Instability of Two Contacting Arches

Introduction

This conceptual example shows how to calculate critical points in models with contact. The model consists of two contacting arches modeled with the Shell interface. During loading, the lower arch exhibits a snap-through behavior. The definition of the problem is based on a benchmark example from Ref. 1.

Model Definition

The model geometry consists of an arch and a block as shown in Figure 1. Since the arches are modeled with the Shell interface, a 3D geometry is used. However, a 2D plane strain behavior is intended, and consequently symmetry conditions are applied to all edges in the y direction to suppress any out-of-plane deformation

.

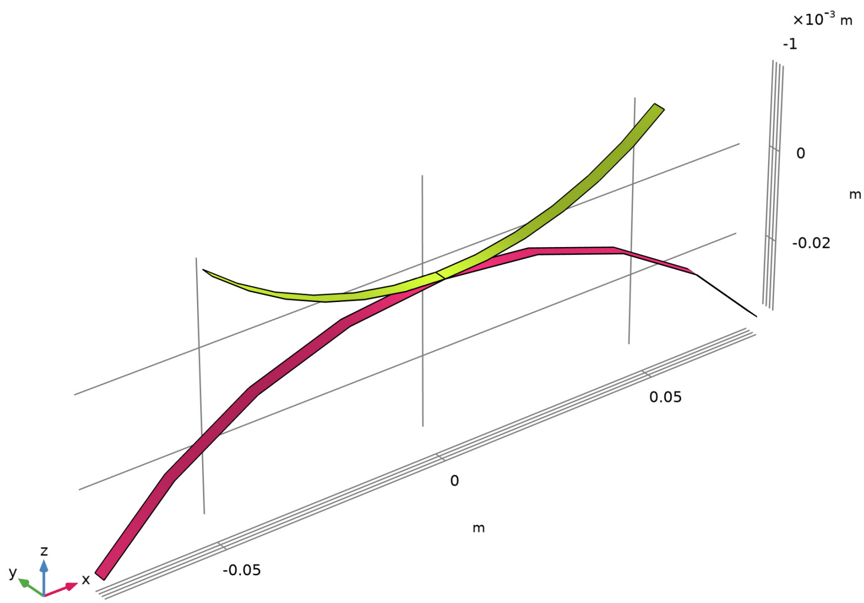


Figure 1: Model geometry.

Only contact without friction is considered and the penalty contact method is used.

The ends of the upper arch are constrained against displacement in the x direction and subjected to vertical edge loads. The magnitude of the edge loads is controlled by the monotonically increasing deflection of the upper arch, which makes it possible to track the entire load path, even though the force does not increase monotonically. The ends of the lower arch are fixed.

Results and Discussion

Figure 2 depicts the deformed shape and the von Mises stress distribution at the last step of the simulation. The snap-through of the lower arch is clearly visible. Both arches are represented by a shell dataset that shows both their top and bottom surfaces.

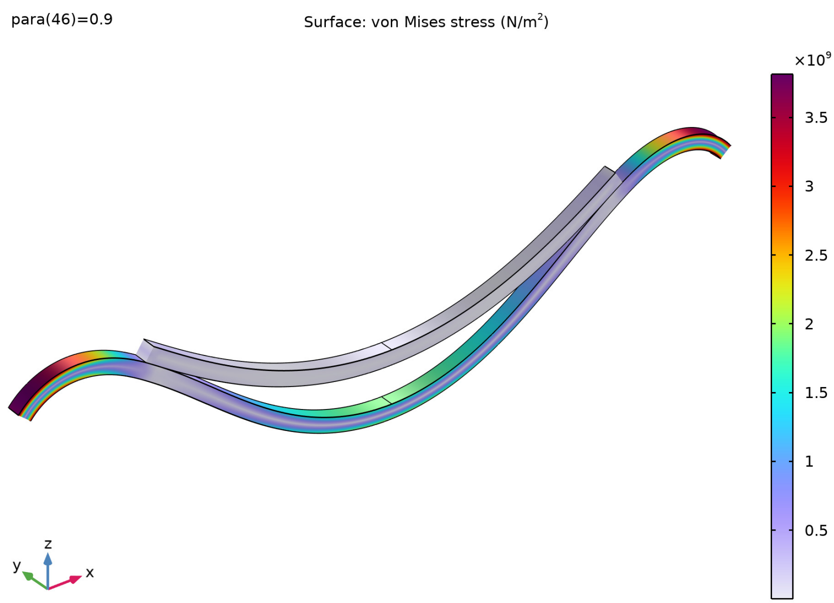


Figure 2: Deformation and von Mises stress at the final step.

