Created in COMSOL Multiphysics 5.6

Postbuckling Analysis of a Hinged Cylindrical Shell

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 linear buckling analysis also often overpredicts the load-carrying capacity of the structure.

In order to accurately determine the critical buckling load or predict the postbuckling behavior, you can 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 process cannot be simulated using prescribed load in a standard nonlinear static solver because the problem becomes numerically singular. Physically speaking, it is a highly transient problem as the structure “jumps” from one state to another. For simple cases with a single point load, it is often possible to replace the point load with a prescribed displacement and then measure the reaction force instead.

For more general problems the post-buckling solution must however be tracked using more sophisticated methods, as shown in this example.

Figure 1 shows the variation of load versus the displacement for such a difficult case. It illustrates the possible computational problem by using either a load control (path A) or a displacement control (path B).

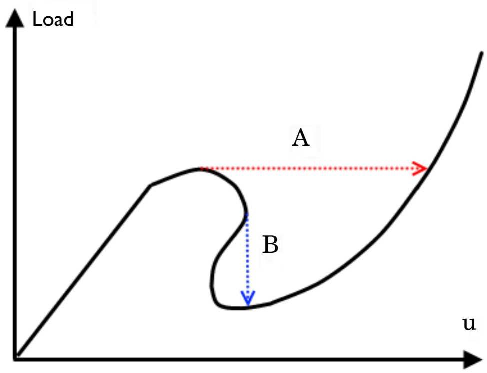

Figure 1: Load versus displacement in snap-through buckling

The shell structure in this example has a behavior similar to this.

Model Definition

The model studied here is a benchmark for a hinged cylindrical panel subjected to a point load at its center; see Ref. 1.

- The radius of the cylinder is $R = 2.54$ m and all edges have a length of $2L = 0.508$ m. The angular span of the panel is thus 0.2 radians. The panel thickness is $th = 6.35$ mm.
- The straight edges are hinged.
- In the study the variation of the panel center vertical displacement with respect to the change of the applied load is of interest.

Due to the double symmetry, only one quarter of the geometry is modeled as shown in Figure 2. The blue lines show the symmetry edge conditions, while the red line shows the location of the hinged edge condition.

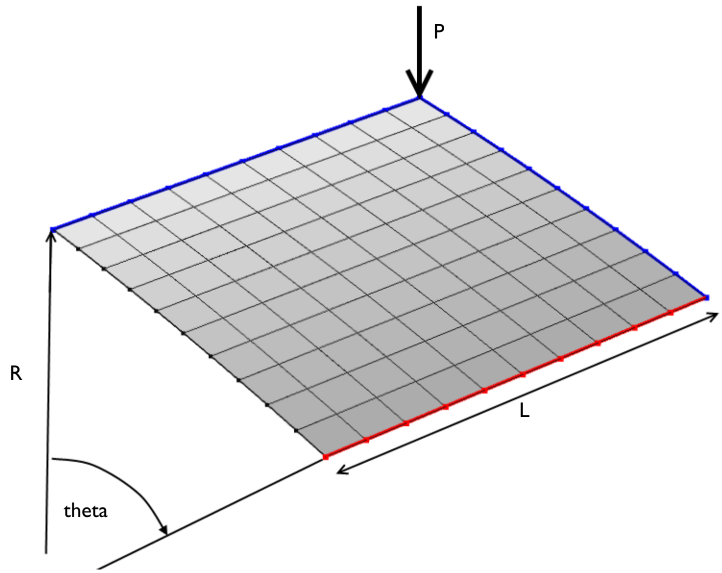

Figure 2: Problem description.

In general, you should be careful with using symmetry in buckling problems, because nonsymmetric solutions may exist.

Results

In Figure 3 you can see the applied load as a function of the panel center displacement. The figure shows clearly a non-unique solution for a given applied load (between -400 N to 600 N) or a given displacement (between 14.4 mm and 17 mm).

