Created in COMSOL Multiphysics 6.1

Mixed-Mode Delamination of a Composite Laminate

Introduction

Interfacial failure or delamination in a composite material can be simulated with a *cohesive zone model* (CZM). A key ingredient of a cohesive zone model is a traction-separation law that describes the softening in the cohesive zone near the delamination tip. This example shows the implementation of a CZM with a bilinear traction-separation law in a laminated composite using the Layered Shell interface. The capabilities of the CZM to predict mixed-mode softening and delamination propagation are demonstrated in a model of mixed-mode bending of a composite beam.

This example is an extension of the Structural Mechanics Module Application Library model *Mixed-Mode Debonding of a Laminated Composite* in which the Layered Shell interface is used to model delamination. The results are compared with the model created using the Solid Mechanics interface.

Model Definition

COHESIVE ZONE MODEL (CZM)

The CZM used in this example is defined using the displacement based damage model available in the **Delamination** node. The model is used to predict crack propagation at the interface of a laminated composite beam under mixed-mode loading. The material properties needed for this constitutive model are summarized in Table 1.

TABLE 1: SUMMARY OF MATERIAL PROPERTIES OF THE CZM INTERFACE. THE VALUES ARE FOR AS4/PEEK.

PROPERTY	SYMBOL	VALUE
Normal tensile strength	σ_t	80 MPa
Shear strength	σ_s	100 MPa
Penalty stiffness	p_n	10^6 N/mm^3
Critical energy release rate, tension	G_{ct}	969 J/m^2
Critical energy release rate, shear	G_{cs}	1719 J/m^2
Exponent of Benzeggagh and Kenane (B-K) criterion	α	2.284

The CZM is defined using a bilinear traction-separation law. Traction increases linearly with a stiffness p_n until the opening crack reaches a damage initiation displacement u_0. When the crack opens beyond u_0, the material softens irreversibly and the stiffness decreases as a function of increasing damage d. The material fails once the stiffness has decreased to zero, that is when $d = 1$. This happens at the ultimate displacement u_f.

The values of u_0 and u_f depend on whether the separation displacement is normal (mode I) or tangential (mode II and III) to an interface. For the mixed mode, a combination is used. For the displacement based damage model, two different criteria are available to define this combination. Here the model by Benzeggagh and Kenane is used.

MIXED-MODE BENDING OF A LAMINATED COMPOSITE BEAM

A commonly used method to measure the delamination resistance of composite materials is the mixed-mode bending (MMB) test, see Ref. 1 and Ref. 2. This experimental procedure is here modeled to demonstrate the capabilities of the CZM.

The geometry of the test specimen is illustrated in Figure 1. It consists of a beam cracked along a ply interface halfway through its thickness. The initial crack length is c_1. The beam is supported at the outermost bottom edges. A mixed-mode bending load is produced as the result of forces applied to the top edges at the cracked end and at the center of the beam.

Because of the symmetry, only half of the beam is modeled and a **Symmetry** boundary condition is applied. As the Layered Shell interface only requires a boundary selection, only the middle surface of the 3D geometry is modeled and the *Midplane on boundary option* is chosen in the **Layered Material Link** node.

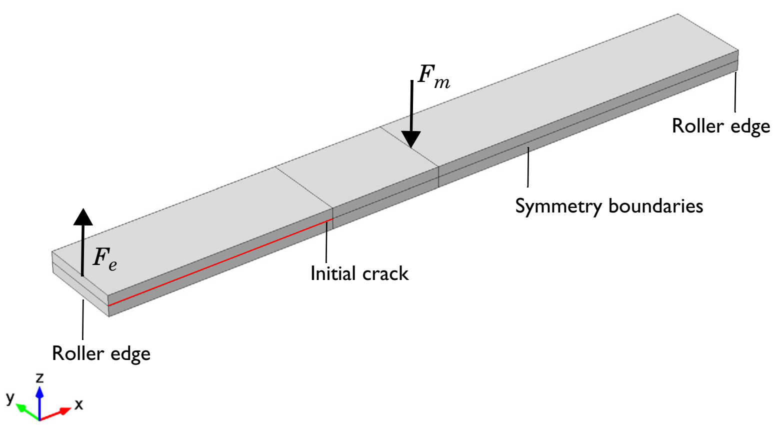

Figure 1: The geometry of the test specimen.

