Model created in COMSOL Multiphysics 6.4

Connecting Layered Shells with Solids and Shells

Introduction

The **Layered Shell** interface is used to model thick to moderately thin composite laminates. These composite laminates are often connected with solid or sufficiently thin structures in different configurations to represent a realistic structure. These solid and thin structures are in general accurately and efficiently modeled using **Solid Mechanics** and **Shell** interfaces respectively.

This tutorial and verification model illustrates how to connect layered shell elements with solid and shell elements in cladding or side-by-side configuration using built-in coupling features. In this example, the results of the layered shell-solid-shell structure is compared with the reference model built using solid elements.

Read more about the Composite Materials Module in the COMSOL blog, Introduction to the Composite Materials Module.

Model Definition

The model (Figure 1) consists of two set of geometries; model geometry and reference geometry. The model geometry consists of layered shell, solid, and shell elements whereas the reference geometry is modeled with only solid elements.

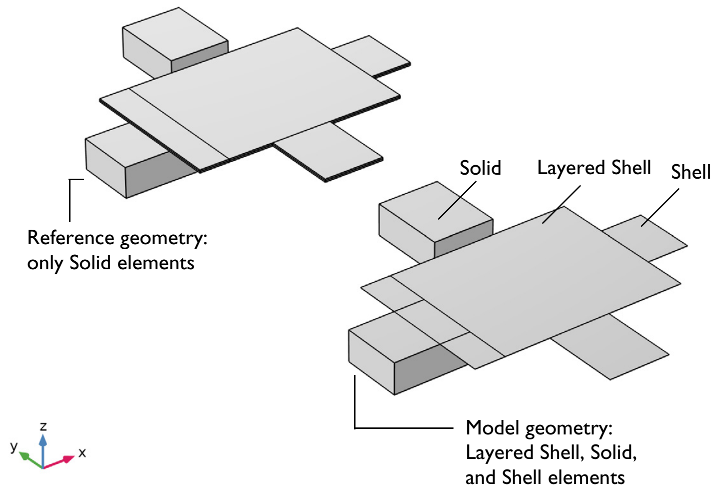

Figure 1: The model and reference geometry showing different structural members in different parts of the geometry.

LAYERED SHELL-STRUCTURE COUPLINGS

The model geometry has three sections. The middle section is modeled as a layered shell elements, while side supports are modeled using solid and shell elements as shown in Figure 1. The model geometry is made up of solid blocks and surfaces whereas the reference geometry is made up of only solid blocks. The connection between the layered shell and other structural elements are defined as below:

- Boundaries of the **Solid Mechanics** interface shared with the **Layered Shell** interface, the connection is set up using **Layered Shell-Structure Cladding** multiphysics coupling.
- Boundaries of the **Solid Mechanics** interface side-by-side with the **Layered Shell** interface, the connection is set up using the **Layered Shell-Structure Transition** multiphysics coupling.
- Boundaries of the **Shell** interface parallel with the **Layered Shell** interface, the connection is set up using **Layered Shell-Structure Cladding** multiphysics coupling.
- Edges of the **Shell** interface side-by-side with the **Layered Shell** interface, the connection is set up using the **Layered Shell-Structure Transition** multiphysics coupling.

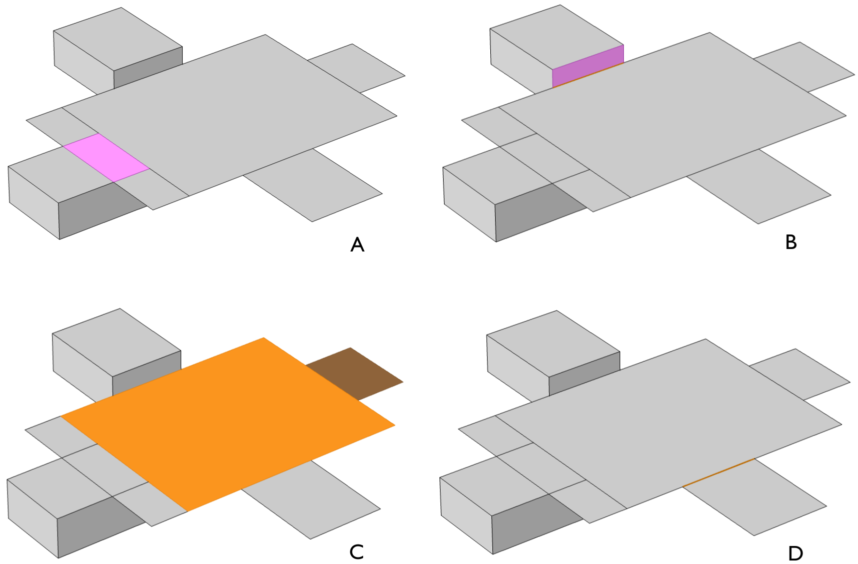

Figure 2: Different connections of layered shell with other structural members on different sides are as follows:(A) layered shell-solid cladding (B) layered shell-solid transition (C) layered shell-shell cladding (D) layered shell-shell transition.

AUXILIARY SLIT

In the layered shell–solid transition coupling, only the bottom layer of layered shell is connected to the solid, whereas the top layer free. A similar connection is achieved in the reference model by disconnecting the degrees of freedom using an **Auxiliary Slit** node in the **Solid Mechanics** interface.

STACKING SEQUENCE

In the model geometry, layered shell and shell members consist of two layers where each layer (ply) has a thickness of 10 mm with [0/45] stacking sequence.

