



Model created in COMSOL Multiphysics 6.4

# Buckling Analysis of a Corrugated Conical Shell

## Introduction

---

This tutorial model demonstrates how to perform a linear buckling analysis using cyclic symmetry with postprocessing on the full geometry. A conical shell with sixteen identical corrugations can be divided into sixteen sectors of symmetry. The model computes the critical load factor for the first buckling mode for the full shell geometry and compares it to the value computed for a single sector with the cyclic symmetry boundary conditions applied on two sector boundaries. The results for one sector are in very good agreement with the computations on the full geometry, while the memory requirements are significantly reduced.

## Model Definition

---

Figure 1 shows the shell geometry.

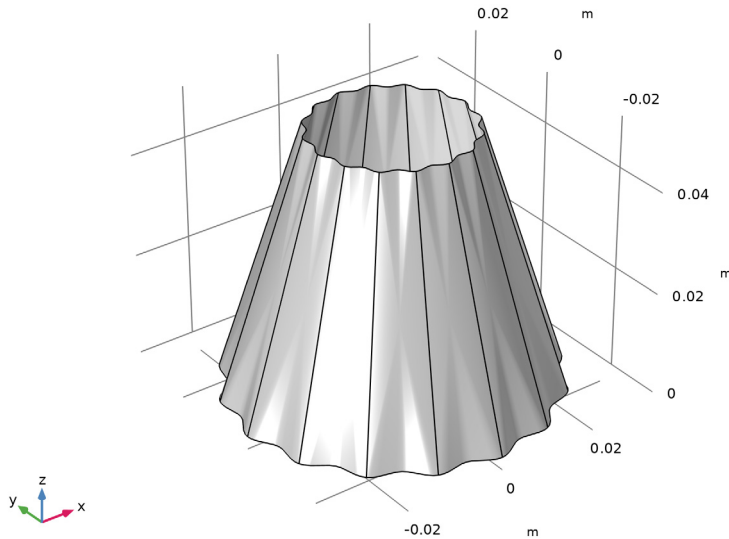
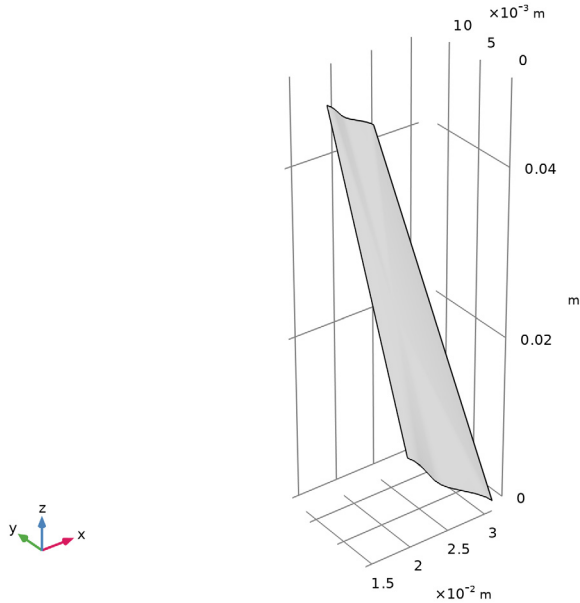


Figure 1: Full shell geometry.

The base radius is 3 cm, and the corrugation size is 1 mm. The structure height is 5 cm, and the shell thickness is 0.5 mm. The thickness is used as a modeling parameter in this tutorial to show its influence on the periodicity of the 1st buckling mode of the structure.

The geometry can be divided into sixteen identical parts, each represented by a sector with an angle  $\theta_s = \pi/8$  with respect to rotation around the  $z$ -axis; see [Figure 2](#)



*Figure 2: Sector geometry.*

The shell is made of aluminum. The bottom edge is fixed, and the load is applied vertically downward on the top edge.