The material properties are those of AS4/PEEK unidirectional laminates. The orthotropic linear elastic properties assume that the longitudinal direction is aligned with the global X direction. The material properties of the laminate composite are listed in Table 2.

TABLE 2: LAMINATED COMPOSITE MATERIAL PROPERTIES.

PROPERTY	SYMBOL	VALUE
Young's modulus, along fibers	E_X	122.7 GPa
Young's modulus, across fibers	$E_Y=E_Z$	10.1 GPa
Poisson's ratio	ν_{YZ}	0.45
Poisson's ratio	$\nu_{XY}=\nu_{XZ}$	0.25
Shear modulus	G_{YZ}	3.7 GPa
Shear modulus	$G_{XY}=G_{XZ}$	5.5 GPa

The beam is supported on the bottom at its outer edges. A lever that sits on top of the beam applies a load. The lever is also attached to the cracked end and swivels around a contact area at the center of the beam. The lever is pushed down at the opposite free end, thereby simultaneously applying mode I and mode II loads on the test specimen. Arbitrary ratios of mixed-mode loading can be adjusted by varying the length of the lever l_l.

In this example, the lever is omitted. Instead, the forces that the lever transmits to the beam are applied directly. A pulling force $\boldsymbol{F}_e$ is acting on the cracked side of the beam. At the center, a force $\boldsymbol{F}_m$ pushes down. The desired mixed-mode ratio m_m regulates the ratio of their magnitudes l_r via

$$l_r = 8\left(\frac{6m_m + \sqrt{3m_m(1-m_m)}}{3 + 9m_m + 8\sqrt{3m_m(1-m_m)}}\right).$$

Further details on the background of the equation above can be found in Ref. 1 and Ref. 2.

Results and Discussion

The model is analyzed for a mixed-mode ratio of 50%. The von Mises stress distribution of the last computed parameter step is shown in Figure 2 for the Layered Shell model. When compared with the Solid Mechanics model (see plot in model), the distribution of stresses from the Layered Shell model matches closely. At this step, the initial crack has propagated along the interface as shown in Figure 3. Here also the interface health predicted by the Layered Shell model show excellent agreement with the Solid Mechanics model.

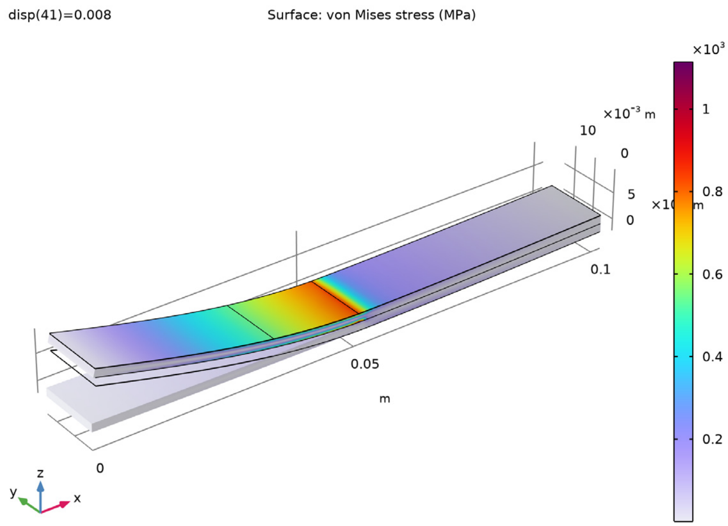

Figure 2: The Von Mises stress distribution at the last computation step.

