



Model created in COMSOL Multiphysics 6.4

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 this example.

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.



Read more about the Composite Materials Module in the COMSOL blog, [Introduction to the Composite Materials Module](#).

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 Benzeggagh and Kenane criterion 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.

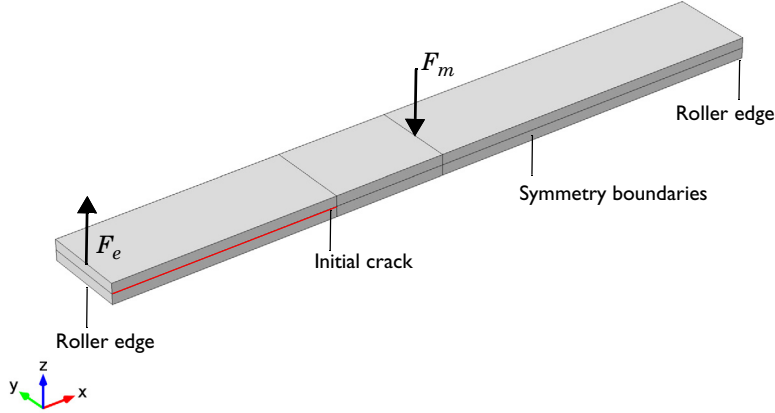


Figure 1: The geometry of the test specimen.

The material properties are those of AS4/PEEK unidirectional laminates and are listed in Table 2. The transverse isotropic linear elastic properties assume that the longitudinal direction is aligned with the global X direction. The AS4/PEEK unidirectional laminate is a built-in material in the **Composites** material library.

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	$G_{XY}=G_{XZ}$	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 F_e is acting on the cracked side of the beam. At the center, a force 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

For composite materials, von Mises stress is generally not an appropriate stress measure. In this model, however, it is applied because the stresses in directions other than the fiber direction are negligible, making von Mises stress a reasonable representation of the fiber stress in absolute terms. It is also used since the benchmark results from the **Solid Mechanics** interface are reported in terms of von Mises stress.

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.

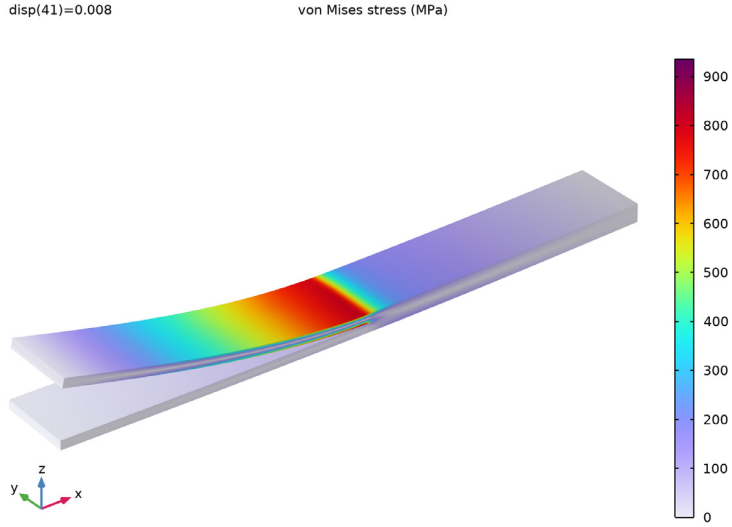


Figure 2: The von Mises stress distribution at the last computation step.