MATERIAL PROPERTIES

In the model geometry, layered shell and shell elements are made up of AS4/APC carbon–thermoplastic composite material. The AS4/APC carbon–thermoplastic is a built-in material in the **Composites** material library. The transversely isotropic material properties (Young's modulus, shear modulus, and Poisson's ratio) are given in Table 1.

TABLE 1: MATERIAL PROPERTIES OF A LAMINA.

Material property	Value
ρ	1570 kg/m^3
$\{E_{11}, E_{22}\}$	{138, 8.7} GPa
G_{12}	5 GPa
$\{\nu_{12}, \nu_{23}\}$	{0.28, 0.45}

The solid elements in the model geometry are made up of structural steel. The stacking sequence and materials in the reference geometry are used as per the model geometry specifications.

BOUNDARY CONDITIONS

- Boundary load of 10 kN is applied at the top surface of the middle plate modeled using layered shell.
- Fixed constraints is used on the outermost boundaries of four support members modeled using solid and shell members.

Results and Discussion

von Mises stress distribution for the given applied load is shown in Figure 3. The stress distribution in the model geometry matches quite well with the same in the reference geometry. The total displacement in both setups are shown in Figure 4 which also matches quite closely with each other. This shows the accuracy of different types of elements and connections used in the model geometry.

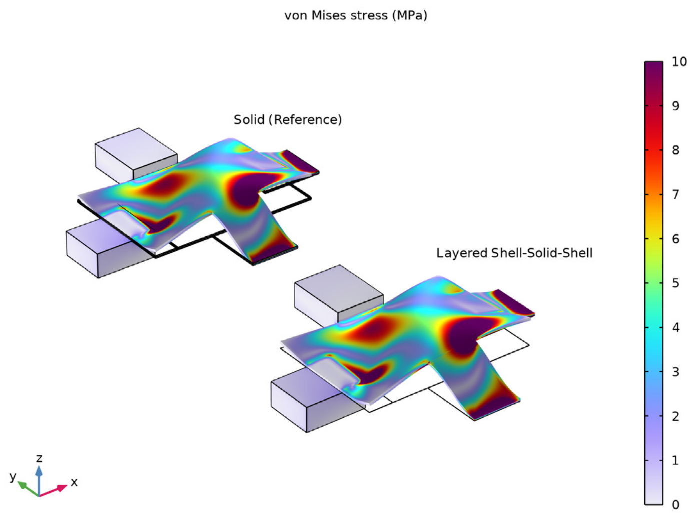

Figure 3: The comparison of von Mises stress distribution in both structural models.

The distribution of von Mises stress in the bottom and top layer of composite laminate modeled using layered shell and solid elements are shown in Figure 5 and Figure 6, respectively.

The distribution of von Mises stress at the common edge between layered shell and different structural elements is also compared with the reference model. Figure 7 through Figure 10 illustrate such a comparison with a good overall qualitative and quantitative match.

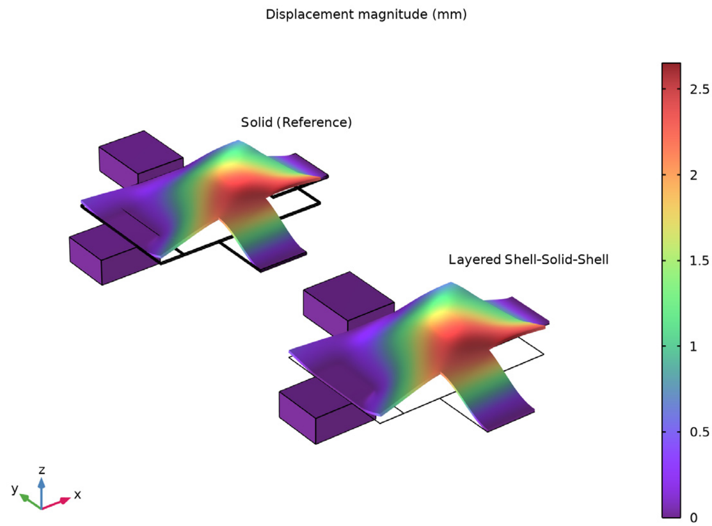

Figure 4: The comparison of total displacement distribution in both structural models.

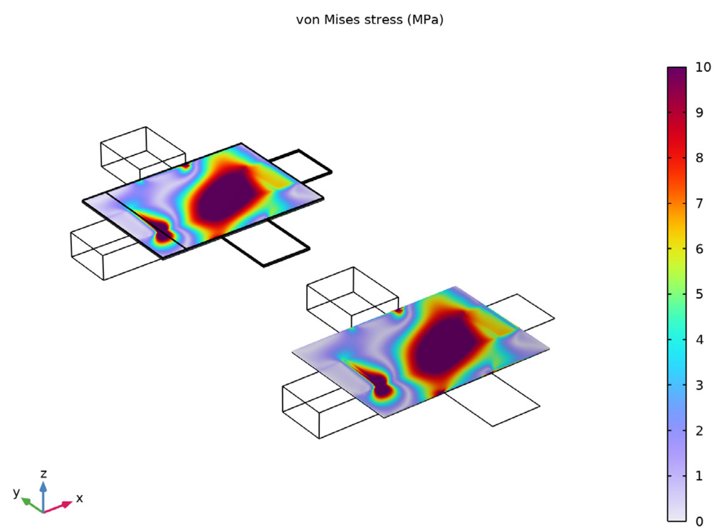

Figure 5: The comparison of von Mises stress distribution in the bottom layer of both structural models.