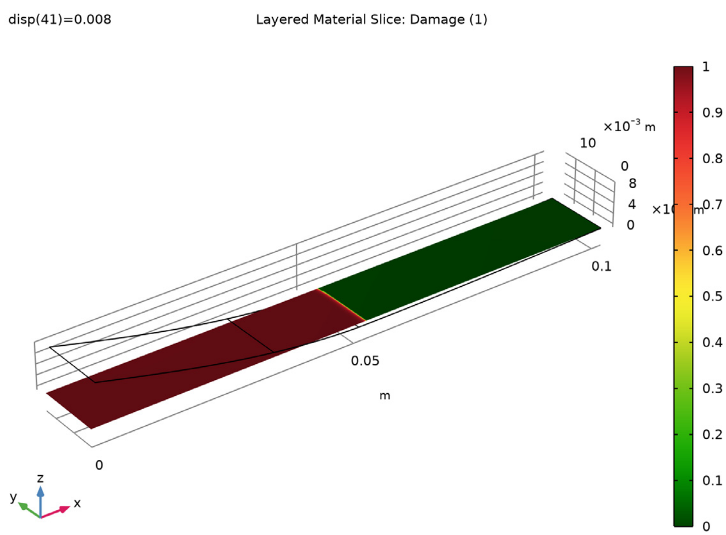

Figure 3: Plot showing the health of the laminate interface. The debonded part is shown in red, the intact part in green.

One of the outputs of the MMB test is a load-displacement curve. Both the load and displacement are measured at the endpoint of the lever that is used to apply the load to the test specimen. Since the lever is not explicitly modeled, the load-displacement data has to be deduced from the simulation results. Details of the analysis are contained in Ref. 1 and Ref. 2, with the following result.

The force $\boldsymbol{F}_{lp}$ at the load point of the lever can be determined from the load applied to the cracked edge in the model $\boldsymbol{F}_e$ and the lengths of the test beam l_b and load lever l_l:

$$\boldsymbol{F}_{lp} = \boldsymbol{F}_e \frac{l_b/2}{l_l} .$$

The length of the load lever above depends on the desired mode mixture m_m:

$$l_l = \frac{(l_b/2)\left(\frac{1}{2}\sqrt{3\frac{1-m_m}{m_m}}+1\right)}{3-\frac{1}{2}\sqrt{3\frac{1-m_m}{m_m}}} .$$

Note, that l_l measures the length from the center of the test specimen to the free end of the load lever.

The displacement at the load point u_{lp} is computed from the mode I opening at the cracked edge u_{Ie} and the z-displacement at the center of the beam w_c according to

$$u_{lp} = \left(\frac{3l_l-l_b/2}{4l_b/2}\right)u_{Ie}+\left(\frac{l_l+l_b/2}{l_b/2}\right)(-w_c+u_{Ie}/4) .$$

The resulting load-displacement curve is shown in Figure 4 for both the Layered Shell and Solid Mechanics models, the two curves closely match with each other. The load-displacement curve confirms what Figure 3 displayed. The maximal load that the beam with the initial crack can carry is exceeded and delamination occurs. After the peak load, the load decreases until the displacement reaches around 7 mm. This point approximately corresponds to when the crack reaches the center of the specimen. Thereafter, the load starts to increase again, but with a much lower stiffness than before delamination.

