Created in COMSOL Multiphysics 5.6

Block Pressing on Arch

Introduction

This conceptual example shows how to calculate critical points in models with contact. The model consists of a block modeled with the Solid Mechanics interface pressing on an arch modeled with the Shell interface and also exemplifies how to model the contact between a shell and a solid. During loading, the 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 arch is 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 boundaries and edges in the *y* direction to suppress any out-of-plane deformation.

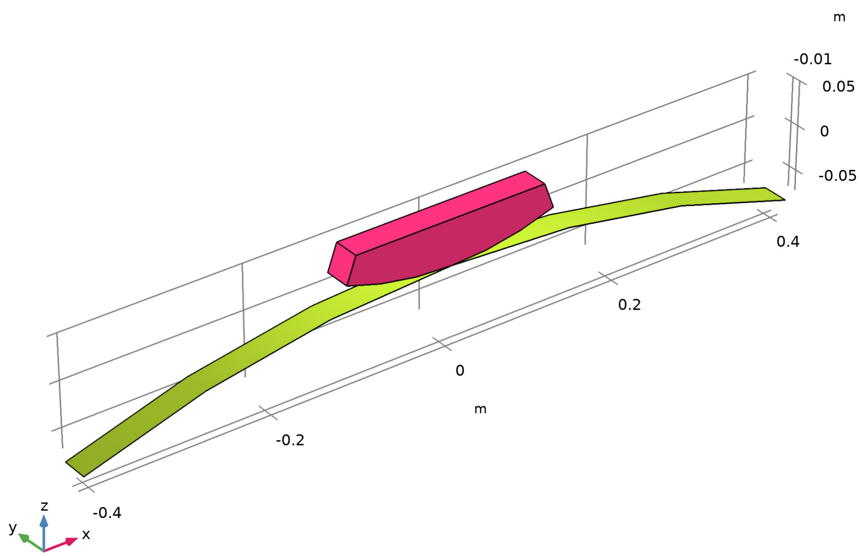


Figure 1: Model geometry

Only contact without friction is considered and the augmented Lagrangian contact method is used.

A boundary load is applied the top surface of the block. Its magnitude is controlled by the monotonically increasing deflection of the arch, which makes it possible to track the entire load path, even though the force does not increase monotonically. The ends of the arch are fixed and the displacement of the block is constrained in the *x* direction.

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 arch is clearly observed. The arch is represented by a shell dataset that shows both its top and bottom surface.

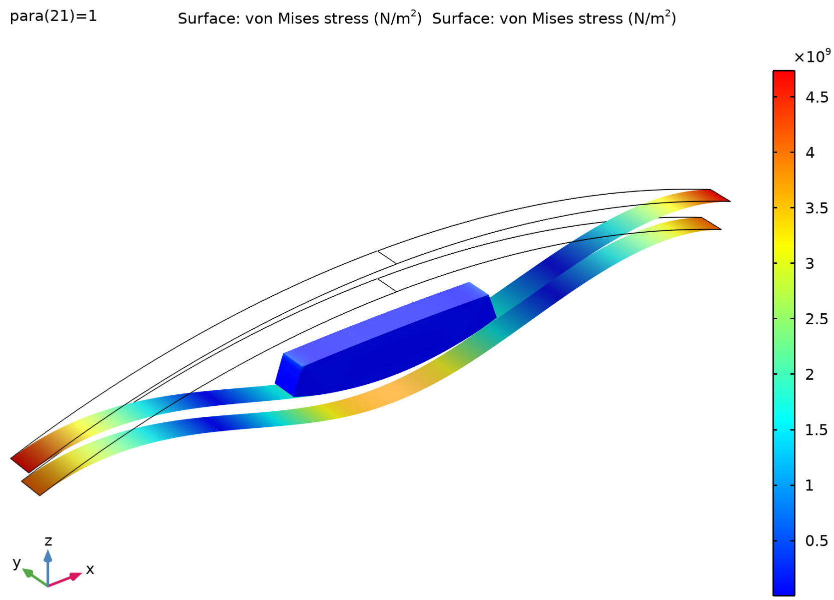


Figure 2: Deformation and von Mises stress at the final step.