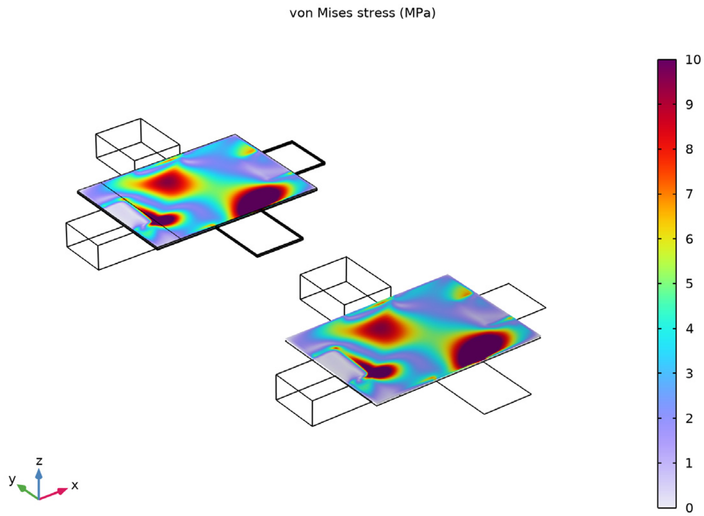

Figure 6: The comparison of von Mises stress distribution in the top layer of both structural models.

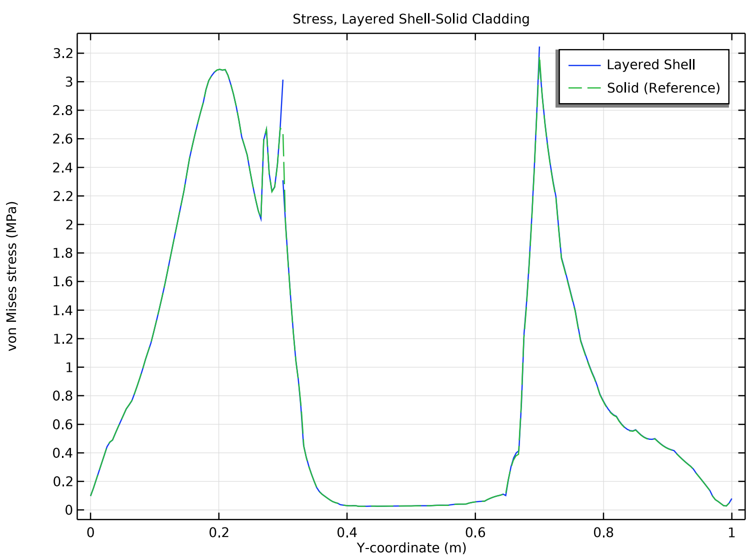

Figure 7: von Mises stress along the common edge for layered shell-solid cladding coupling.

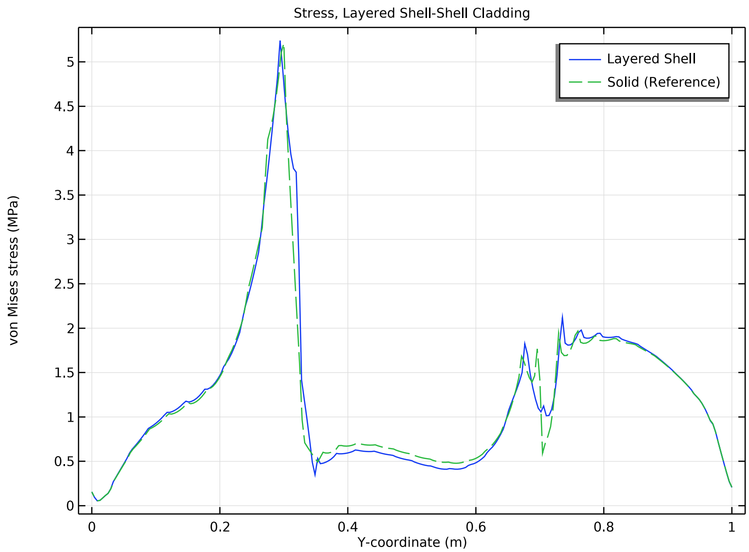

Figure 8: von Mises stress along the common edge for layered shell-shell cladding coupling.

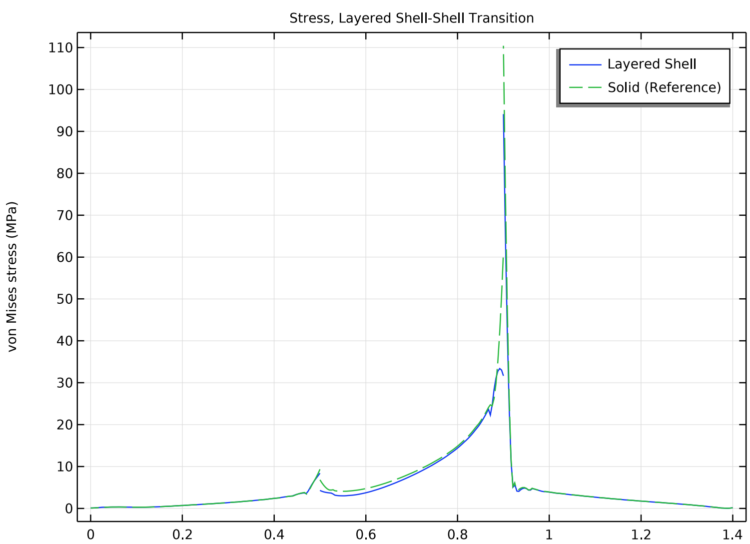

Figure 9: von Mises stress along the common edge for layered shell-shell transition coupling.