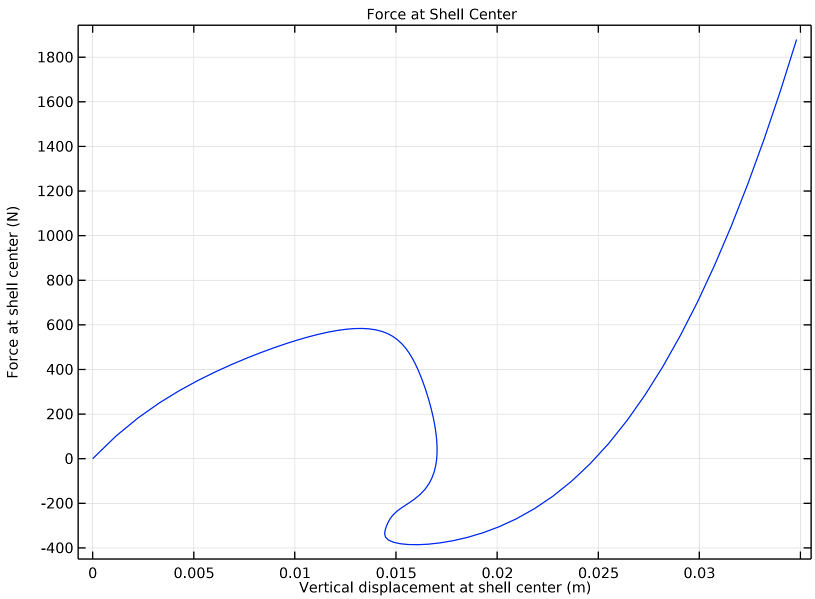

Figure 3: Applied load versus panel center displacement.

As shown in Table 1, the results agree well with the target data from Ref. 1.

TABLE 1: COMPARISON BETWEEN TARGET AND COMPUTED DATA.

Applied Load (N)	**Displacement target (mm)**	**Displacement computed (mm)**	**Difference (%)**
155.1	1.846	1.818	1.52
574.2	11.904	12.05	1.23
485.1	15.501	15.56	0.38
24.9	17.008	17.028	0.12
-300.3	14.520	14.537	0.12
-381.3	16.961	16.77	1.13
-1.8	24.824	24.81	0.06
1469.4	33.388	33.34	0.14

Notes About the COMSOL Implementation

The main feature of this model is that a limit point instability occurs at the buckling load. Neither a load control, nor a point displacement control, would be able to track the jump between the stable solution paths (see Figure 1). To solve this type of problem it is important to find a proper parameter that increases monotonically.

In this example, a good such parameter is the average of the displacement in the direction of the applied force. You use a nonlocal average coupling to measure the displacement and then add a global equation to compute the appropriate point load for each prescribed parameter value.

There is no general way to determine which controlling parameter to use, so it is necessary to use some physical insight.

Reference

1. K.Y. Sze, X.H. Liua, and S.H. Lob, "Popular Benchmark Problems for Geometric Nonlinear Analysis of Shells," *Finite Element in Analysis and Design*, vol. 40, issue 11, pp. 1551–1569, 2004.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/postbuckling_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**.

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
`R`	`2540[mm]`	2.54 m	`Panel radius`
`L`	`254[mm]`	0.254 m	`Panel length`
`thic`	`6.35[mm]`	0.00635 m	`Panel thickness`
`theta`	`0.1[rad]`	0.1 rad	`Panel section angle`
`E0`	`3.103[GPa]`	3.103E9 Pa	`Young's modulus`
`nu0`	`0.3`	0.3	`Poisson's ratio`
`disp`	`0`	0	`Displacement parameter`

GEOMETRY 1

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **xz-plane**.

4 Click **Show Work Plane**.

Work Plane 1 (wp1)>Line Segment 1 (ls1)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 Locate the **Starting Point** section. In the **yw** text field, type `R`.