### *Results and Discussion*

[Figure 3](#) shows the critical load factors computed using a unit sector for three different values and the shell thickness and different values of the azimuthal mode number. The latter parameter controls the periodicity with respect to the full geometry of the buckling mode computed using cyclic symmetry conditions. Thus, the angle of periodicity is

$\phi = m_a \theta$ , where the azimuthal mode number  $m_a$  can vary from 0 to  $N_s/2$ , with  $N_s$  being the total number of sectors so that  $\theta_s = 2\pi/N_s$ .

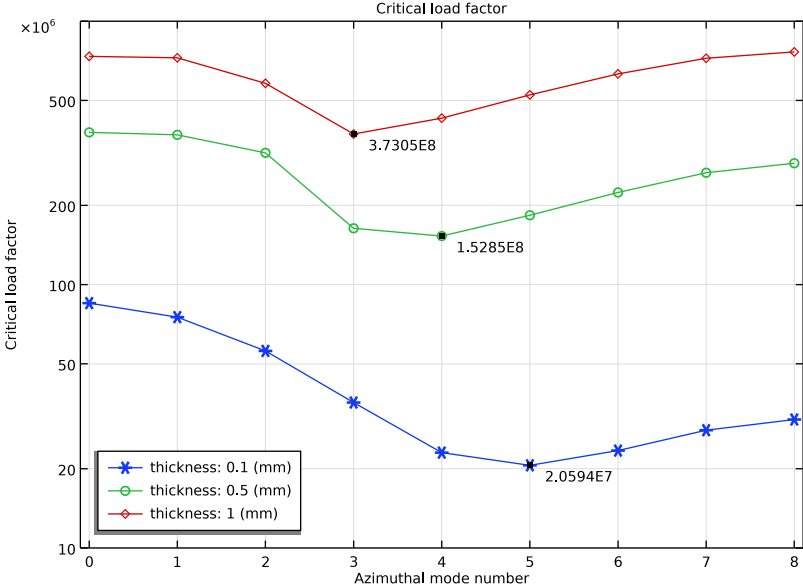
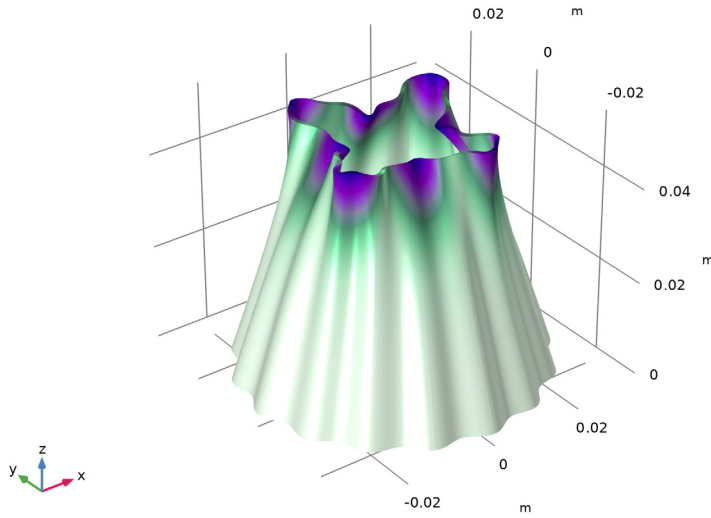


Figure 3: Critical load factor.

For example, [Figure 4](#) shows the buckling mode computed using a unit sector for a shell thickness of 0.5 mm and azimuthal number  $m_a = 4$ .