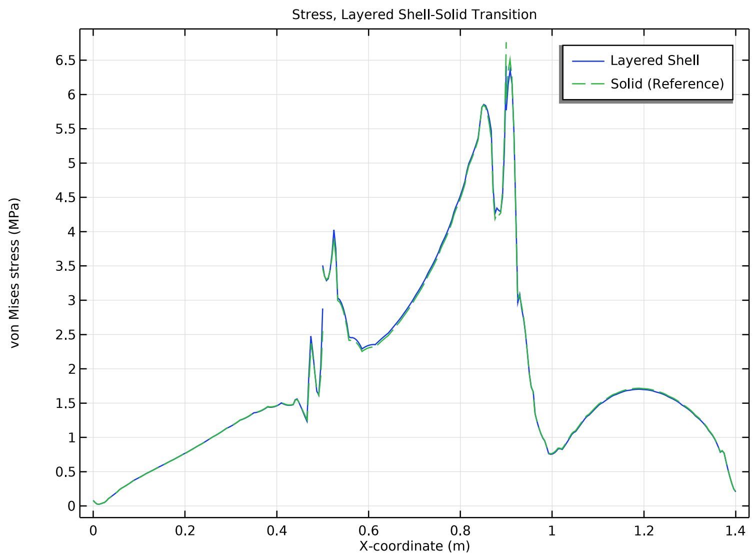

Figure 10: von Mises stress along the common edge for layered shell-solid transition coupling.

Notes About the COMSOL Implementation

- Modeling a composite laminate as a layered shell requires a surface geometry, in general referred to as a base surface, and a **Layered Material** node which adds an extra dimension (1D) to the base surface geometry in the surface normal direction. You can use the **Layered Material** functionality to model several layers stacked on top of each other having different thicknesses, material properties, and fiber orientations. You can optionally specify the interface materials between the layers, and control the number of through-thickness mesh elements for each layer.
- The third direction for the selected coordinate system in the **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack** represents the normal direction of the **Layered Shell** or **Shell** physics. This is also the direction in which the layer stacking is interpreted from bottom to top, and therefore, it is crucial to know it during modeling. There are two ways to achieve this:
  - Using physics symbols: Go to the physics settings, find the **Physics Symbols** section, and select the **Enable physics symbols** checkbox. Then go to the material feature, for

instance, **Linear Elastic Material**, to see the normal direction represented by green arrows in the geometry.

- Using result templates: When a solution dataset is available, use the result template **Thickness and Orientation** to plot the normal direction.

• From a constitutive model point of view, you can either use the *Layerwise (LW)* theory based **Layered Shell** interface or the *Equivalent Single Layer (ESL)* theory based **Linear Elastic Material, Layered** node in the **Shell** interface.

• The **Layered Shell - Structure Cladding** multiphysics coupling is used to model cladding between a **Layered Shell** interface and a **Solid Mechanics**, **Shell**, or **Membrane** interface. In the **Connection Settings** section, shared and parallel boundaries options are provided to connect boundaries of different structural physics interfaces.¨

• The **Layered Shell - Structure Transition** multiphysics coupling is used to couple side-by-side structural connection between a **Layered Shell** interface and a **Solid Mechanics** or **Shell** interface. This is a layered multiphysics coupling and in the **Shell Properties** section, it is possible to select only few layers for the connection.

• The built-in **Composites** material library contains data for fiber and matrix constituents as well as for unidirectional and bidirectional laminae.

Application Library path: `Composite_Materials_Module/Tutorials/layered_shell_structure_connection`

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** > **Solid Mechanics (solid)**, **Structural Mechanics** > **Shell (shell)**, and **Structural Mechanics** > **Layered Shell (lshell)**.

3 Right-click and choose **Add Physics**.

4 Click **Study**.

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

6 Click **Done**.

LAYERED SHELL (LSHELL)

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Layered Shell (lshell)** and choose **Move Up**.

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
F	10[kN]	10000 N	Total load

GEOMETRY 1

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `layered_shell_structure_connection_geom_sequence.mph`.

3 In the **Geometry** toolbar, click **Build All**.

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

Complete geometry instructions can be found in the Appendix — Geometry Modeling Instructions section.

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 **Geometric Entity Selection** section.

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `4,6` in the **Selection** text field.

6 Click **OK**.

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

8 In the table, enter the following settings:

Name	Expression	Unit	Description
misesTop_solid	solid.mises	N/m²	von Mises stress

Variables 2

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `3,5` in the **Selection** text field.

6 Click **OK**.

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

8 In the table, enter the following settings:

Name	Expression	Unit	Description
misesBot_solid	solid.mises	N/m²	von Mises stress

Variables 3

1 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
misesTop_lshell	lshell.atxd1(lshell.d, mean(lshell.mises))		von Mises stress

Add required materials and layered material first.

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Built-in** > **Structural steel**.

4 Right-click and choose **Add to Global Materials**.

5 In the tree, select **Composites** > **Laminae** > **Unidirectional fiber lamina: AS4/APC2 carbon/PEEK thermoplastic [fiber volume fraction 58%]**.

