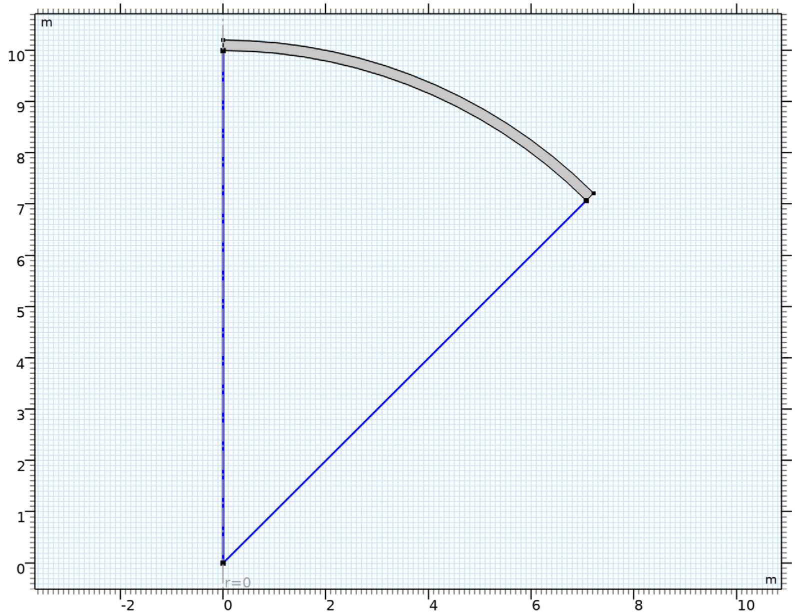
Layer name	Thickness (m)
Layer 1	th

9. 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 **c1**, select Boundaries 1 and 2 only.

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



4 In the **Settings** window for **Delete Entities**, click **Build Selected**.

Use the same material through a material link for the **Solid Mechanics** and **Shell** interfaces.

GLOBAL DEFINITIONS

Material 1 (mat1)

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

MATERIALS

Material Link 1 (matlnk1)

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

Material Link 2 (matlnk2)

1 Right-click **Materials** and choose **More Materials>Material Link**.

2 In the **Settings** window for **Material Link**, locate the **Geometric Entity Selection** section.

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

4 Select Boundary 3 only.

It might be easier to select the correct boundary 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.)

GLOBAL DEFINITIONS

Material 1 (mat1)

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Material 1 (mat1)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	EE	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	Nu	1	Young's modulus and Poisson's ratio
Density	rho	Rho	kg/m³	Basic

DEFINITIONS

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Point**.

4 Select Point 1 only.

5 Locate the **Advanced** section. From the **Method** list, choose **Summation over nodes**.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Fixed Constraint**.

2 Select Boundary 2 only.

Now add a global equation for a point load, so that the crown displacement equals the prescribed one. For that, you need to show advanced physics options.

3 Click the **Show More Options** button in the **Model Builder** toolbar.

4 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.

5 Click **OK**.

Global Equations 1 (ODE1)

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
F1	intop1(w)-disp	0	0	

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `force` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Force (N)**.

8 Click **OK**.

9 In the **Settings** window for **Global Equations**, locate the **Units** section.

10 Click **Select Source Term Quantity**.

11 In the **Physical Quantity** dialog box, type `disp` in the text field.

12 Click **Filter**.

13 In the tree, select **General>Displacement (m)**.

14 Click **OK**.

Point Load (on Axis) 1

1 In the **Physics** toolbar, click **Points** and choose **Point Load (on Axis)**.

2 Select Point 1 only.

3 In the **Settings** window for **Point Load (on Axis)**, locate the **Force** section.

4 From the F_z list, choose **State variable F1 (solid/ge1)**.

SHELL (SHELL)

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

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

3 Click **Clear Selection**.

4 Select Boundary 3 only.

In order to model the solid midplane using the Shell interface, assign a proper offset from the **Thickness and Offset** feature. The shell normal is pointing inward which can be verified in the postprocessing plot.

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 `th`.

4 From the **Position** list, choose **Top surface on boundary**.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Points** and choose **Fixed Constraint**.