The load versus deflection curve is shown in Figure 3. The load is in the figure represented by a dimensionless load factor. Two limit points can be observed, the first occurs for a load factor equal to 18 and a deflection of 36 mm. At this point the arch becomes unstable and a snap-through occurs. When the deflection of the arch reaches 80 mm, the load factor has decreased to 14. At this point the second limit point is reached, and the arch finds a new stable configuration. After this point the load factor increases with increasing deflection.

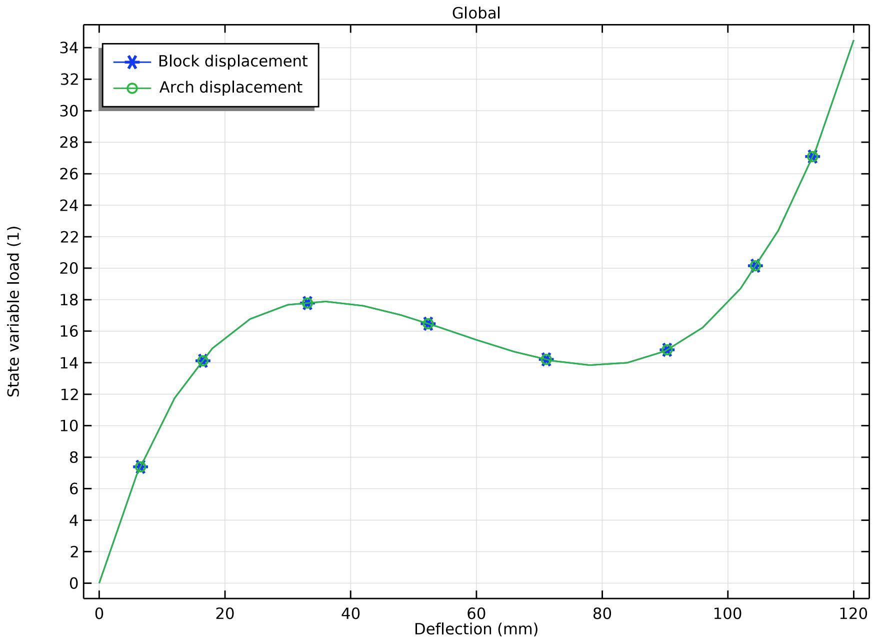


Figure 3: Load versus deflection curve.

The progressive deformation of the block and the arch, including the snap-through of the arch, is shown in Figure 4 for six values of the continuation parameter. Figure 5 shows the contact pressure exerted by the block on the arch during the snap-through.

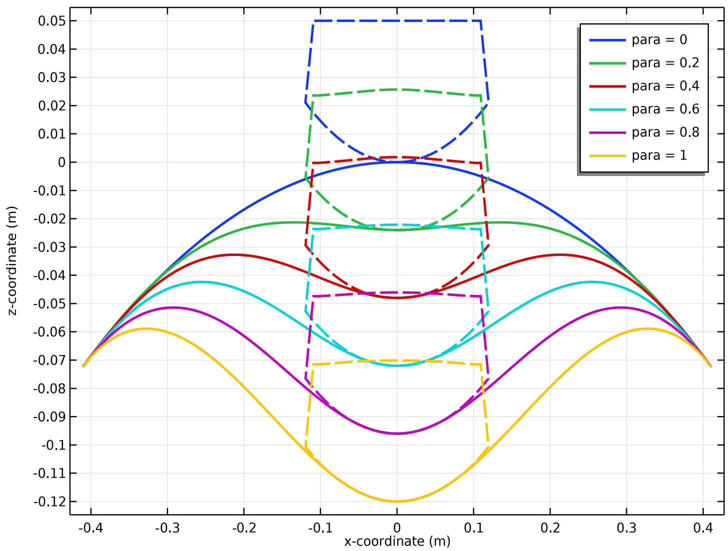


Figure 4: Deformation of the model for six different parameter values.

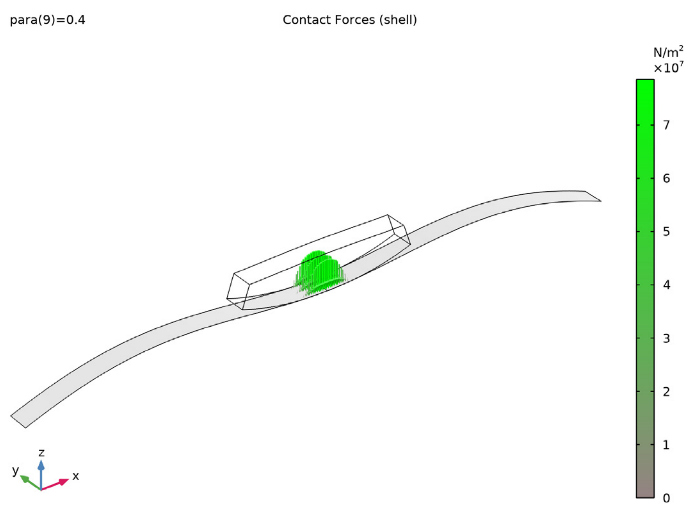


Figure 5: Contact pressure acting on the arch.