6 Right-click and choose **Add to Global Materials**.

7 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

GLOBAL DEFINITIONS

Layered Material 1 (lmat1)

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

2 In the **Settings** window for **Layered Material**, locate the **Layer Definition** section.

3 In the table, enter the following settings:

Layer	Material	Rotation	Value	Thickness	Mesh elements
Layer 1	Unidirectional fiber lamina: AS4/APC2 carbon/PEEK thermoplastic [fiber volume fraction 58%] (mat2)	0.0	0 rad	1e-2[m]	1

4 Click **Add**.

5 In the table, enter the following settings:

Layer	Material	Rotation	Value	Thickness	Mesh elements
Layer 2	Unidirectional fiber lamina: AS4/APC2 carbon/PEEK thermoplastic [fiber volume fraction 58%] (mat2)	45	0.7854 rad	1e-2[m]	1

MATERIALS

Layered Material Link 1 (llmat1)

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

2 In the **Settings** window for **Layered Material Link**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `11-13, 25, 37, 69` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Layered Material Link**, locate the **Orientation and Position** section.

8 From the **Position** list, choose **Bottom side on boundary**.

Material Link 1 (matlnk1)

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 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `3-6, 8, 9, 11, 12` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Material Link**, locate the **Link Settings** section.

7 From the **Material** list, choose **Unidirectional fiber lamina: AS4/APC2 carbon/ PEEK thermoplastic [fiber volume fraction 58%] (mat2)**.

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 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `1, 2, 7, 10` in the **Selection** text field.

5 Click **OK**.

Set linear elastic material in all physics interfaces to orthotropic. The isotropic properties of **Structural Steel** is automatically converted to orthotropic properties.

Set the discretization of **Solid Mechanics** interface to quadratic Lagrange in order to have a proper structural connection with other interfaces having quadratic Lagrange discretization.

SOLID MECHANICS (SOLID)

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

2 In the **Settings** window for **Solid Mechanics**, click to expand the **Discretization** section.

3 From the **Displacement field** list, choose **Quadratic Lagrange**.

Linear Elastic Material 1

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

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

3 From the **Material symmetry** list, choose **Orthotropic**.

Linear Elastic Material 2

1 In the **Physics** toolbar, click **Domains** and choose **Linear Elastic Material**.

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

3 From the **Material symmetry** list, choose **Orthotropic**.

4 Locate the **Domain Selection** section. Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `4, 6, 9, 12` in the **Selection** text field.

6 Click **OK**.

DEFINITIONS (COMP1)

Rotated System 2 (sys2)

1 In the **Definitions** toolbar, click **Coordinate Systems** and choose **Rotated System**.

2 In the **Settings** window for **Rotated System**, locate the **Rotation** section.

3 Find the **Euler angles** subsection. In the α text field, type `pi/4`.

SOLID MECHANICS (SOLID)

Linear Elastic Material 2

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** click **Linear Elastic Material 2**.

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

3 From the **Coordinate system** list, choose **Rotated System 2 (sys2)**.

The solid domains form a geometric union. To disconnect two adjacent domains, add an **Auxiliary Slit** feature. To see the feature you first need to enable the **Equation-Based Contributions**.

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

5 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics** > **Equation Contributions**.

6 Click **OK**.

Auxiliary Slit 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `57` in the **Selection** text field.

5 Click **OK**.

In any node in the Model Builder, you can add comments to explain the settings. Right click on the node to select the **Properties and Comments** option to add the comment.

Boundary Load 1

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

2 Select Boundaries 20 and 33 only.

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

4 From the **Load type** list, choose **Total force**.

5 Specify the $\mathbf{F}_{tot}$ vector as

0	x
0	y
F	z

Fixed Constraint 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `1, 6, 43, 45, 48, 60, 82, 83` in the **Selection** text field.

5 Click **OK**.

LAYERED SHELL (LSHELL)

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

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

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type 11-13, 25 in the **Selection** text field.

6 Click **OK**.

Face Load 1

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

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

3 From the **Selection** list, choose **All boundaries**.

4 Locate the **Interface Selection** section. From the **Apply to** list, choose **Top interface**.

5 Locate the **Force** section. From the **Load type** list, choose **Total force**.

6 Specify the $\mathbf{F}_{tot}$ vector as

0	x
0	y
F	z

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 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type 37, 69 in the **Selection** text field.

6 Click **OK**.

Add **Linear Elastic Material, Layered** to shell interface and set it to orthotropic.

Linear Elastic Material, Layered 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Linear Elastic Material, Layered**.

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

3 From the **Material symmetry** list, choose **Orthotropic**.

4 Locate the **Boundary Selection** section. Click **Paste Selection**.

5 In the **Paste Selection** dialog, type 37, 69 in the **Selection** text field.

6 Click **OK**.

Fixed Constraint 1

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

2 Select Edges 68 and 163 only.

MULTIPHYSICS

Add different layered shell-structure multiphysics couplings for appropriate selections.

Layered Shell–Structure Cladding 1 (lssc1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Global** > **Layered Shell–Structure Cladding**.

2 In the **Settings** window for **Layered Shell–Structure Cladding**, locate the **Connection Settings** section.

3 From the **Layered shell boundary** list, choose **Bottom**.

Layered Shell–Structure Transition 1 (lsst1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Edge** > **Layered Shell–Structure Transition**.

2 For this coupling only first layer is connected, so deselect the second layer. To get proper solid boundary selection activate the manual control of solid selections.