2 Select Point 3 only.

Global Equations 1 (ODE2)

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
`F2`	`intop1(w2)-disp`	`0`	`0`	

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `force` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Force (N)**.

8 Click **OK**.

9 In the **Settings** window for **Global Equations**, locate the **Units** section.

10 Click **Select Source Term Quantity**.

11 In the **Physical Quantity** dialog box, type `disp` in the text field.

12 Click **Filter**.

13 In the tree, select **General>Displacement (m)**.

14 Click **OK**.

Point Load (on Axis) 1

1. In the **Physics** toolbar, click **Points** and choose **Point Load (on Axis)**.
2. Select Point 1 only.
3. In the **Settings** window for **Point Load (on Axis)**, locate the **Force** section.
4. From the F_z list, choose **State variable F2 (shell/ge1)**.

Use different **Mesh** nodes in order to use different discretizations for Solid Mechanics and Shell interfaces as given in the benchmark example.

MESH 2

In the **Mesh** toolbar, click **Add Mesh** and choose **Add Mesh**.

MESH: SOLID MECHANICS

1. In the **Model Builder** window, under **Component 1 (comp1)>Meshes** click **Mesh 1**.
2. In the **Settings** window for **Mesh**, type `Mesh: Solid Mechanics` in the **Label** text field.

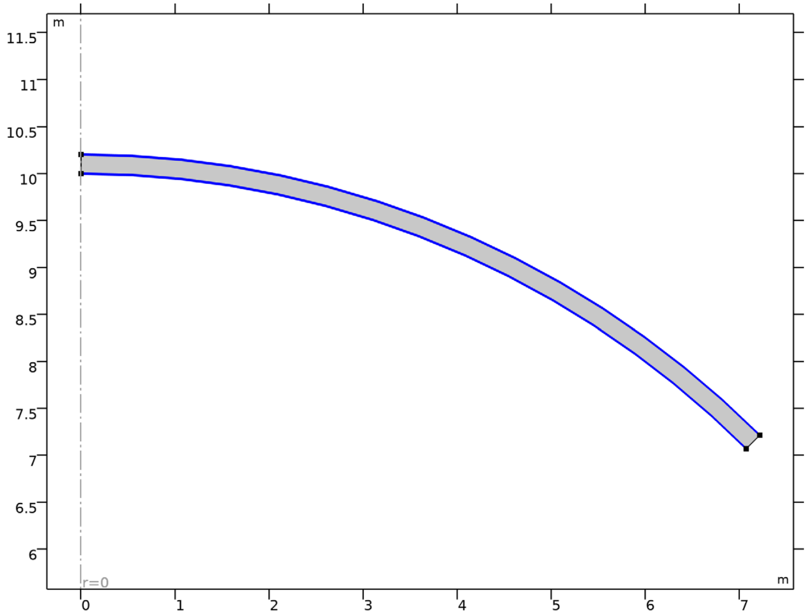
Mapped 1

In the **Mesh** toolbar, click **Mapped**.

Distribution 1

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

2 Select Boundaries 3 and 4 only.



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

4 In the **Number of elements** text field, type `40`.

5 Click **Build Selected**.

MESH: SHELL

1 In the **Model Builder** window, under **Component 1 (comp1)>Meshes** click **Mesh 2**.

2 In the **Settings** window for **Mesh**, type `Mesh: Shell` in the **Label** text field.

Edge 1

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

2 Select Boundary 3 only.

Distribution 1

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

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

3 In the **Number of elements** text field, type `meshdist`.

4 Click **Build Selected**.

Add a stationary study to the **Solid Mechanics** interface.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Shell (shell)**.

5 Click **Add Study** in the window toolbar.

6 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY: SOLID MECHANICS

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for **Study**, type `Study: Solid Mechanics` in the **Label** text field.

Step 1: Stationary

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

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

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

4 Click to expand the **Mesh Selection** section. In the table, enter the following settings:

Component	Mesh
Component 1	Mesh: Solid Mechanics

5 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

6 Click **Add**.

7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
disp (Displacement parameter)	`range(0,-0.01,-6.2)`	`m`

8 In the **Home** toolbar, click **Compute**.

Add a stationary study to the **Shell** interface. Parameterize the mesh discretization using a parametric sweep.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Solid Mechanics (solid)**.