Three different load versus deflection curves are shown in Figure 3. The load is represented by a dimensionless load factor, and is plotted against either the mid deflections of the two arches or the average deflection of the ends of the upper arch. Several critical points can be observed. For example, looking at the lower arch, a first limit point is reached for a load factor equal to 107.5 and a deflection of 13 mm. At this point the lower arch becomes unstable and a snap-through occurs. When the deflection reaches 45 mm, the load factor has decreased to 45. At this point a second limit point is reached, and the model finds a new stable configuration. After this point the load factor increases with increasing deflection.

Several bifurcation points are also present, indicating the unstable nature of the problem and possible branching of the load path. A first point is, for example, visible already at a deflection of 1 mm, where there is a clear change in the slope of the load-deflection curve.

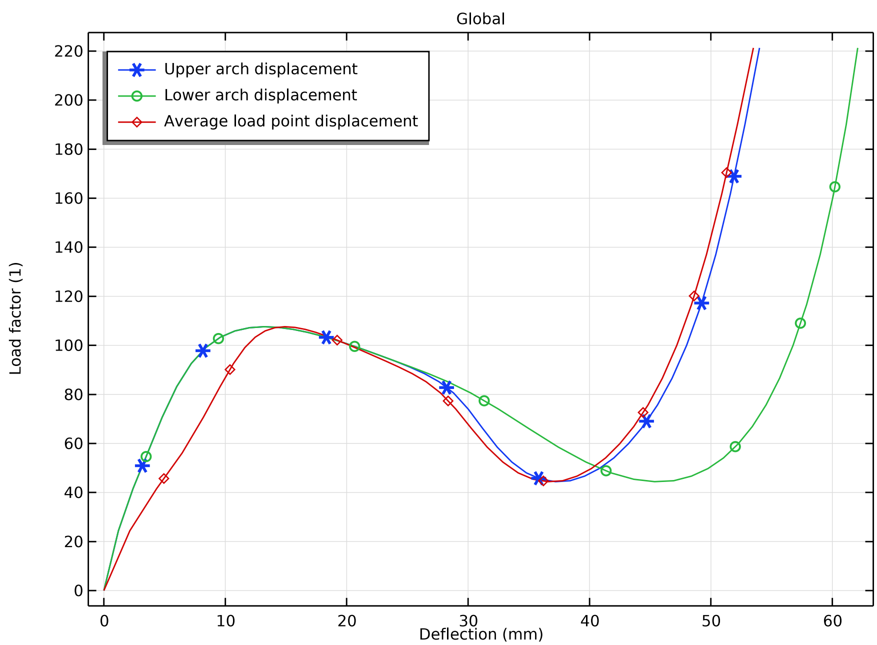


Figure 3: Load versus deflection curves.

The progressive deformation of the two arches, including the snap-through of the lower arch, is shown in Figure 4 for five values of the continuation parameter. In the figure, it also is clearly visible how the contact problem changes throughout the simulation. Figure 5 shows the contact pressure exerted by the upper arch on the lower arch during the postcritical stage.