3 In the **Settings** window for **Layered Shell–Structure Transition**, locate the **Shell Properties** section.

4 Clear the **Use all layers** checkbox.

5 In the **Selection** table, clear the checkbox for **Layer 2**.

6 Locate the **Edge Selection** section. Click **Clear Selection**.

7 Click **Paste Selection**.

8 In the **Paste Selection** dialog, type 74 in the **Selection** text field.

9 Click **OK**.

10 In the **Settings** window for **Layered Shell–Structure Transition**, locate the **Connection Settings** section.

11 Select the **Manual control of selections** checkbox.

12 Locate the **Boundary Selection, Solid** section. Click **Clear Selection**.

13 Click **Paste Selection**.

14 In the **Paste Selection** dialog, type 39, 41 in the **Selection** text field.

15 Click **OK**.

Layered Shell–Structure Cladding 2 (lssc2)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Global** > **Layered Shell–Structure Cladding**.

2 In the **Settings** window for **Layered Shell–Structure Cladding**, locate the **Coupled Interfaces** section.

3 From the **Structure** list, choose **Shell (shell)**.

4 Locate the **Connection Settings** section. From the **Connection type** list, choose **Parallel boundaries**.

5 Locate the **Boundary Selection, Layered Shell** section. Click to select the **Activate Selection** toggle button.

6 Select Boundary 25 only.

7 Locate the **Boundary Selection, Structure** section. Click to select the **Activate Selection** toggle button.

8 Select Boundary 69 only.

9 Locate the **Connection Settings** section. From the **Layered shell boundary** list, choose **Bottom**.

10 From the **Shell boundary** list, choose **Top**.

Layered Shell–Structure Transition 2 (lsst2)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Edge** > **Layered Shell–Structure Transition**.

2 In the **Settings** window for **Layered Shell–Structure Transition**, locate the **Coupled Interfaces** section.

3 From the **Structure** list, choose **Shell (shell)**.

4 Locate the **Edge Selection** section. Click **Clear Selection**.

5 Select Edge 69 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 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `11, 13, 25, 37, 69` in the **Selection** text field.

5 Click **OK**.

Swept 1

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

Size

1. In the **Model Builder** window, click **Size**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. Click the **Custom** button.
4. Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `0.03`.
5. In the **Minimum element size** text field, type `9.0E-4`.
6. In the **Maximum element growth rate** text field, type `1.3`.
7. In the **Curvature factor** text field, type `0.2`.
8. In the **Resolution of narrow regions** text field, type `1`.
9. Click **Build All**.

STUDY 1

Switch off the generation of default plots, since for this study new custom plots are needed.

1. In the **Model Builder** window, click **Study 1**.
2. In the **Settings** window for **Study**, locate the **Study Settings** section.
3. Clear the **Generate default plots** checkbox.
4. In the **Study** toolbar, click **Compute**.

Set default units for result presentation.

RESULTS

Preferred Units 1

1. In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.
2. In the **Settings** window for **Preferred Units**, locate the **Units** section.
3. Click **Add Physical Quantity**.
4. In the **Physical Quantity** dialog, select **General** > **Displacement (m)** in the tree.
5. Click **OK**.
6. In the **Settings** window for **Preferred Units**, locate the **Units** section.

7 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Displacement	m	mm

8 Click **Add Physical Quantity**.

9 In the **Physical Quantity** dialog, select **Solid Mechanics** > **Stress tensor (N/m^2)** in the tree.

10 Click **OK**.

11 In the **Settings** window for **Preferred Units**, locate the **Units** section.

12 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Stress tensor	N/m^2	MPa

13 Select the **Apply conversions to expressions with the same dimensions** checkbox.

14 Click **Apply**.

As generation of default plots are switched off, create custom **Layered Material** datasets and plots.

Layered Material 1

In the **Results** toolbar, click **More Datasets** and choose **Layered Material**.

Layered Material: Bottom Layer

1 In the **Results** toolbar, click **More Datasets** and choose **Layered Material**.

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

3 Find the **Layer selection** subsection. Clear the **Use all layers** checkbox.

4 In the table, clear the checkbox for **Layer 2**.

5 In the **Label** text field, type `Layered Material: Bottom Layer`.

Layered Material: Top Layer

1 In the **Results** toolbar, click **More Datasets** and choose **Layered Material**.

2 In the **Settings** window for **Layered Material**, type `Layered Material: Top Layer` in the **Label** text field.

3 Locate the **Layers** section. Find the **Layer selection** subsection. Clear the **Use all layers** checkbox.

4 In the table, clear the checkbox for **Layer 1**.

5 In the **Model Builder** window, collapse the **Results** > **Datasets** node.

Stress

1. In the **Results** toolbar, click **3D Plot Group**.
2. In the **Settings** window for **3D Plot Group**, type `Stress` in the **Label** text field.

Surface 1

1. Right-click **Stress** and choose **Surface**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. In the **Expression** text field, type `solid.mises`.
4. Locate the **Coloring and Style** section. From the **Color table** list, choose **Prism**.
5. Click to expand the **Range** section. Select the **Manual color range** checkbox.
6. In the **Maximum** text field, type `10`.

Deformation 1

Right-click **Surface 1** and choose **Deformation**.