Notes About the COMSOL Implementation

When a Shell interface is used in a contact simulation, it is recommended that the destination boundary always belongs to the shell. Moreover, the contact definition should be made in the Shell interface. In this example, the block modeled with a Solid Mechanics interface is thus, in the **Contact** node, considered as external to the current physics.

Contact problems are often unstable in their initial configuration. To help the solver find an initial solution, a **Spring Foundation** is added to the otherwise unconstrained block during the first parameter step.

Modeling the post-critical 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 boundary load is controlled through the monotonically increasing deflection of the arch. Alternatively, the vertical displacement could be prescribed on the top surface of the block, but this is a less general technique that fails for some cases. Also, a prescribed displacement would not give an evenly distributed load.

Reference

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

Application Library path: `Structural_Mechanics_Module/Verification_Examples/block_on_arch`

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 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

5 Click **Add**.

6 In the **Displacement field** text field, type `u`.

7 Click **Study**.

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

9 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 `block_on_arch_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 **Show Work Plane**.

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 `R_arch`.

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

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

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

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**, select Boundary 1 only.

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

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

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

3 In the **Radius** text field, type `R_block`.

4 In the **Sector angle** text field, type `seg_block`.

5 Locate the **Position** section. In the **yw** text field, type `R_block`.

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

7 Click **Build Selected**.

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

Work Plane 1 (wp1)>Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `R_block`.

4 In the **Height** text field, type `height_block`.

5 Locate the **Position** section. In the **xw** text field, type `-R_block/2`.

6 Click **Build Selected**.

Work Plane 1 (wp1)>Intersection 1 (int1)

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

2 Select the objects **c2** and **r1** only.

Work Plane 1 (wp1)

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

Arch

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

2 In the **Settings** window for **Explicit Selection**, type `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(1)** only.

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

6 Click **Build Selected**.

Block

1 Right-click **Arch** and choose **Duplicate**.

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

3 Locate the **Entities to Select** section. In the list, select **ext1(1)**.

4 Select the object **ext1(2)** 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 **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	10[GPa]	Pa	Basic
Poisson's ratio	nu	0.2	1	Basic
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 **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Arch**.

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	70[GPa]	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
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 11 only.

Average 2 (aveop2)

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

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

3 Click **Clear Selection**.

4 Select Point 3 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_block	aveop1(-w)	m	Block displacement
disp_arch	aveop2(-w)	m	Arch displacement

Contact Pair 1 (p1)

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

2 Select Boundaries 4 and 8 only.

3 Click the **Go to Default View** button in the **Graphics** toolbar.

4 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

5 From the **Selection** list, choose **Arch**.

The destination boundary should be on a boundary modeled with the Shell interface.

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 **Arch**.

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 `d`.

4 From the **Offset definition** list, choose **Relative offset**.

5 In the $z_{\mathrm{reloffset}}$ text field, type `-1`.

Prescribed Displacement/Rotation 1

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 Select the **Prescribed in x direction** check box.

5 Select the **Prescribed in z direction** check box.

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

Symmetry 1

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

2 Select Edges 2, 3, 5, and 6 only.

Contact 1

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

2 In the **Settings** window for **Contact**, locate the **Pair Selection** section.

3 Under **Pairs**, click **Add**.

4 In the **Add** dialog box, select **Contact Pair 1 (p1)** in the **Pairs** list.

5 Click **OK**.

6 In the **Settings** window for **Contact**, locate the **Contact Method** section.

7 From the **Formulation** list, choose **Augmented Lagrangian**.

8 Locate the **Contact Surface** section. Select the **Source external to current physics** check box.

The source boundary is in the Solid Mechanics interface.

