Created in COMSOL Multiphysics 6.1

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.

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 members whereas the reference geometry is modeled with only solid members.

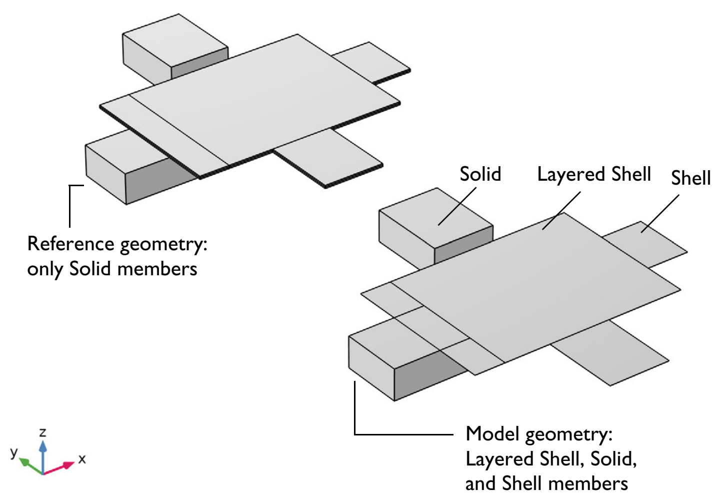

Figure 1: The model and reference geometry showing different structural members in different parts of the geometry.

LAYERED SHELL-STRUCTURE COUPLINGS

The model geometry has three sections. The middle section is modeled as a layered shell member, while side supports are modeled using solid and shell members 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 members 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.

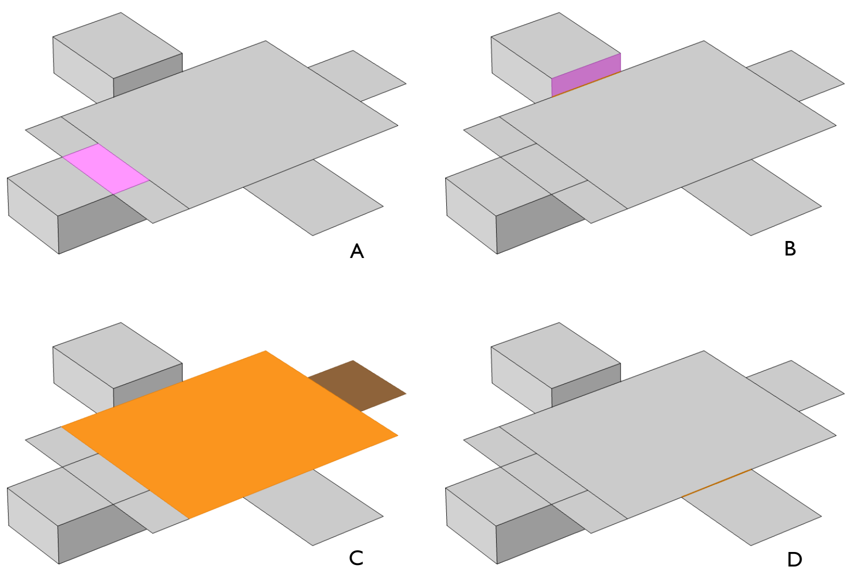

Figure 2: Different connections of layered shell with other structural members on different sides are as follows:(A) layered shell-solid cladding (B) layered shell-solid transition (C) layered shell-shell cladding (D) layered shell-shell transition.

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.

Note that in case of layered shell-solid transition coupling, only bottom layer of layered shell is connected to the solid whereas top layer is set to free. Similar connection is achieved in the reference model by slitting the degrees of freedom using Thin Elastic Layer node in the Solid Mechanics interface.

MATERIAL PROPERTIES

In the model geometry, layered shell and shell members are made up of carbon-epoxy composite material. The homogenized transversely isotropic material properties (Young's modulus, shear modulus, and Poisson's ratio) are given in Table 1 and density of the lamina is taken as 1700 kg/m^3.

TABLE 1: MATERIAL PROPERTIES OF A LAMINA.

Material property	Value
$\{E_1, E_2\}$	{134,9.2}(GPa)
$\{G_{12}\}$	{4.8}(GPa)
$\{\upsilon_{12}, \upsilon_{23}\}$	{0.28,0.28}

The solid members 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.