Surface 2

1. In the **Model Builder** window, right-click **Stress** and choose **Surface**.
2. In the **Settings** window for **Surface**, locate the **Data** section.
3. From the **Dataset** list, choose **Layered Material 1**.
4. Locate the **Expression** section. In the **Expression** text field, type `lshell.mises`.
5. Click to expand the **Title** section. From the **Title type** list, choose **None**.
6. Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Deformation 1

1. Right-click **Surface 2** and choose **Deformation**.
2. In the **Settings** window for **Deformation**, locate the **Expression** section.
3. In the **x-component** text field, type `u3`.
4. In the **y-component** text field, type `v3`.
5. In the **z-component** text field, type `w3`.

Surface 3

1. In the **Model Builder** window, right-click **Stress** and choose **Surface**.
2. In the **Settings** window for **Surface**, locate the **Data** section.
3. From the **Dataset** list, choose **Layered Material 1**.
4. Locate the **Expression** section. In the **Expression** text field, type `shell.mises`.
5. Locate the **Title** section. From the **Title type** list, choose **None**.

6 Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Deformation 1

1 Right-click **Surface 3** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Expression** section.

3 In the **x-component** text field, type `u2`.

4 In the **y-component** text field, type `v2`.

5 In the **z-component** text field, type `w2`.

Stress

In the **Model Builder** window, under **Results** click **Stress**.

Table Annotation 1

1 In the **Stress** toolbar, click **More Plots** and choose **Table Annotation**.

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

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	z-coordinate	Annotation
1.5	1.5	0	Layered Shell-Solid-Shell
1.5	4	0	Solid (Reference)

5 Locate the **Coloring and Style** section. Clear the **Show point** checkbox.

Stress

1 In the **Model Builder** window, collapse the **Results** > **Stress** node.

2 In the **Model Builder** window, click **Stress**.

3 In the **Stress** toolbar, click **Plot**.

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

Displacement

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

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

Surface 1

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

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

3 In the **Expression** text field, type `solid.disp`.

4 Locate the **Range** section. Clear the **Manual color range** checkbox.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **SpectrumLight**.

Surface 2

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

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

3 In the **Expression** text field, type `lshell.disp`.

Surface 3

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

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

3 In the **Expression** text field, type `shell.disp`.

Displacement

1 In the **Model Builder** window, collapse the **Results** > **Displacement** node.

2 In the **Model Builder** window, click **Displacement**.

3 In the **Displacement** toolbar, click **Plot**.

Stress: Layered Shell, Bottom Layer

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Stress: Layered Shell, Bottom Layer` in the **Label** text field.

Surface 1

1 Right-click **Stress: Layered Shell, Bottom Layer** and choose **Surface**.

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

3 In the **Expression** text field, type `misesBot_solid`.

4 Locate the **Range** section. Select the **Manual color range** checkbox.

5 In the **Maximum** text field, type `10`.

6 Locate the **Coloring and Style** section. From the **Color table** list, choose **Prism**.

Surface 2

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

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

3 From the **Dataset** list, choose **Layered Material: Bottom Layer**.

4 Locate the **Expression** section. In the **Expression** text field, type `lshell.mises`.

5 Locate the **Title** section. From the **Title type** list, choose **None**.

6 Locate the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Stress: Layered Shell, Bottom Layer

1 In the **Model Builder** window, collapse the **Results** > **Stress: Layered Shell, Bottom Layer** node.

2 In the **Model Builder** window, click **Stress: Layered Shell, Bottom Layer**.

3 In the **Stress: Layered Shell, Bottom Layer** toolbar, click **Plot**.

Stress: Layered Shell, Top Layer

1 Right-click **Stress: Layered Shell, Bottom Layer** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Stress: Layered Shell, Top Layer` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Stress: Layered Shell, Top Layer** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `misesTop_solid`.

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 **Dataset** list, choose **Layered Material: Top Layer**.

Stress: Layered Shell, Top Layer

1 In the **Model Builder** window, collapse the **Results** > **Stress: Layered Shell, Top Layer** node.

2 In the **Model Builder** window, click **Stress: Layered Shell, Top Layer**.

3 In the **Stress: Layered Shell, Top Layer** toolbar, click **Plot**.

Stress, Layered Shell-Solid Cladding

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Stress, Layered Shell-Solid Cladding` in the **Label** text field.

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

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** checkbox. In the associated text field, type `Y-coordinate (m)`.

6 Select the **y-axis label** checkbox. In the associated text field, type `von Mises stress (MPa)`.

Line Graph 1

1 Right-click **Stress, Layered Shell-Solid Cladding** and choose **Line Graph**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `17, 19, 21` in the **Selection** text field.

5 Click **OK**.

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

7 In the **Expression** text field, type `misesTop_lshell`.

8 Click to expand the **Legends** section. Select the **Show legends** checkbox.

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

10 In the table, enter the following settings:

Legends
`Layered Shell`

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 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `30` in the **Selection** text field.

6 Click **OK**.

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

8 In the **Expression** text field, type `solid.mises`.

9 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

10 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Solid (Reference)`

Stress, Layered Shell-Solid Cladding

1 In the **Model Builder** window, collapse the **Results** > **Stress, Layered Shell-Solid Cladding** node.

2 In the **Model Builder** window, click **Stress, Layered Shell-Solid Cladding**.

3 In the **Stress, Layered Shell-Solid Cladding** toolbar, click **Plot**.

Stress, Layered Shell-Shell Cladding

1 Right-click **Stress, Layered Shell-Solid Cladding** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Stress, Layered Shell-Shell Cladding` in the **Label** text field.

Line Graph 1

1 In the **Model Builder** window, expand the **Stress, Layered Shell-Shell Cladding** node, then click **Line Graph 1**.