5 Click **Add Study** in the window toolbar.

6 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY: SHELL

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Study: Shell` in the **Label** text field.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
meshdist (Mesh distribution parameter)	`4,8,16`	

Step 1: Stationary

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

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

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

4 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
disp (Displacement parameter)	`range(0,-0.01,-6.2)`	m

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

RESULTS

Revolution 2D

1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Revolution 2D**.

2 In the **Settings** window for **Revolution 2D**, click to expand the **Revolution Layers** section.

3 In the **Start angle** text field, type `45`.

4 In the **Revolution angle** text field, type `-90`.

Revolution 2D 2

1 In the **Model Builder** window, click **Revolution 2D 2**.

2 In the **Settings** window for **Revolution 2D**, locate the **Revolution Layers** section.

3 In the **Start angle** text field, type `45`.

4 In the **Revolution angle** text field, type `-90`.

Stress (solid)

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Plot Settings** section.

3 From the **Frame** list, choose **Material (R, PHI, Z)**.

In order to visualize the softening and stiffening effect after the critical point, generate a 3D displacement plot of the spherical cap at the critical point, and on the unstable and stable part of the equilibrium path after the critical point.

Total Displacement, 3D (solid)

1 In the **Model Builder** window, under **Results** click **Stress, 3D (solid)**.

2 In the **Settings** window for **3D Plot Group**, type `Total Displacement, 3D (solid)` in the **Label** text field.

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

4 In the **Title** text area, type `Total Displacement (m)`.

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Surface 1

1 In the **Model Builder** window, expand the **Total Displacement, 3D (solid)** node, then click **Surface 1**.

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

3 From the **Dataset** list, choose **Revolution 2D**.

4 From the **Parameter value (disp (m))** list, choose **-4.7**.

5 Locate the **Expression** section. In the **Expression** text field, type `solid.disp`.

6 Locate the **Coloring and Style** section. Click **Change Color Table**.

7 In the **Color Table** dialog box, select **Rainbow>SpectrumLight** in the tree.

8 Click **OK**.

Annotation 1

1 In the **Model Builder** window, expand the **Surface 1** node.

2 Right-click **Total Displacement, 3D (solid)** and choose **Annotation**.

3 In the **Settings** window for **Annotation**, locate the **Data** section.

4 From the **Dataset** list, choose **Revolution 2D**.

5 From the **Parameter value (disp (m))** list, choose **-4.7**.

6 Locate the **Annotation** section. In the **Text** text field, type `F1=eval(F1)`.

7 From the **Geometry level** list, choose **Global**.

8 Locate the **Position** section. In the **Z** text field, type `a-4.7`.

9 Click to expand the **Advanced** section. Locate the **Coloring and Style** section. From the **Anchor point** list, choose **Lower right**.

Surface 1

In the **Model Builder** window, right-click **Surface 1** and choose **Duplicate**.

Annotation 1

In the **Model Builder** window, right-click **Annotation 1** and choose **Duplicate**.

Surface 2

1 In the **Model Builder** window, click **Surface 2**.

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

3 From the **Parameter value (disp (m))** list, choose **-5.2**.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Annotation 2

1 In the **Model Builder** window, click **Annotation 2**.

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

3 From the **Parameter value (disp (m))** list, choose **-5.2**.

4 Locate the **Position** section. In the **Z** text field, type `a-5.2`.

Surface 2

In the **Model Builder** window, right-click **Surface 2** and choose **Duplicate**.

Annotation 2

In the **Model Builder** window, right-click **Annotation 2** and choose **Duplicate**.

Surface 3

1 In the **Model Builder** window, click **Surface 3**.

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

3 From the **Parameter value (disp (m))** list, choose **-5.8**.

Annotation 3

1 In the **Model Builder** window, click **Annotation 3**.

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

3 From the **Parameter value (disp (m))** list, choose **-5.8**.

4 Locate the **Position** section. In the **Z** text field, type `a-5.8`.

5 In the **Total Displacement, 3D (solid)** toolbar, click **Plot**.

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

Stress (shell)

1 In the **Model Builder** window, under **Results** click **Stress (shell)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Plot Settings** section.