6 Locate the **Endpoint** section. In the **xw** text field, type `L` and **yw** to `R`.

7 Click **Build Selected**.

Revolve 1 (rev1)

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

2 In the **Settings** window for **Revolve**, locate the **Revolution Angles** section.

3 Click the **Angles** button.

4 In the **End angle** text field, type `theta`.

5 Locate the **Revolution Axis** section. Find the **Direction of revolution axis** subsection. In the **xw** text field, type `1`.

6 In the **yw** text field, type `0`.

7 Click **Build Selected**.

DEFINITIONS

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

Average 1 (aveop1)

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

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

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

4 Select Boundary 1 only.

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 4 only.

Variables 1

1 In the **Definitions** toolbar, click **Local Variables**.

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

3 In the table, enter the following settings:

Name	**Expression**	**Unit**
w_center	-intop1(w)	m

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 text field, type `thic`.

Linear Elastic Material 1

1 In the **Model Builder** window, click **Linear Elastic Material 1**.

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

3 From the E list, choose **User defined**. In the associated text field, type `E0`.

4 From the ν list, choose **User defined**. In the associated text field, type `nu0`.

Symmetry 1

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

2 Select Edge 3 only.

Symmetry 2

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

2 Select Edge 4 only.

3 In the **Settings** window for **Symmetry**, locate the **Coordinate System Selection** section.

4 From the **Coordinate system** list, choose **Global coordinate system**.

5 Locate the **Symmetry** section. From the **Axis to use as symmetry plane normal** list, choose **1**.

Pinned 1

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

2 Select Edge 2 only.

Point Load 1

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

2 Select Point 4 only.

Apply 1/4th of the total load because of the double symmetry used in this model.

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

4 Specify the $\mathbf{F}_\mathrm{P}$ vector as

0	x
0	y
-P/4	z

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

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

7 Click **OK**.

Global Equations 1

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
P	aveop1(-w)-disp	0	0	Force at shell center

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 `displacement` in the text field.

12 Click **Filter**.

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

14 Click **OK**.

MESH 1

Mapped 1

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

2 Select Boundary 1 only.

Distribution 1

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

2 Select Edges 1 and 2 only.

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

4 In the **Number of elements** text field, type `10`.

5 Click **Build Selected**.

STUDY 1

Step 1: Stationary

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

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

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list
disp (Displacement parameter)	`range(0,2e-4,1)`

6 Locate the **Study Settings** section. Select the **Include geometric nonlinearity** check box.

Sometimes it is not straightforward to guess the maximum value of the parameter used. You can then instead set a stop condition for the parametric solver based on something that is known.

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.

4 Right-click **Parametric 1** and choose **Stop Condition**.

5 In the **Settings** window for **Stop Condition**, locate the **Stop Expressions** section.

6 Click **Add**.

7 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
`comp1.w_center>0.035`	True (>=1)	√	`Stop expression 1`

Specify that the solution is to be stored just before the stop condition is reached.

8 Locate the **Output at Stop** section. From the **Add solution** list, choose **Step before stop**.

9 Clear the **Add warning** check box.

10 In the **Model Builder** window, click **Stationary Solver 1**.

11 In the **Settings** window for **Stationary Solver**, click to expand the **Output** section.

12 Clear the **Reaction forces** check box.

13 Click **Compute**.

RESULTS

Force at Shell Center

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

2 In the **Settings** window for **1D Plot Group**, type `Force at Shell Center` 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 `Force at Shell Center`.

Point Graph 1

1 Right-click **Force at Shell Center** and choose **Point Graph**.

2 Select Point 4 only.

3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Shell>P - Force at shell center - N**.

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

5 In the **Expression** text field, type `w_center`.

6 Select the **Description** check box.

7 In the associated text field, type `Vertical displacement at shell center`.

8 In the **Force at Shell Center** toolbar, click **Plot**.