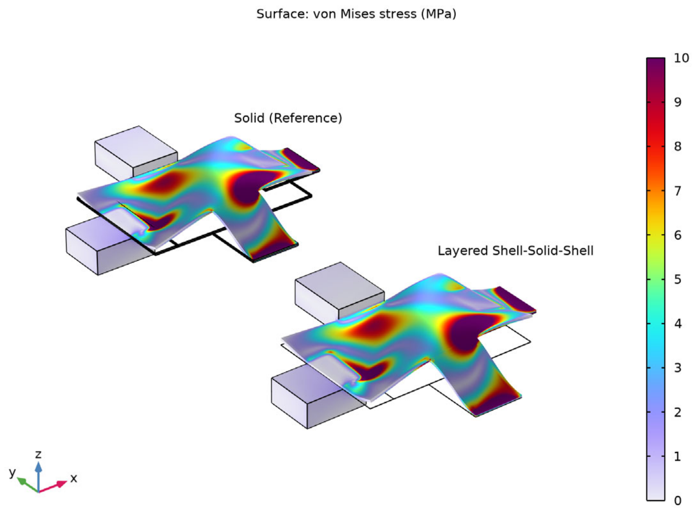

Figure 3: The comparison of von Mises stress distribution in both structural models.

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 members is also compared with the reference model. Figure 7 through Figure 10 illustrate such a comparison with a good overall qualitative and quantitative match.

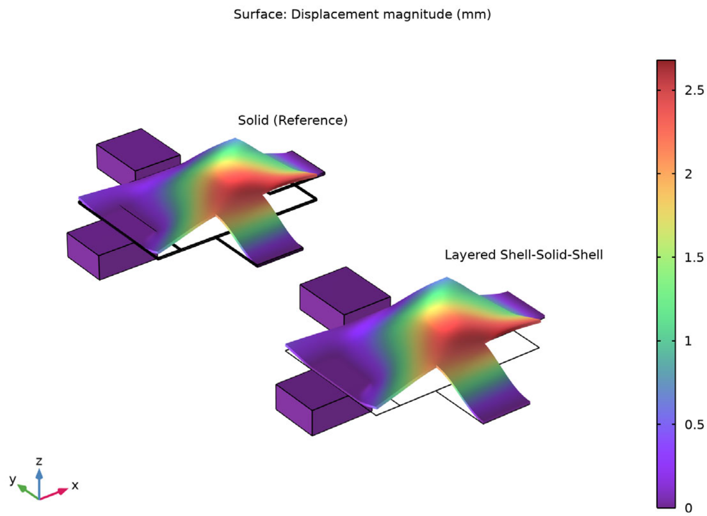

Figure 4: The comparison of total displacement distribution in both structural models.

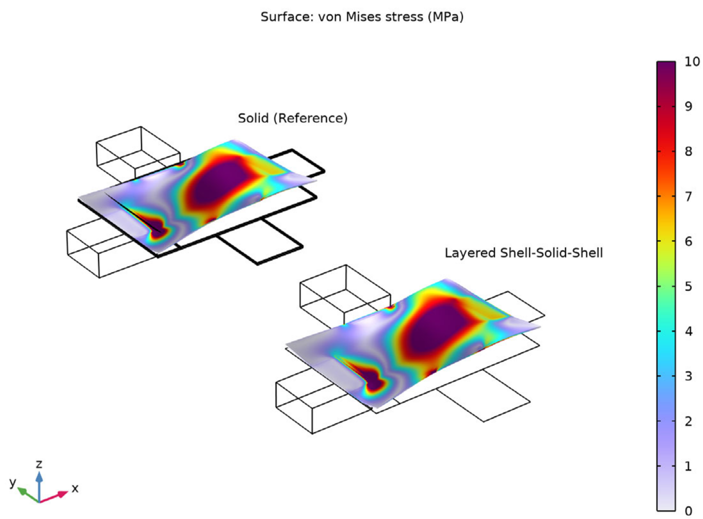

Figure 5: The comparison of von Mises stress distribution in the bottom layer of both structural models.