SOLID MECHANICS (SOLID)

In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

Prescribed Displacement 1

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

2 Select Edges 13 and 19 only.

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

4 Select the **Prescribed in x direction** check box.

Symmetry 1

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

2 Select Boundaries 5 and 6 only.

Boundary Load 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundary 7 only.

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

4 Specify the $\mathbf{F}_A$ 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

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_block-para*max_disp	0	0	

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**.

Add a small spring stiffness to the block to stabilize the model during the initial step.

Spring Foundation 1

1 In the **Physics** toolbar, click **Domains** and choose **Spring Foundation**.

2 In the **Settings** window for **Spring Foundation**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Block**.

4 Locate the **Spring** section. In the $\mathbf{k}_V$ text field, type `1e3*(para<0.01)`.

MESH 1

Mapped 1

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

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

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

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_arch`.

Mapped 2

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

2 Select Boundary 5 only.

Distribution 1

1 Right-click **Mapped 2** and choose **Distribution**.

2 Select Edges 10 and 17 only.

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

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

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 2** and choose **Distribution**.

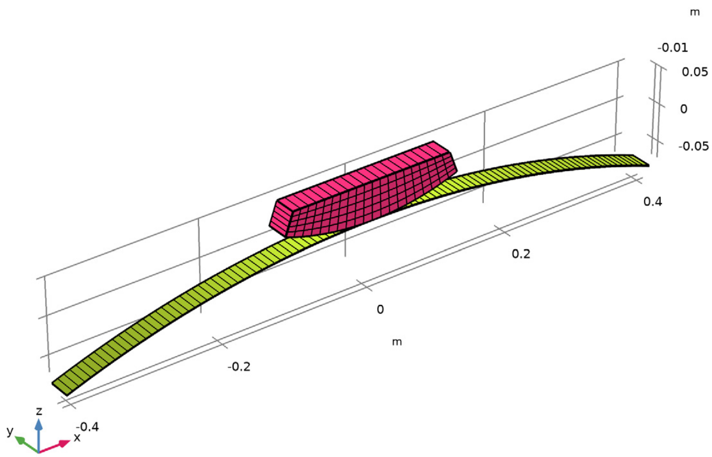
2 Select Edges 9 and 20 only.

Swept 1

1 In the **Mesh** toolbar, click **Swept**.

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

3 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(0,0.05,1)`	

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)>Dependent Variables 1** node, then click **State variable load (comp1.ODE1)**.

6 In the **Settings** window for **State**, locate the **Scaling** section.

7 From the **Method** list, choose **Manual**.

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

9 In the **Settings** window for **Segregated Step**, locate the **General** section.

10 Under **Variables**, click **Add**.

11 In the **Add** dialog box, select **State variable load (comp1.ODE1)** in the **Variables** list.

12 Click **OK**.

13 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1>Segregated 1** right-click **Solid Mechanics** and choose **Delete**.

Structural mechanics interfaces should be solved in a single segregated step.

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

RESULTS

Surface 2

1 In the **Model Builder** window, expand the **Stress (shell)** node.

2 Right-click **Results>Stress (shell)>Surface 1** and choose **Duplicate**.

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

4 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

5 From the **Solution parameters** list, choose **From parent**.

6 Locate the **Expression** section. In the **Expression** text field, type `solid.mises`.

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

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

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

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

Contact Forces (shell)

1 In the **Model Builder** window, click **Contact Forces (shell)**.

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

3 From the **Parameter value (para)** list, choose **0.4**.

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.
4 In the associated text field, type `5e-10`.

Selection 1

1 In the **Model Builder** window, expand the **Results>Contact Forces (shell)>Gray Surfaces** node, then click **Selection 1**.
2 Select Boundary 1 only.
3 In the **Settings** window for **Selection**, locate the **Selection** section.
4 From the **Selection** list, choose **Arch**.
5 In the **Contact Forces (shell)** toolbar, click **Plot**.

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 Right-click **Animation 1** and choose **Play**.

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_block	mm	Block displacement
disp_arch	mm	Arch 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**.

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. Select the **x-axis label** check box.
6. 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 (1-21)** text field, type `range(1,4,21)`.
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. In the **Width** text field, type `2`.

Line Graph 2

1. Right-click **Line Graph 1** and choose **Duplicate**.
2. In the **Settings** window for **Line Graph**, locate the **Selection** section.
3. Select the **Activate Selection** toggle button.
4. Select Edges 9, 10, 14, 17, and 20 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 **Include** 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.