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

3 Click **Clear Selection**.

4 Select Edge 151 only.

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 Edge 156 only.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Reversed arc length**.

Stress, Layered Shell-Shell Cladding

1 In the **Model Builder** window, collapse the **Results > Stress, Layered Shell-Shell Cladding** node.

2 In the **Model Builder** window, click **Stress, Layered Shell-Shell Cladding**.

3 In the **Stress, Layered Shell-Shell Cladding** toolbar, click **Plot**.

Stress, Layered Shell-Shell Transition

1 Right-click **Stress, Layered Shell-Shell Cladding** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Stress, Layered Shell-Shell Transition` in the **Label** text field.

3 Locate the **Plot Settings** section. In the **x-axis label** text field, type `X-coordinate (m)`.

Line Graph 1

1 In the **Model Builder** window, expand the **Stress, Layered Shell-Shell Transition** node, then click **Line Graph 1**.

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

3 Click **Clear Selection**.

4 Select Edges 18, 42, 69, and 108 only.

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 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `31, 56, 92, 124` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Line Graph**, locate the **x-Axis Data** section.

8 From the **Parameter** list, choose **Arc length**.

Stress, Layered Shell-Shell Transition

1 In the **Model Builder** window, collapse the **Results** > **Stress, Layered Shell-Shell Transition** node.

2 In the **Model Builder** window, click **Stress, Layered Shell-Shell Transition**.

3 In the **Stress, Layered Shell-Shell Transition** toolbar, click **Plot**.

Stress, Layered Shell-Solid Transition

1 Right-click **Stress, Layered Shell-Shell Transition** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Stress, Layered Shell-Solid Transition` in the **Label** text field.

Line Graph 1

1 In the **Model Builder** window, expand the **Stress, Layered Shell-Solid Transition** node, then click **Line Graph 1**.

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

3 Click **Clear Selection**.

4 Select Edges 23, 48, 74, and 112 only.

Line Graph 2

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

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

3 In the **Expression** text field, type `misesTop_solid`.

4 Locate the **Selection** section. Click **Clear Selection**.

5 Click **Paste Selection**.

6 In the **Paste Selection** dialog, type `40, 66, 101, 132` in the **Selection** text field.

7 Click **OK**.

Stress, Layered Shell-Solid Transition

1 In the **Model Builder** window, collapse the **Results** > **Stress, Layered Shell-Solid Transition** node.

2 In the **Model Builder** window, click **Stress, Layered Shell-Solid Transition**.

3 In the **Stress, Layered Shell-Solid Transition** toolbar, click **Plot**.

Appendix — Geometry 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 Click **Done**.

GEOMETRY 1

Work Plane 1 (wp1)

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

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

3 Clear the **Unite objects** checkbox.

4 Click **Go to Plane Geometry**.

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 `0.4`.

4 In the **Height** text field, type `0.5`.

5 Locate the **Position** section. In the **xw** text field, type `0.3`.

6 In the **yw** text field, type `-0.5`.

Work Plane 1 (wp1) > Rotate 1 (rot1)

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

2 Select the object **r1** 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 `90 180 270`.

6 Locate the **Center of Rotation** section. In the **xw** text field, type `0.5`.

7 In the **yw** text field, type `0.5`.

8 In the **Work Plane** toolbar, click **Build All**.

Extrude 1 (ext1)

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

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

3 From the **Extrude from** list, choose **Faces**.

4 On the object **wp1.rot1(3)**, select Boundary 1 only.

5 On the object **wp1.rot1(2)**, select Boundary 1 only.

6 Locate the **Distances** section. In the table, enter the following settings:

Distances (m)
0.2

7 Select the **Reverse direction** checkbox.

8 Click **Build Selected**.

Move 1 (mov1)

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

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

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

4 In the **z** text field, type `0.1`.

5 Click **Build Selected**.

Move 2 (mov2)

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

2 Select the object **wp1.rot1(1)** only.

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

4 In the **z** text field, type `-0.02`.

5 Click **Build Selected**.

Work Plane 2 (wp2)

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

Work Plane 2 (wp2) > Plane Geometry

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

Work Plane 2 (wp2) > 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 `1.4`.

4 In the **Height** text field, type `1`.

5 Locate the **Position** section. In the **xw** text field, type `-0.2`.

Work Plane 2 (wp2) > Rectangle 2 (r2)

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 `0.2`.

4 In the **Height** text field, type `1`.

5 Locate the **Position** section. In the **xw** text field, type `-0.2`.

Move 3 (mov3)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Transforms** > **Move**.

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

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

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

5 Locate the **Displacement** section. In the **y** text field, type `2.5`.

6 Click **Build Selected**.

Extrude 2 (ext2)

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

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

3 Select the object **wp2** only.

4 In the **Settings** window for **Extrude**, locate the **General** section.

5 From the **Extrude from** list, choose **Faces**.

6 On the object **mov3(4)**, select Boundary 1 only.

7 On the object **mov3(5)**, select Boundaries 1 and 2 only.

8 Locate the **Distances** section. In the table, enter the following settings:

Distances (m)
0.01
0.02

9 Click **Build Selected**.

Extrude 3 (ext3)

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

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

3 From the **Extrude from** list, choose **Faces**.

4 On the object **mov3(3)**, select Boundary 1 only.

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

Distances (m)
0.01
0.02

Form Union (fin)

In the **Geometry** toolbar, click **Build All**.