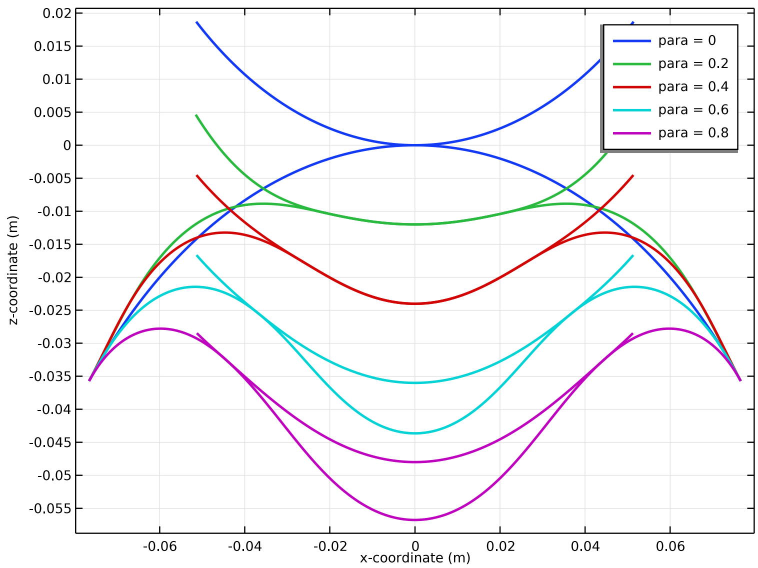


Figure 4: Deformation of the model for five different parameter values.

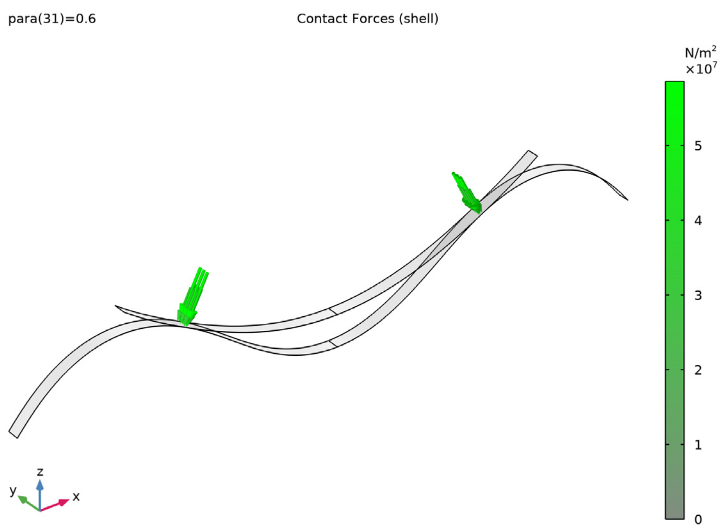


Figure 5: Contact pressure acting on the lower arch.

Notes About the COMSOL Implementation

Modeling the postcritical behavior of a system is not possible by incrementally increasing the boundary load. The unstable behavior is even more pronounced when contact is present. To be able to find all limit points and to track the full load versus deflection curve, a displacement controlled load scheme is used by adding a **Global Equation**. Here, the magnitude of the edge loads is controlled through the monotonically increasing deflection of the upper arch. Alternatively, the vertical displacement could be prescribed on end points of the upper arch, but this is a less general technique that fails for some cases.

This problem is highly unstable and several branches of the equilibrium path are possible. To suppress these so that a stable solution is obtained, the mid-point of both arches is constrained against sideways displacement through a symmetry condition. By deactivating this constraint, it is possible to study the branching of the equilibrium path.

Reference

1. P. Wriggers, *Computational Contact Mechanics*, Springer-Verlag, 2006

Application Library path: `Structural_Mechanics_Module/Verification_Examples/two_arches`

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 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `two_arches_parameters.txt`.

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 **Go to Plane Geometry**.

Work Plane 1 (wp1)>Circle 1 (c1)

1 In the **Work Plane** 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 **Radius** text field, type `Ri_upper`.

5 In the **Sector angle** text field, type `seg_upper`.

6 Locate the **Position** section. In the **yw** text field, type `Ri_upper`.

7 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90-seg_upper/2`.

8 Click **Build Selected**.

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

Work Plane 1 (wp1)>Circle 2 (c2)

1 In the **Work Plane** 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 **Radius** text field, type `Ri_lower`.

5 In the **Sector angle** text field, type `seg_lower`.

6 Locate the **Position** section. In the **yw** text field, type `-Ri_lower`.

7 Locate the **Rotation Angle** section. In the **Rotation** text field, type `90-seg_lower/2`.

8 Click **Build Selected**.

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

Work Plane 1 (wp1)>Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Plane Geometry** and choose **Delete Entities**.

2 On the object **c1**, select Boundaries 2 and 3 only.

3 On the object **c2**, select Boundaries 3 and 4 only.

Work Plane 1 (wp1)>Partition Edges 1 (pare1)

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Partition Edges**.