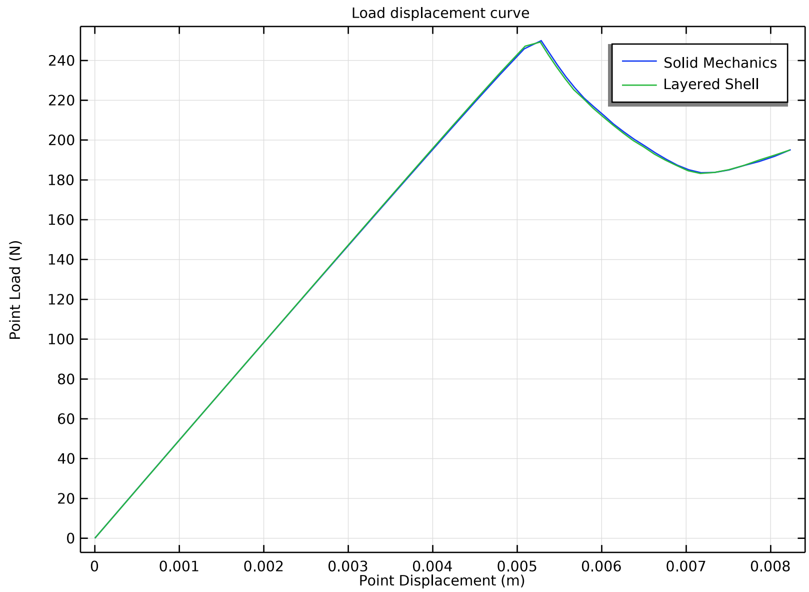

Figure 4: Load-displacement curve of the MMB test at 50% mixed-mode loading.

Notes About the COMSOL Implementation

- To implement a cohesive zone model in the **Layered Shell** interface, use the **Delamination** node, in which you can model adhesion, delamination and contact after delamination. There are two different ways to specify adhesion stiffness, with the default being taken from the interface material properties. Delamination laws based on either displacement or energy are used to model the separation of the interface. The contact after delamination is modeled by pressure penalty contact method.
- The **Delamination** node can be used to model already delaminated region by setting initial state to *delaminated*. To model the portion of interface that is not delaminated, set the initial state to *bonded*.
- The **Delamination** node is only applicable to internal interface of composite laminates.
- Modeling a composite laminated shell requires a surface geometry (2D), in general called a base surface, and a **Layered Material** node that 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.

References

1. P.P. Camanho, C.G. Davila, and M.F. De Moura, “Numerical Simulation of Mixed-mode Progressive Delamination in Composite Materials,” *Journal of composite materials*, vol. 37, no. 16, pp. 1415–1438, 2003.

2. J.R. Reeder, and J.R. Crews Jr., “Mixed-mode bending method for delamination testing,” *AiAA Journal*, vol. 28, no. 7, pp. 1270–1276, 2003.

Application Library path: `Composite_Materials_Module/Delamination/mixed_mode_delamination`

Modeling Instructions

ROOT

In this example you will start from an existing model that is an example in the Structural Mechanics Module.

APPLICATION LIBRARIES

1 From the **File** menu, choose **Application Libraries**.

2 In the **Application Libraries** window, select **Structural Mechanics Module>Contact and Friction>cohesive_zone_debonding** in the tree.

3 Click **Open**.

COMPONENT [SOLID MECHANICS]

1 In the **Model Builder** window, click **Component 1 (comp1)**.

2 In the **Settings** window for **Component**, type `Component [Solid Mechanics]` in the **Label** text field.

STUDY [SOLID MECHANICS]

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

2 In the **Settings** window for **Study**, type `Study [Solid Mechanics]` in the **Label** text field.

RESULTS

Interface Health, Stress (solid)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Stress (solid)** and **Interface Health**.

2 Right-click and choose **Group**.

Solid Mechanics Plots

1 In the **Settings** window for **Group**, type `Solid Mechanics Plots` in the **Label** text field.

2 In the **Model Builder** window, collapse the **Solid Mechanics Plots** node.

Add a new component in order to set up a similar model with the Layered Shell interface.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component>3D**.

COMPONENT [LAYERED SHELL]

In the **Settings** window for **Component**, type `Component [Layered Shell]` in the **Label** text field.

GEOMETRY 2

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 `lb`.

4 In the **Height** text field, type `wb/2`.

Work Plane 1 (wp1)>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 `lb/2-cl`.