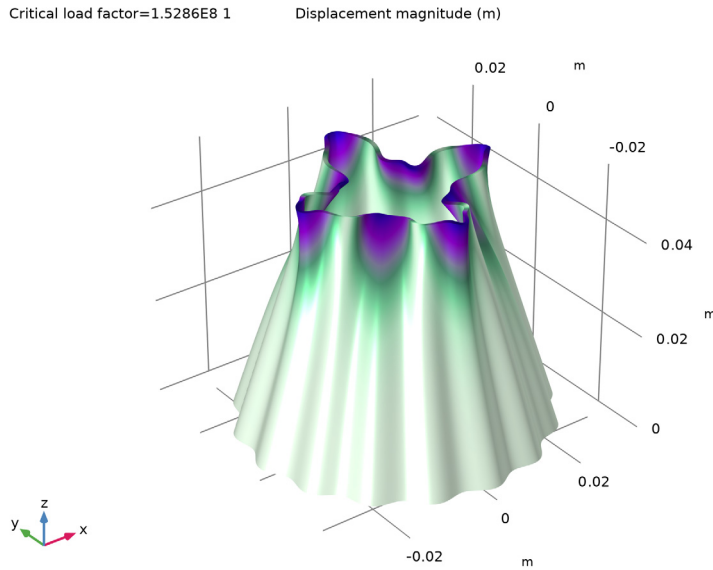
th(2)=0.5 mm Critical load factor=1.5285E8 1, ma=4 Displacement magnitude (m)



*Figure 4: The first buckling mode with periodicity of four computed using a single sector and visualized on full geometry.*

In [Figure 3](#), the minimum value of the critical load factor computed for each given shell thickness indicates the symmetry type for the 1st buckling mode on the full geometry.

Thus, [Figure 5](#) show the 1st buckling mode computed on the full geometry for shell thickness of 0.5 mm.



*Figure 5: The first buckling mode computed using full geometry.*

Note that the modes computed using a unit sector ([Figure 4](#)) and the full geometry ([Figure 4](#)) differ only by a rigid body rotation around the  $z$ -axis; something that cannot be controlled during the full geometry buckling eigenvalue analysis. The computed critical load factors are in very good agreement with each other.

### *Notes About the COMSOL Implementation*

---

To set up the cyclic symmetry conditions, you use the predefined functionality available in COMSOL Multiphysics within the **Shell** physics interface under the **Periodic Condition** boundary feature. This imposes the proper boundary coupling condition on the sector side boundaries. As the orientation of the edge normals needs to be specified, you add a cylindrical coordinate system to do that.

You visualize the results computed for one sector over the full geometry by making use of a predefined type of derived dataset called **Sector 3D**. Such a data set will be added automatically when the physics sets up cyclic symmetry using a **Periodic Condition** node.

Note that in general it is not possible to know the azimuthal mode number of the critical buckling load. Thus, you always have to perform a sweep to find the lowest value, as shown in this example.

The solution time for a sector is, however, much smaller than the solution time for the full geometry. So even though you have to solve the sector model several times using a parametric sweep, the total solution time will still be smaller than for the full model. The memory requirements when using the sector symmetry approach are much smaller than when using the full model.

---

**Application Library path:** Structural\_Mechanics\_Module/  
Buckling\_and\_Wrinkling/buckling\_corrugated\_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 **Preset Studies for Selected Physics Interfaces > Linear Buckling**.
- 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
Ns	16	16	Number of sectors
hc	1[mm]	0.001 m	Corrugation size
Rc	3[cm]	0.03 m	Base radius
thetaS	2*pi/Ns	0.3927	Sector angle (rad)
Lc	5[cm]	0.05 m	Heights
th	0.5[mm]	5E-4 m	Shell thickness
ma	4	4	Azimuthal mode number
sf	0.5	0.5	Scaling factor for inclination

Create the sector geometry.

### GEOMETRY I

*Work Plane 1 (wp1)*

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

*Work Plane 1 (wp1) > Plane Geometry*

In the **Model Builder** window, click **Plane Geometry**.

*Work Plane 1 (wp1) > Parametric Curve 1 (pc1)*

1 In the **Work Plane** toolbar, click  **More Primitives** and choose **Parametric Curve**.

2 In the **Settings** window for **Parametric Curve**, locate the **Parameter** section.

3 In the **Maximum** text field, type thetaS.

4 Locate the **Expressions** section. In the **xw** text field, type  $(Rc+hc*\cos(Ns*s))*\cos(s)$ .

5 In the **yw** text field, type  $(Rc+hc*\cos(Ns*s))*\sin(s)$ .

*Extrude 1 (ext1)*

1 In the **Model Builder** window, under **Component 1 (comp1) > Geometry 1** right-click **Work Plane 1 (wp1)** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (m)
Lc

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

Scales xw	Scales yw
sf	sf

5 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

6 In the **New Cumulative Selection** dialog, type Sector in the **Name** text field.

7 Click **OK**.

8 In the **Settings** window for **Extrude**, click  **Build Selected**.

Also create the full geometry of the structure for verification computations.