2 On the object **del1(1)**, select Boundary 1 only.

Work Plane 1 (wp1)

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

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

3 Clear the **Unite objects** check box.

Extrude 1 (ext1)

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

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

3 In the table, enter the following settings:

Distances (m)
d

4 Click **Build Selected**.

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

Upper Arch

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Upper Arch` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Object**.

4 Select the object **ext1(2)** only.

5 Locate the **Color** section. From the **Color** list, choose **Color 4**.

6 Click **Build Selected**.

7 Right-click **Upper Arch** and choose **Duplicate**.

Lower Arch

1. In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Upper Arch 1 (sel2)**.
2. In the **Settings** window for **Explicit Selection**, type `Lower Arch` in the **Label** text field.
3. Locate the **Entities to Select** section. In the list, select **ext1(2)**.
4. Select the object **ext1(1)** only.
5. Locate the **Color** section. From the **Color** list, choose **Color 12**.

Form Union (fin)

1. In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.
2. In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.
3. From the **Action** list, choose **Form an assembly**.
4. Click **Build Selected**.
5. Click the **Zoom Extents** button in the **Graphics** toolbar.

MATERIALS

Material 1 (mat1)

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Lower Arch**.
4. Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	`40[GPa]`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.2`	1	Young's modulus and Poisson's ratio
Density	rho	1	kg/m³	Basic

Material 2 (mat2)

1. Right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Upper Arch**.

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	20[GPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's ratio
Density	rho	1	kg/m³	Basic

DEFINITIONS

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 **Point**.

4 Select Point 9 only.

5 Right-click **Average 1 (aveop1)** and choose **Duplicate**.

Average 2 (aveop2)

1 In the **Model Builder** window, click **Average 2 (aveop2)**.

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

3 Click **Clear Selection**.

4 Select Point 3 only.

5 Right-click **Average 2 (aveop2)** and choose **Duplicate**.

Average 3 (aveop3)

1 In the **Model Builder** window, click **Average 3 (aveop3)**.

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

3 Click **Clear Selection**.

4 Select Points 7 and 11 only.

Variables 1

1 In the **Model Builder** window, 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
disp_upper	aveop1(-w)	m	Upper arch displacement
disp_lower	aveop2(-w)	m	Lower arch displacement
disp_load	aveop3(-w)	m	Average load point displacement

Contact Pair 1 (p1)

1 In the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 In the **Settings** window for **Pair**, locate the **Source Boundaries** section.

3 From the **Selection** list, choose **Upper Arch**.

4 Locate the **Destination Boundaries** section. From the **Selection** list, choose **Lower Arch**.

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 d.

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

Symmetry 1

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

2 Select Edges 2, 3, 5, 6, 9, 10, 12, and 13 only.

Prescribed Displacement/Rotation 1

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

2 Select Edges 8 and 14 only.

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

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

5 From the **Displacement in y direction** list, choose **Prescribed**.

Prescribed Displacement/Rotation 2

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

2 Select Edges 1 and 7 only.

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

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

5 From the **Displacement in y direction** list, choose **Prescribed**.

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

7 Locate the **Prescribed Rotation** section. From the **By** list, choose **Rotation**.

Edge Load 1

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

2 Select Edges 8 and 14 only.

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

4 Specify the $\mathbf{F}_L$ vector as

0	x
0	y
load*F_ref	z

The dependent variable `load` will be created in the next step using a global equation.

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 (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
load	disp_up per-max_disp*para	0	0	Load factor

4 Locate the **Units** section. Click **Select Source Term Quantity**.

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

6 Click **Filter**.

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

8 Click **OK**.

Several possible branches are possible during the snap-through. Adding a constraint to each arch enforces a symmetric and stable solution.

Symmetry 2

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

2 Select Edges 4 and 11 only.

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 2 and 5 only.