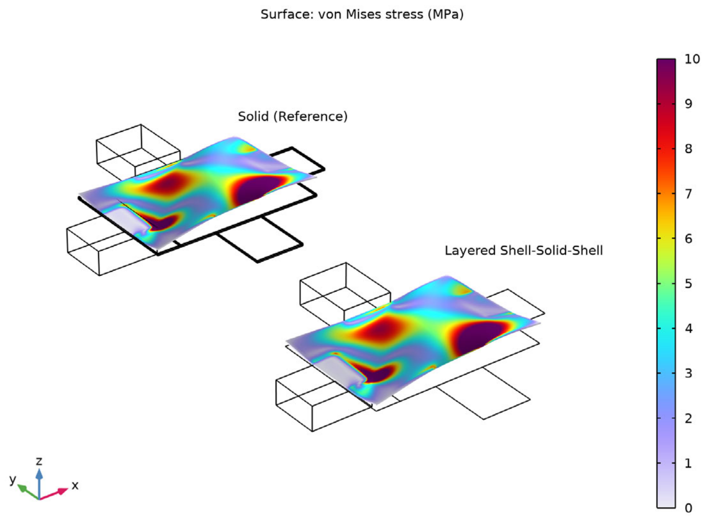

Figure 6: The comparison of von Mises stress distribution in the top layer of both structural models.

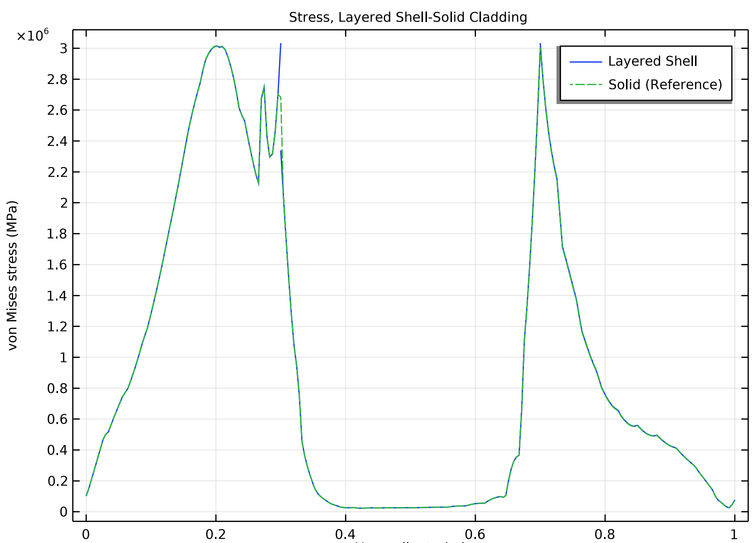

Figure 7: von Mises stress along the common edge for layered shell-solid cladding coupling.

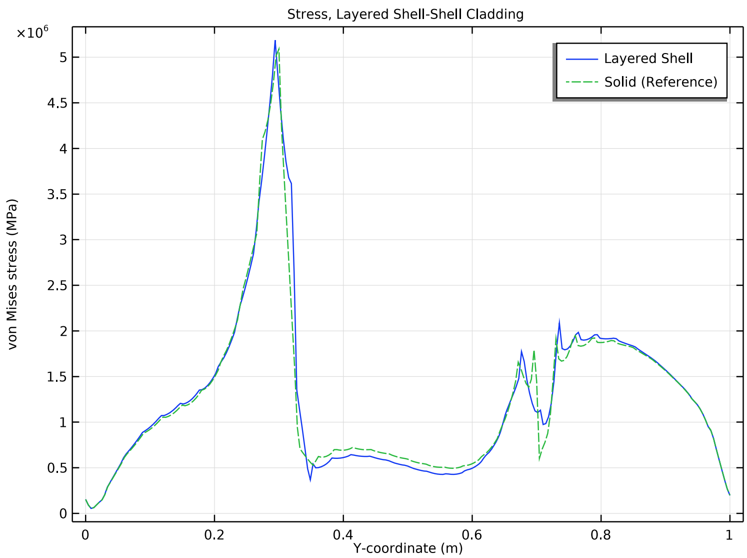

Figure 8: von Mises stress along the common edge for layered shell-shell cladding coupling.

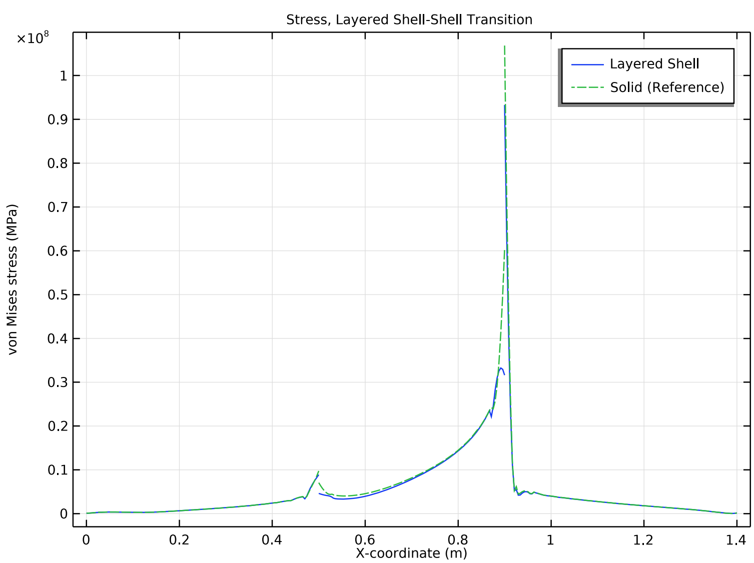

Figure 9: von Mises stress along the common edge for layered shell-shell transition coupling.