*Rotate 1 (rot1)*

1 In the **Geometry** toolbar, click  **Transforms** and choose **Rotate**.

2 Select the object **ext1** only.


3 In the **Settings** window for **Rotate**, locate the **Input** section.

4 Select the **Keep input objects** checkbox.

5 Locate the **Rotation** section. In the **Angle** text field, type  $360/Ns * \text{range}(1, 1, Ns - 1)$ .

6 Locate the **Selections on Input Objects** section. Clear the **Propagate selections to resulting objects** checkbox.

7 Click  **Build All Objects**.

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

#### ADD MATERIAL FROM LIBRARY

In the **Home** toolbar, click  **Windows** and choose **Add Material from Library**.

#### ADD MATERIAL


1 Go to the **Add Material** window.

2 In the tree, select **Built-in > Aluminum**.

3 Right-click and choose **Add to Component 1 (comp1)**.

#### MATERIALS


*Aluminum (mat1)*

Click the  **Go to XY View** button in the **Graphics** toolbar.


Create a mapped mesh for the sector.

## 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 **Sector**.


### *Distribution 1*

- 1 Right-click **Mapped 1** and choose **Distribution**.
- 2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 3 In the **Number of elements** text field, type 8.
- 4 Locate the **Edge Selection** section. From the **Selection** list, choose **Sector**.
- 5 Locate the **Distribution** section. Select the **Equidistant** checkbox.
- 6 Click  **Build All**.

Add a cylindrical coordinate system which you will use in the periodic boundary conditions to specify the edge orientation.

## DEFINITIONS

### *Cylindrical System 2 (sys2)*

- 1 In the **Definitions** toolbar, click  **Coordinate Systems** and choose **Cylindrical System**.
- 2 In the **Settings** window for **Cylindrical System**, locate the **Coordinate Names** section.
- 3 From the **Frame** list, choose **Material (X, Y, Z)**.

Set up the physics on the sector.

## 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 From the **Selection** list, choose **Sector**.

### *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.

### *Fixed Constraint I*

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

Select the bottom edge of the sector to be fixed.

2 Select Edge 48 only.

### *Edge Load I*

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

Specify the buckling load.

Select the top edge of the sector.

2 Select Edge 42 only.

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

4 From the **Load type** list, choose **Force per reference area**.

5 Specify the  $\mathbf{f}_A$  vector as

-1 [N/m<sup>2</sup>] z

It is conventional to use a unit load, but the load value is in fact arbitrary. The actual buckling load will be a product of this value and the critical load factor computed during the buckling study.

### *Periodic Condition I*

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

Select the side edges of the sector.

2 Select Edges 43 and 44 only.

3 Right-click **Periodic Condition I** and choose **Manual Destination Selection**.

4 In the **Settings** window for **Periodic Condition**, locate the **Destination Selection** section.

5 In the list box, select **44**.

6 Click  **Clear Selection**.

7 Click  **Paste Selection**.

8 In the **Paste Selection** dialog, type 43 in the **Selection** text field.

9 Click **OK**.

It is conventional to use a unit load, but the load value is in fact arbitrary. The actual buckling load will be a product of this value and the critical load factor computed during the buckling study.

10 In the **Settings** window for **Periodic Condition**, locate the **Periodicity Settings** section.

11 From the **Type of periodicity** list, choose **Cyclic symmetry**.

12 In the  $\theta_S$  text field, type  $\theta a_S$ .

13 In the  $m$  text field, type  $ma$ .

Use the cylindrical coordinate system to specify the orientation of the source and destination edges.