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

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

Distribution 2

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

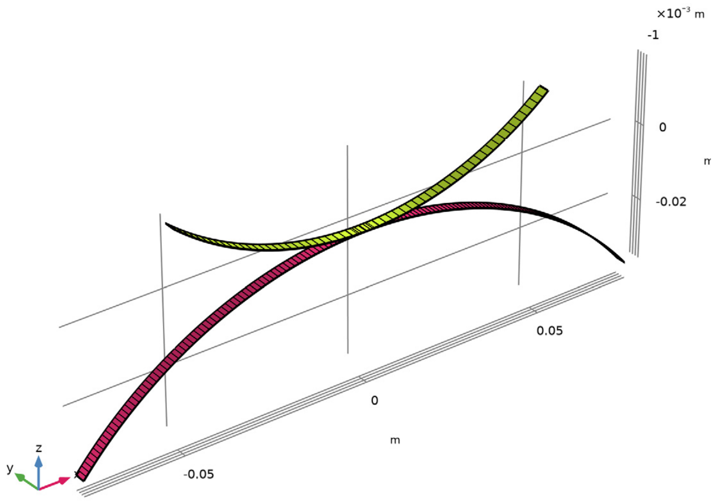
2 Select Edges 9 and 12 only.

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

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

5 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

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



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**, 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	Parameter unit
para (Load parameter)	`range(0,0.02,1)`	

6 In the table, click to select the cell at row number 1 and column number 3.

Solution 1 (sol1)

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

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

3 In the **Settings** window for **Stationary Solver**, locate the **General** section.

4 In the **Relative tolerance** text field, type `0.0005`.

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

6 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.

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

8 In the **Minimum step size** text field, type `1e-6`.

9 Right-click **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1> Parametric 1** and choose **Stop Condition**.

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

11 Click **Add**.

12 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
`comp1.load/250`	True (>=1)	√	`Stop expression 1`

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

14 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Stationary Solver 1** click **Fully Coupled 1**.

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

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

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

RESULTS

Stress (shell)

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

2 Click the **Show Grid** button in the **Graphics** toolbar.

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

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 1/Solution 1 (sol1)>Shell>Contact Forces (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.

RESULTS

Contact Forces (shell)

1 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

2 From the **Parameter value (para)** list, choose **0.6**.

Contact 1, Pressure

1 In the **Model Builder** window, expand the **Contact Forces (shell)** node, then click **Contact 1, Pressure**.

2 In the **Settings** window for **Arrow Surface**, locate the **Coloring and Style** section.

3 Select the **Scale factor** check box. In the associated text field, type `2e-10`.

Gray Surfaces

In the **Model Builder** window, click **Gray Surfaces**.

Animation 1

1 In the **Contact Forces (shell)** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, locate the **Frames** section.

3 From the **Frame selection** list, choose **All**.

4 Click the **Play** button in the **Graphics** toolbar.

Load vs. Deflection

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. Deflection` in the **Label** text field.

Global 1

1 Right-click **Load vs. Deflection** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
disp_upper	mm	Upper arch displacement
disp_lower	mm	Lower arch displacement
disp_load	mm	Average load point displacement

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

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

6 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

7 From the **Positioning** list, choose **Interpolated**.

Load vs. Deflection

1 In the **Model Builder** window, click **Load vs. Deflection**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **Flip the x- and y-axes** check box.

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

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** check box. In the associated text field, type `Deflection (mm)`.

7 In the **Load vs. Deflection** toolbar, click **Plot**.

Deformation

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

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

3 Locate the **Data** section. From the **Parameter selection (para)** list, choose **Manual**.

4 In the **Parameter indices (1-46)** text field, type `range(1,10,41)`.

5 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Line Graph 1

1 Right-click **Deformation** and choose **Line Graph**.

2 Select Edges 2 and 5 only.

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

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

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

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

7 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

8 Right-click **Line Graph 1** and choose **Duplicate**.

Line Graph 2

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

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

3 Click **Clear Selection**.

4 Select Edges 9 and 12 only.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

6 From the **Color** list, choose **Cycle (reset)**.

Line Graph 1

1 In the **Model Builder** window, click **Line Graph 1**.

2 In the **Settings** window for **Line Graph**, click to expand the **Legends** section.

3 Select the **Show legends** check box.

4 Find the **Prefix and suffix** subsection. In the **Prefix** text field, type `para = `.

5 In the **Deformation** toolbar, click **Plot**.

Stress (shell)

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