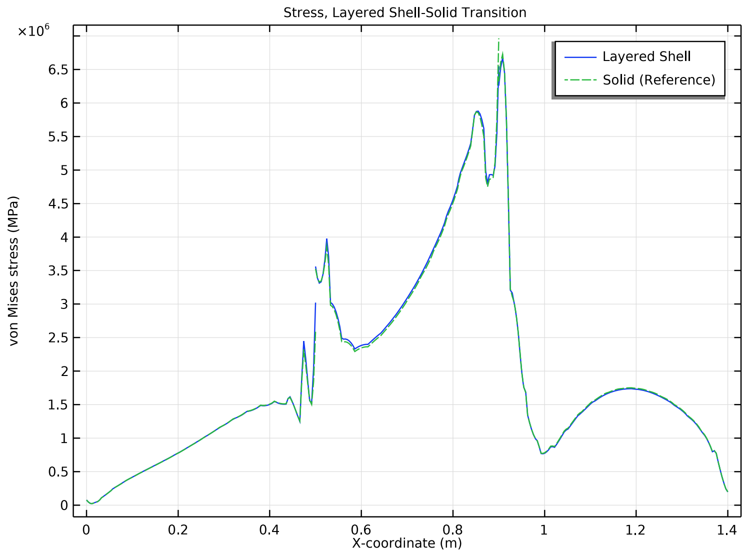

Figure 10: von Mises stress along the common edge for layered shell-solid transition coupling.

Notes About the COMSOL Implementation

- Modeling a composite laminate requires a surface geometry (2D), in general called 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. Optionally, you can also specify the interface materials between the layers and control mesh elements in each layer.
- 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 **Layered Linear Elastic Material** 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.

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
a	1[m]	1 m	Side length
th	1e-2[m]	0.01 m	Layer thickness
F	10[kN]	10000 N	Total load

If you do not want to build all the geometry, you can load the geometry sequence from the stored model. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Geometry 1** and choose **Insert Sequence**. Browse to the model's Application Libraries folder and double-click the file `layered_shell_structure_connection.mph`. You can then continue to the **Definitions** section below.

To build the geometry from scratch, continue here.

GEOMETRY 1

Work Plane 1 (wp1)

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

Work Plane 1 (wp1)>Plane Geometry

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

Work Plane 1 (wp1)>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*a`.
4. In the **Height** text field, type `0.5*a`.
5. Locate the **Position** section. In the **xw** text field, type `0.3*a`.
6. In the **yw** text field, type `-0.5*a`.

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** check box.
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*a`.

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

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

Split 1 (spl1)

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

2 In the **Settings** window for **Split**, locate the **Input** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `wp1` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Split**, click **Build Selected**.

Extrude 1 (ext1)

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 **spl1(1)**, select Boundary 1 only.

5 On the object **spl1(3)**, select Boundary 1 only.

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

Distances (m)
`20*th`

7 Select the **Reverse direction** check box.

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 `10*th`.

5 Click **Build Selected**.

Move 2 (mov2)

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

2 Select the object **spl1(4)** only.

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

4 In the **z** text field, type `-2*th`.

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*a`.

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

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

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*a`.

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

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

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** check box.

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

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)
th
2*th

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)
th
2*th

6 Click **Build Selected**.

Form Union (fin)

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

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

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

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 box, 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 box, 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(2*th, mean(lshell.mises))		von Mises stress

Add required materials and layered material first.

ADD MATERIAL

1 In the **Home** 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 **Home** toolbar, click **Add Material** to close the **Add Material** window.

GLOBAL DEFINITIONS

Carbon-Epoxy

1 In the **Model Builder** window, under **Global Definitions** right-click **Materials** and choose **Blank Material**.
2 In the **Settings** window for **Material**, type `Carbon-Epoxy` in the **Label** text field.