14 Click to expand the **Orientation of Source** section. From the

**Transform to intermediate map** list, choose **Cylindrical System 2 (sys2)**.

15 Click to expand the **Orientation of Destination** section. From the

**Transform to intermediate map** list, choose **Cylindrical System 2 (sys2)**.


Set up a parametric sweep to find the first buckling modes for shell of various thickness.

### STUDY 1: SECTOR

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

2 In the **Settings** window for **Study**, type Study 1: Sector 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
th (Shell thickness)	0.1 0.5 1	mm

#### Step 2: Linear Buckling

1 In the **Model Builder** window, click **Step 2: Linear Buckling**.


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

3 Select the **Auxiliary sweep** checkbox.

4 Click **+ Add**.

5 In the table, enter the following settings:


Parameter name	Parameter value list	Parameter unit
ma (Azimuthal mode number)	range (0, 1, $N_s/2$ )	

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

Plot all computed buckling critical load factors.

## RESULTS

### *Critical load factor*

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type **Critical load factor** 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 **Critical load factor**.
- 5 Locate the **Axis** section. Select the **y-axis log scale** checkbox.
- 6 Locate the **Grid** section. Select the **Manual spacing** checkbox.
- 7 Locate the **Axis** section. Select the **Manual axis limits** checkbox.
- 8 In the **x minimum** text field, type **-0.1**.
- 9 In the **x maximum** text field, type **8.1**.
- 10 In the **y minimum** text field, type **1E7**.
- 11 In the **y maximum** text field, type **1E9**.
- 12 Locate the **Legend** section. From the **Position** list, choose **Lower left**.

### *Global I*

- 1 Right-click **Critical load factor** and choose **Global**.
- 2 In the **Settings** window for **Global**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Study I: Sector/Parametric Solutions I (sol3)**.
- 4 From the **Parameter selection (th)** list, choose **From list**.
- 5 In the **Parameter values (th (mm))** list box, select **0.1**.
- 6 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component I (comp1) > Shell > shell.LFcrit - Critical load factor**.
- 7 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Inner solutions**.
- 8 From the **Parameter** list, choose **Expression**.
- 9 In the **Expression** text field, type **ma**.
- 10 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.
- 11 Click to expand the **Legends** section. Find the **Include** subsection. Clear the **Solution** checkbox.
- 12 Clear the **Description** checkbox.

**13** Find the **Prefix and suffix** subsection. In the **Suffix** text field, type thickness: `eval(th, mm) (mm)`.

**14** In the **Critical load factor** toolbar, click  **Plot**.


#### *Graph Marker 1*

- 1** Right-click **Global 1** and choose **Graph Marker**.
- 2** In the **Settings** window for **Graph Marker**, locate the **Display** section.
- 3** From the **Display** list, choose **Min**.
- 4** Locate the **Text Format** section. In the **Precision** text field, type 5.

#### *Global 2*


- 1** In the **Model Builder** window, under **Results > Critical load factor** right-click **Global 1** and choose **Duplicate**.
- 2** In the **Settings** window for **Global**, locate the **Data** section.
- 3** In the **Parameter values (th (mm))** list box, select **0.5**.

#### *Global 3*

- 1** Right-click **Global 2** and choose **Duplicate**.
- 2** In the **Settings** window for **Global**, locate the **Data** section.
- 3** In the **Parameter values (th (mm))** list box, select **1**.
- 4** In the **Critical load factor** toolbar, click  **Plot**.

The software automatically generates a special sector data set to visualize the periodic solution for a full geometry. Inspect the case, for which the shell thickness is 0.5 mm, and azimuthal mode number is four.


#### *Mode Shape, Full Geometry (shell)*

- 1** In the **Model Builder** window, under **Results** click **Mode Shape, Full Geometry (shell)**.
- 2** In the **Settings** window for **3D Plot Group**, locate the **Data** section.
- 3** From the **Parameter value (th (mm))** list, choose **0.5**.
- 4** From the **Parameter value (ma)** list, choose **4**.
- 5** Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.
- 6** In the **Mode Shape, Full Geometry (shell)** toolbar, click  **Plot**.

You use this case to compare the result with that computed using the true full geometry for the same shell thickness.