3 From the **Frame** list, choose **Material (R, PHI, Z)**.

Total Displacement, 3D (shell)

1 In the **Model Builder** window, under **Results** click **Stress, 3D (shell)**.

2 In the **Settings** window for **3D Plot Group**, type `Total Displacement, 3D (shell)` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Total Displacement (m)`.

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Surface 1

1 In the **Model Builder** window, expand the **Total Displacement, 3D (shell)** node, then click **Surface 1**.

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

3 From the **Dataset** list, choose **Revolution 2D 2**.

4 From the **Parameter value (disp (m))** list, choose **-4.7**.

5 Locate the **Expression** section. In the **Expression** text field, type `shell.disp`.

6 Locate the **Coloring and Style** section. Click **Change Color Table**.

7 In the **Color Table** dialog box, select **Rainbow>SpectrumLight** in the tree.

8 Click **OK**.

Annotation 1

1 In the **Model Builder** window, expand the **Surface 1** node.
2 Right-click **Total Displacement, 3D (shell)** and choose **Annotation**.
3 In the **Settings** window for **Annotation**, locate the **Data** section.
4 From the **Dataset** list, choose **Revolution 2D 2**.
5 From the **Parameter value (disp (m))** list, choose **-4.7**.
6 Locate the **Annotation** section. In the **Text** text field, type `F2=eval(F2)`.
7 From the **Geometry level** list, choose **Global**.
8 Locate the **Position** section. In the **z** text field, type `a-4.7`.
9 Locate the **Coloring and Style** section. From the **Anchor point** list, choose **Lower right**.

Surface 1

In the **Model Builder** window, right-click **Surface 1** and choose **Duplicate**.

Annotation 1

In the **Model Builder** window, right-click **Annotation 1** and choose **Duplicate**.

Surface 2

1 In the **Model Builder** window, click **Surface 2**.
2 In the **Settings** window for **Surface**, locate the **Data** section.
3 From the **Parameter value (disp (m))** list, choose **-5.2**.
4 Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Annotation 2

1 In the **Model Builder** window, click **Annotation 2**.
2 In the **Settings** window for **Annotation**, locate the **Data** section.
3 From the **Parameter value (disp (m))** list, choose **-5.2**.
4 Locate the **Position** section. In the **z** text field, type `a-5.2`.

Surface 2

In the **Model Builder** window, right-click **Surface 2** and choose **Duplicate**.

Annotation 2

In the **Model Builder** window, right-click **Annotation 2** and choose **Duplicate**.

Surface 3

1 In the **Model Builder** window, click **Surface 3**.
2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Parameter value (disp (m))** list, choose **-5.8**.

Annotation 3

1 In the **Model Builder** window, click **Annotation 3**.

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

3 From the **Parameter value (disp (m))** list, choose **-5.8**.

4 Locate the **Position** section. In the **z** text field, type `a-5.8`.

5 In the **Total Displacement, 3D (shell)** toolbar, click **Plot**.

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

Visualize the shell thickness and normal orientation.

ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Add Predefined Plot** to open the **Add Predefined Plot** window.

2 Go to the **Add Predefined Plot** window.

3 In the tree, select **Study: Shell/Parametric Solutions 1 (sol3)>Shell>Thickness and Orientation (shell)**.

4 Click **Add Plot** in the window toolbar.

5 In the **Home** toolbar, click **Add Predefined Plot** to close the **Add Predefined Plot** window.

6 In the **Home** toolbar, click **Add Predefined Plot** to open the **Add Predefined Plot** window.

In order to better visualize the shell normal in the **Thickness and Orientation** plot, reduce the number of arrows.

RESULTS

Shell Local System

1 In the **Model Builder** window, expand the **Thickness and Orientation (shell)** node, then click **Shell Local System**.