4 In the **Height** text field, type `wb/2`.

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

Work Plane 1 (wp1)>Union 1 (uni1)

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

2 Click the **Select Box** button in the **Graphics** toolbar.

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

Form Union (fin)

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

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

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

DEFINITIONS (COMP2)

Integration Edge

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `Integration Edge` in the **Label** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Point**.

4 Select Point 1 only.

5 Locate the **Advanced** section. From the **Frame** list, choose **Material (X, Y, Z)**.

Integration Center

1 Right-click **Integration Edge** and choose **Duplicate**.

2 In the **Settings** window for **Integration**, type `Integration Center` in the **Label** text field.

3 Locate the **Source Selection** section. Click **Clear Selection**.

4 Select Point 5 only.

Load Point Variables

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

2 In the **Settings** window for **Variables**, type `Load Point Variables` in the **Label** text field.

3 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
u_IeL	intop3(lshell.atxd1(hb, w2))		Displacement at edge [Layered Shell]
w_cL	intop4(lshell.atxd1(hb, w2))		Displacement at center [Layered Shell]
u_lpL	(3*l1-lb/2)/4/(lb/2)*u_IeL+((l1+lb/2)/(lb/2))*(-w_cL+u_IeL/4)		Load point displacement [Layered Shell]
F_lpL	forceL*lb/2/l1		Load point force [Layered Shell]

MATERIALS

AS4/PEEK (mat1)

1 In the **Model Builder** window, expand the **Component [Solid Mechanics] (comp1)>Materials** node.

2 Right-click **Component [Solid Mechanics] (comp1)>Materials>AS4/PEEK (mat1)** and choose **Copy**.

COMPONENT [SOLID MECHANICS] (COMP1)

In the **Model Builder** window, collapse the **Component [Solid Mechanics] (comp1)** node.

GLOBAL DEFINITIONS

AS4/PEEK (mat2)

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

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	AS4/PEEK (mat2)	0.0	hb/2	2

4 Click + **Add**.

MATERIALS

Layered Material Link 1 (llmat1)

In the **Model Builder** window, under **Component [Layered Shell] (comp2)** right-click **Materials** and choose **Layers>Layered Material Link**.

The geometry is in an XY-plane in which the fibers are oriented with respect to the X direction. Hence set the first axis of the laminate coordinate system in the X direction. Also set the frame of the **Boundary System** to reference configuration.

DEFINITIONS (COMP2)

Boundary System 2 (sys2)

1 In the **Settings** window for **Boundary System**, locate the **Settings** section.
2 Find the **Coordinate names** subsection. From the **Axis** list, choose **x**.
3 From the **Frame** list, choose **Reference configuration**.

ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.
2 Go to the **Add Physics** window.
3 In the tree, select **Structural Mechanics>Layered Shell (lshell)**.
4 Click **Add to Component [Layered Shell]** in the window toolbar.
5 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

LAYERED SHELL (LSHELL)

For the portion of interface that is initially delaminated, the initial state in **Delamination** node can be set to **Delaminated**.

Delamination 1

1 Right-click **Component [Layered Shell] (comp2)>Layered Shell (lshell)** and choose **Material Models>Delamination**.
2 Select Boundary 1 only.
3 In the **Settings** window for **Delamination**, locate the **Initial State** section.
4 From the list, choose **Delaminated**.
5 Locate the **Contact** section. In the p_{n} text field, type `pn`.

   For the portion of interface that is not yet delaminated, the initial state in **Delamination** node can be set to **Bonded**. To model contact between delaminated interfaces, the penalty factor taken from the adhesive stiffness.