Add one more Shell physics interface, a new mesh, and a new buckling study to compute using the full geometry.

## ADD PHYSICS


- 1 In the **Home** toolbar, click  **Windows** and choose **Add Physics**.
- 2 Go to the **Add Physics** window.
- 3 In the tree, select **Structural Mechanics > Shell (shell)**.
- 4 Click the **Add to Component 1** button in the window toolbar.

## SHELL 2 (SHELL2)

### *Thickness and Offset 1*


- 1 In the **Settings** window for **Thickness and Offset**, locate the **Thickness and Offset** section.
- 2 In the  $d_0$  text field, type th.

### *Fixed Constraint 1*

- 1 In the **Physics** toolbar, click  **Edges** and choose **Fixed Constraint**.  
Select all bottom edges to be fixed.
- 2 Select Edges 1, 2, 4, 6, 8, 10, 17, 19, 25, 29, 38, 39, and 45–48 only.

Specify the buckling load.

### *Edge Load 1*


- 1 In the **Physics** toolbar, click  **Edges** and choose **Edge Load**.  
Select all top edges.
- 2 Select Edges 12–15, 20–23, 26, 28, 30, 32, 34, 36, 40, and 42 only.
- 3 In the **Settings** window for **Edge Load**, locate the **Force** section.
- 4 From the **Load type** list, choose **Force per reference area**.
- 5 Specify the  $\mathbf{f}_A$  vector as

$-1[\text{N}/\text{m}^2]$	$\mathbf{z}$
---------------------------	--------------





## MESH 2

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

### *Reference 1*


- 1 In the **Mesh** toolbar, click  **Modify** and choose **Reference**.
- 2 In the **Settings** window for **Reference**, locate the **Reference** section.
- 3 From the **Mesh** list, choose **Mesh 1**.

### *Copy Face 1*

- 1 In the **Mesh** toolbar, click  **Copy** and choose **Copy Face**.
- 2 In the **Settings** window for **Copy Face**, locate the **Source Boundaries** section.
- 3 From the **Selection** list, choose **Sector**.
- 4 Locate the **Destination Boundaries** section. Click to select the  **Activate Selection** toggle button.  
Select all other boundaries as destination.
- 5 Click  **Paste Selection**.
- 6 In the **Paste Selection** dialog, type 1-15 in the **Selection** text field.
- 7 Click **OK**.
- 8 In the **Settings** window for **Copy Face**, click  **Build All**.

Add a new study to compute using the full geometry.

### **ADD STUDY**

- 1 In the **Home** toolbar, click  **Windows** and choose **Add Study**.
- 2 Go to the **Add Study** window.
- 3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces > Linear Buckling**.
- 4 Right-click and choose **Add Study**.


### **STUDY 2**

#### *Step 1: Stationary*

- 1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.
- 2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
- 3 Select the **Modify model configuration for study step** checkbox.
- 4 In the tree, select **Component 1 (comp1) > Shell (shell)**.
- 5 Right-click and choose **Disable in Model**.



#### *Step 2: Linear Buckling*

- 1 In the **Model Builder** window, click **Step 2: Linear Buckling**.
- 2 In the **Settings** window for **Linear Buckling**, locate the **Physics and Variables Selection** section.
- 3 Select the **Modify model configuration for study step** checkbox.
- 4 In the tree, select **Component 1 (comp1) > Shell (shell)**.

- 5 Right-click and choose **Disable in Model**.
- 6 In the **Model Builder** window, click **Study 2**.
- 7 In the **Settings** window for **Study**, type Study 2: Verification in the **Label** text field.
- 8 In the **Study** toolbar, click  **Compute**.

## RESULTS

*Mode Shape, Verification (shell2)*

- 1 In the **Settings** window for **3D Plot Group**, type Mode Shape, Verification (shell2) in the **Label** text field.
- 2 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.
- 3 In the **Mode Shape, Verification (shell2)** toolbar, click  **Plot**.
- 4 Click the  **Go to Default View** button in the **Graphics** toolbar.  
Compare the result to that computed using a single sector.