2 In the **Settings** window for **Coordinate System Line**, locate the **Positioning** section.

3 In the **Number of points** text field, type `20`.

4 In the **Thickness and Orientation (shell)** toolbar, click **Plot**.

Plot a 1D curve showing the relationship between the axial displacement and the point load at the crown.

Load vs. Displacement at Crown

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

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

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

4 In the **Title** text area, type `Load vs. Displacement at Crown`.

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

Point Graph 1

1 Right-click **Load vs. Displacement at Crown** and choose **Point Graph**.

2 Select Point 1 only.

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

4 In the **Expression** text field, type `Fn1`.

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

6 In the **Expression** text field, type `wn1`.

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

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

9 In the table, enter the following settings:

Legends
`Solid Mechanics, 40 Elements`

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

Point Graph 2

1 In the **Model Builder** window, click **Point Graph 2**.

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

3 From the **Dataset** list, choose **Study: Shell/Parametric Solutions 1 (sol3)**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `Fn2`.

5 Locate the **x-Axis Data** section. In the **Expression** text field, type `wn2`.

6 Locate the **Legends** section. From the **Legends** list, choose **Evaluated**.

7 In the **Legend** text field, type `Shell, eval(meshdist) Elements`.

8 In the **Load vs. Displacement at Crown** toolbar, click **Plot**.

Load vs. Displacement at Crown

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

2 In the **Settings** window for **Evaluation Group**, type `Load vs. Displacement at Crown` in the **Label** text field.

Solid Mechanics, 40 Elements

1 Right-click **Load vs. Displacement at Crown** and choose **Point Evaluation**.
2 In the **Settings** window for **Point Evaluation**, type `Solid Mechanics, 40 Elements` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study: Solid Mechanics/ Solution 1 (sol1)**.
4 Select Point 1 only.
5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`disp`	`m`	`Displacement parameter`
`Fn1`	`1`	`Nondimensional force (Solid Mechanics, 40 Elements)`
`wn1`	`1`	`Nondimensional displacement (Solid Mechanics, 40 Elements)`

Shell, 4 Elements

1 In the **Model Builder** window, right-click **Load vs. Displacement at Crown** and choose **Point Evaluation**.
2 In the **Settings** window for **Point Evaluation**, type `Shell, 4 Elements` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study: Shell/ Parametric Solutions 1 (sol3)**.
4 From the **Parameter selection (meshdist)** list, choose **From list**.
5 In the **Parameter values (meshdist)** list, select **4**.
6 Select Point 1 only.
7 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`Fn2`	`1`	`Nondimensional force (Shell, 4 Elements)`
`wn2`	`1`	`Nondimensional displacement (Shell, 4 Elements)`

8 Right-click **Shell, 4 Elements** and choose **Duplicate**.

Shell, 8 Elements

1 In the **Model Builder** window, under **Results>Load vs. Displacement at Crown** click **Shell, 4 Elements 1**.

2 In the **Settings** window for **Point Evaluation**, type `Shell, 8 Elements` in the **Label** text field.

3 Locate the **Data** section. In the **Parameter values (meshdist)** list, select **8**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`Fn2`	`1`	`Nondimensional force (Shell, 8 Elements)`
`wn2`	`1`	`Nondimensional displacement (Shell, 8 Elements)`

5 Right-click **Shell, 8 Elements** and choose **Duplicate**.

Shell, 16 Elements

1 In the **Model Builder** window, under **Results>Load vs. Displacement at Crown** click **Shell, 8 Elements 1**.

2 In the **Settings** window for **Point Evaluation**, type `Shell, 16 Elements` in the **Label** text field.

3 Locate the **Data** section. In the **Parameter values (meshdist)** list, select **16**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`Fn2`	`1`	`Nondimensional force (Shell, 16 Elements)`
`wn2`	`1`	`Nondimensional displacement (Shell, 16 Elements)`

Load vs. Displacement at Crown

1 In the **Model Builder** window, click **Load vs. Displacement at Crown**.

2 In the **Settings** window for **Evaluation Group**, click to expand the **Format** section.

3 From the **Include parameters** list, choose **Off**.

4 In the **Load vs. Displacement at Crown** toolbar, click **Evaluate**.