Layered Material 1 (lmat1)

1 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 (deg)	Thickness	Mesh elements
`Layer 1`	Carbon-Epoxy (mat2)	`0.0`	`th`	`1`

4 Click **Add**.
5 In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
`Layer 2`	Carbon-Epoxy (mat2)	`45`	`th`	`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 box, 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 box, 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 **Carbon-Epoxy (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 box, 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**.

4 Select the **Transversely isotropic** check box.

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 Select the **Transversely isotropic** check box.

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

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

7 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 (Z-X-Z)** 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)**.

To model the solid domains as disconnected, which are geometrically in union state, add **Thin Elastic Layer** node with zero stiffness.

Thin Elastic Layer 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Thin Elastic Layer**.

2 In the **Settings** window for **Thin Elastic Layer**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

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

5 Click **OK**.

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 $\mathbf{F}_{\text{tot}}$ 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 box, 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 box, type `11-13, 25` in the **Selection** text field.

6 Click **OK**.

Face Load 1

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

Linear Elastic Material 1

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

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

3 Select the **Transversely isotropic** check box.

Face Load 1

1 In the **Model Builder** window, click **Face Load 1**.

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 box, type `37, 69` in the **Selection** text field.

6 Click **OK**.

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

Layered Linear Elastic Material 1

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

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

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

4 Select the **Transversely isotropic** check box.

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

6 In the **Paste Selection** dialog box, type `37, 69` in the **Selection** text field.

7 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** check box.

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

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

7 Click **Paste Selection**.

8 In the **Paste Selection** dialog box, 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** check box.

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

13 Click **Paste Selection**.

14 In the **Paste Selection** dialog box, 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.

Now enter the orthotropic material properties of Carbon-Epoxy material.

GLOBAL DEFINITIONS

Carbon-Epoxy (mat2)

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Carbon-Epoxy (mat2)**.

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	{Evect1, Evect2}	`{134e9, 9.2e9}`	Pa	Transversely isotropic
Poisson's ratio	{nuvect1, nuvect2}	`{0.28, 0.28}`	1	Transversely isotropic

Property	Variable	Value	Unit	Property group
Shear modulus	Gvect1	{4.8e9}	N/m²	Transversely isotropic
Density	rho	1700	kg/m³	Basic

MESH 1

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`.

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 Click **Paste Selection**.

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

5 Click **OK**.

6 In the **Settings** window for **Mapped**, 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** check box.

4 In the **Home** toolbar, click **Compute**.

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

RESULTS

Layered Material 1

1 In the **Model Builder** window, expand the **Results** node.

2 Right-click **Results>Datasets** and choose **More Datasets>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** check box.

4 In the table, clear the check box 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** check box.

4 In the table, clear the check box 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 From the **Unit** list, choose **MPa**.

5 Locate the **Coloring and Style** section. Click **Change Color Table**.

6 In the **Color Table** dialog box, select **Rainbow>Prism** in the tree.

7 Click **OK**.

8 In the **Settings** window for **Surface**, click to expand the **Range** section.

9 Select the **Manual color range** check box.

10 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 From the **Unit** list, choose **MPa**.

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

7 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 From the **Unit** list, choose **MPa**.

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

7 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** check box.

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 From the **Unit** list, choose **mm**.

5 Locate the **Range** section. Clear the **Manual color range** check box.

6 Locate the **Coloring and Style** section. Click **Change Color Table**.

7 In the **Color Table** dialog box, select **Rainbow>SpectrumLight** in the tree.

8 Click **OK**.

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`.

4 From the **Unit** list, choose **mm**.

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`.

4 From the **Unit** list, choose **mm**.

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 **Model Builder** window, right-click **Stress** and choose **Duplicate**.

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

Surface 3

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

2 Right-click **Results>Stress: Layered Shell, Bottom Layer>Surface 3** and choose **Delete**.

Surface 1

1 In the **Model Builder** window, under **Results>Stress: Layered Shell, Bottom Layer** click **Surface 1**.

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

3 In the **Expression** text field, type `misesBot_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: Bottom Layer**.

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 **Home** toolbar, click **Add Plot Group** and choose **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** check box. In the associated text field, type `Y-coordinate (m)`.

6 Select the **y-axis label** check box. 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 box, 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 From the **Unit** list, choose **MPa**.

9 Click to expand the **Legends** section. Select the **Show legends** check box.

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

11 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 box, 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.

4 Select the **x-axis label** check box. In the associated 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 box, 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 box, 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**.