Delamination 2

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

2 Select Boundaries 2 and 3 only.

3 In the **Settings** window for **Delamination**, locate the **Adhesion** section.

4 From the **Adhesive stiffness** list, choose **User defined**.

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

`pn`	t1
`pn`	t2
`pn`	n

6 Locate the **Delamination** section. In the σ_t text field, type `sigmat`.

7 In the σ_s text field, type `sigmas`.

8 In the G_{ct} text field, type `Gct`.

9 In the G_{cs} text field, type `Gcs`.

10 From the **Mixed mode criterion** list, choose **Benzeggagh-Kenane**.

11 In the α text field, type `alpha`.

12 Locate the **Contact** section. From the **Penalty factor** list, choose **From adhesive stiffness**.

Symmetry 1

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

2 Select Edges 2, 5, and 8 only.

Load on Cracked Edge (Fe)

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

2 In the **Settings** window for **Edge Load**, type `Load on Cracked Edge (Fe)` in the **Label** text field.

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

4 Select Edge 1 only.

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

`0`	x
`0`	y
`forceL`	z

Load on Middle Edge (Fm)

1 Right-click **Load on Cracked Edge (Fe)** and choose **Duplicate**.

2 In the **Settings** window for **Edge Load**, type `Load on Middle Edge (Fm)` in the **Label** text field.

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

4 Select Edge 7 only.

5 Locate the **Force** section. Specify the $\mathbf{F}_{\mathrm{tot}}$ vector as

`0`	x
`0`	y
`-lr*forceL`	z

Prescribed Displacement, Interface 1

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

2 Select Edges 1 and 10 only.

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

4 From the **Apply to** list, choose **Bottom interface**.

5 Locate the **Prescribed Displacement** section. Select the **Prescribed in z direction** check box.

Prescribed Displacement, Interface 2

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

2 Select Point 1 only.

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

4 From the **Apply to** list, choose **Bottom interface**.

5 Locate the **Prescribed Displacement** section. Select the **Prescribed in x direction** check box.

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

7 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.

8 Click **OK**.

This is to make **Global Equations** accessible. Add a global equation to control the applied load with a monotonically increasing parameter.

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
`forceL`	`disp-u_IeL`	`0`	`0`	

4 Locate the **Units** section. Click **Define Dependent Variable Unit**.

5 Click **Select Dependent Variable Quantity**.

6 In the **Physical Quantity** dialog box, type `force` in the text field.

7 Click **Filter**.

8 In the tree, select **General>Force (N)**.

9 Click **OK**.

10 In the **Settings** window for **Global Equations**, locate the **Units** section.

11 Click **Select Source Term Quantity**.

12 In the **Physical Quantity** dialog box, type `length` in the text field.

13 Click **Filter**.

14 In the tree, select **General>Length (m)**.

15 Click **OK**.

Use the same meshing as for the solid model.

MESH 2

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 **All boundaries**.

Distribution 1

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

2 Select Edges 1, 4, 7, and 10 only.

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

4 In the **Number of elements** text field, type 3.

Distribution 2

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

2 Select Edge 2 only.

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

4 In the **Number of elements** text field, type 10.

Distribution 3

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

2 Select Edge 5 only.

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

4 In the **Number of elements** text field, type 50.

Distribution 4

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

2 Select Edges 8 and 9 only.

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

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type 20.

6 In the **Element ratio** text field, type 5.

7 Click **Build All**.

Disable the **Layered Shell** interface in the first study.

STUDY [SOLID MECHANICS]

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study [Solid Mechanics]** node, then click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Layered Shell (lshell)**.

Add a new study for the **Layered Shell** interface and disable the **Solid Mechanics** interface.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

4 Right-click and choose **Add Study**.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY [LAYERED SHELL]

In the **Settings** window for **Study**, type `Study [Layered Shell]` in the **Label** text field.

Step 1: Stationary

1 In the **Model Builder** window, under **Study [Layered Shell]** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Solid Mechanics (solid)**.

4 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
disp (Displacement parameter)	`range(0,2e-4,8e-3)`	

Solution 2 (sol2)

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** 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 `1e-4`.

5 In the **Model Builder** window, expand the **Study [Layered Shell]>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** node, then click **Displacement field (material and geometry frames) (comp2.u2)**.

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

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

8 In the **Scale** text field, type `1e-3`.

9 In the **Model Builder** window, under **Study [Layered Shell]>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** click **State variable forceL (comp2.ODE2)**.

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

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

12 In the **Scale** text field, type `200`.

Use a linear predictor.

13 In the **Model Builder** window, expand the **Study [Layered Shell]>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** node, then click **Parametric 1**.

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

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

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

17 From the **Predictor** list, choose **Linear**.

Switch to an undamped Newton method.

18 In the **Model Builder** window, under **Study [Layered Shell]>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** click **Fully Coupled 1**.

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

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

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

RESULTS

Surface 1

1 In the **Model Builder** window, expand the **Stress (lshell)** node, then click **Surface 1**.

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

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

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

Interface Health (lshell)

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Interface Health (lshell)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study [Layered Shell]/Solution 2 (4) (sol2)**.

4 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

Layered Material Slice 1

1 In the **Interface Health (lshell)** toolbar, click **More Plots** and choose **Layered Material Slice**.

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

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

4 Locate the **Through-Thickness Location** section. From the **Location definition** list, choose **Interfaces**.
5 Locate the **Coloring and Style** section. Click **Change Color Table**.
6 In the **Color Table** dialog box, select **Traffic>Traffic** in the tree.
7 Click **OK**.
8 In the **Interface Health (lshell)** toolbar, click **Plot**.

Interface Health (lshell), Stress (lshell)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Stress (lshell)** and **Interface Health (lshell)**.
2 Right-click and choose **Group**.

Layered Shell Plots

1 In the **Settings** window for **Group**, type `Layered Shell Plots` in the **Label** text field.
2 Right-click **Layered Shell Plots** and choose **Move Up**.
3 In the **Model Builder** window, collapse the **Layered Shell Plots** node.

Load Displacement Curve

1 In the **Model Builder** window, click **Load Displacement Curve**.
2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.
3 Select the **x-axis label** check box. In the associated text field, type `Point Displacement (m)`.
4 Select the **y-axis label** check box. In the associated text field, type `Point Load (N)`.

Global 1

1 In the **Model Builder** window, expand the **Load Displacement Curve** node, then click **Global 1**.
2 In the **Settings** window for **Global**, click to expand the **Title** section.
3 From the **Title type** list, choose **Manual**.
4 In the **Title** text area, type `Load Displacement Curve`.
5 Locate the **x-Axis Data** section. From the **Unit** list, choose **m**.
6 Click to expand the **Legends** section. Select the **Show legends** check box.
7 From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
Solid Mechanics

Global 2

1 Right-click **Results>Load Displacement Curve>Global 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study [Layered Shell]/Solution 2 (4) (sol2)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
2*comp2.F_lpL		Load

5 Locate the **x-Axis Data** section. In the **Expression** text field, type `comp2.u_lpL`.

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

7 Click to expand the **Coloring and Style** section. Locate the **Legends** section. In the table, enter the following settings:

Legends
Layered Shell

8 In the **Load Displacement Curve** toolbar, click **Plot**.

Global Evaluation 2

1 In the **Model Builder** window, under **Results>Derived Values** right-click **Global Evaluation 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study [Layered Shell]/Solution 2 (4) (sol2)**.

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

Expression	Unit	Description
2*comp2.F_lpL		Load

5 Click **Evaluate**.

Animation: Stress [Layered Shell]

1 In the **Results** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, type `Animation: Stress [Layered Shell]` in the **Label** text field.

3 Locate the **Scene** section. From the **Subject** list, choose **Stress (lshell)**.

4 Locate the **Playing** section. In the **Display each frame for** text field, type `0.3`.

5 Click to expand the **Advanced** section. Click **Show Frame**.

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