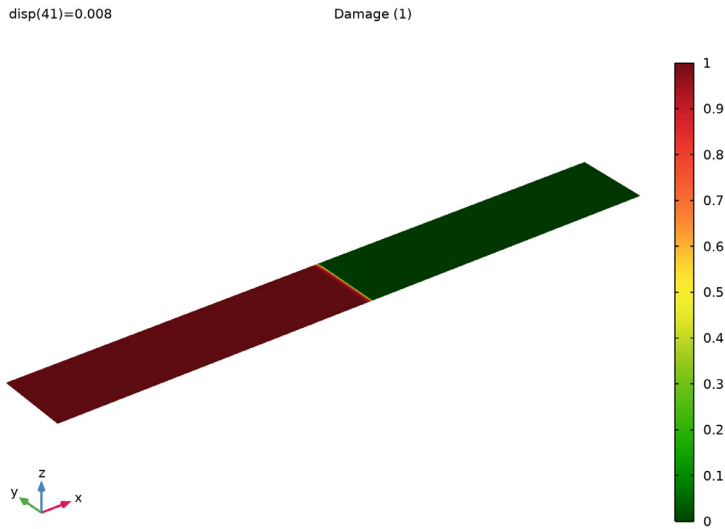


Figure 3: Plot showing the health of the laminate interface. The debonded part is shown in red, the intact part in green.

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 F_{lp} at the load point of the lever can be determined from the load applied to the cracked edge in the model F_e and the lengths of the test beam l_b and load lever l_l :

$$F_{lp} = F_e \frac{l_b/2}{l_l}.$$

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.

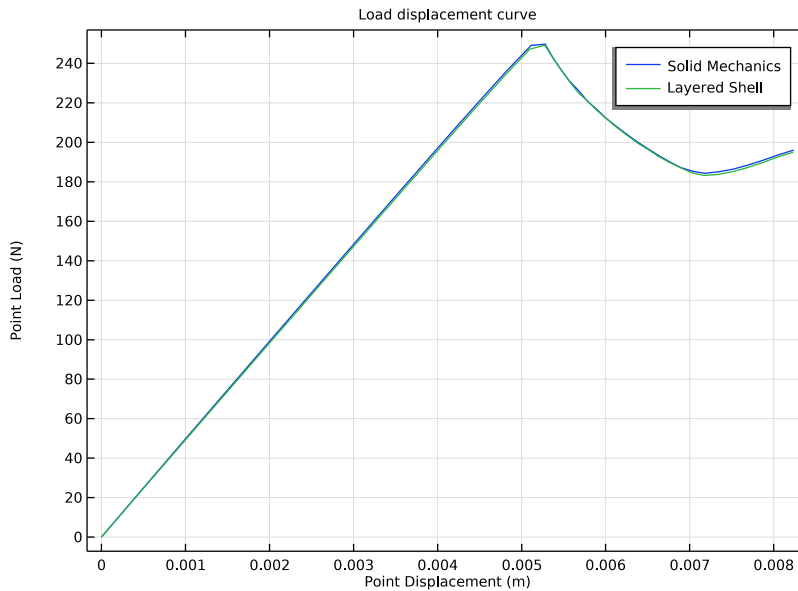


Figure 4: Load-displacement curve of the MMB test at 50% mixed-mode loading.

Notes About the COMSOL Implementation

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

References

1. P.P. Camanho, C.G. Davila, and M.F. De Moura, “Numerical Simulation of Mixed-mode Progressive Delamination in Composite Materials,” *J. Compos. Mater.*, 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 J.*, 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, Interface Health: Gauss Points, Stress (solid)

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

- 1 In the **Geometry** toolbar, click  **Work Plane**.
- 2 In the **Settings** window for **Work Plane**, click  **Go to Plane Geometry**.


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 $1b$.
- 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 $1b/2 - c1$.
- 4 In the **Height** text field, type $wb/2$.
- 5 Locate the **Position** section. In the **xw** text field, type $c1$.

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(1shell.atxd1(hb, w2))		Displacement at edge [Layered Shell]
w_cL	intop4(1shell.atxd1(hb, w2))		Displacement at center [Layered Shell]
u_lpL	$(3 \cdot l_1 - l_b/2)/4 / (l_b/2) \cdot u_{IeL} + ((l_1 + l_b/2) / (l_b/2)) \cdot (-w_{cL} + u_{IeL}/4)$		Load point displacement [Layered Shell]
F_lpL	forceL * l_b/2/l_1		Load point force [Layered Shell]


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 50%] (mat1)	0.0	0 rad	hb/2	2

4 Click  **Add**.

MATERIALS

Layered Material Link 1 (lmat1)

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 the **Add to Component [Layered Shell]** button 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 In the **Physics** toolbar, click  **Boundaries** and choose **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 p_n .

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


p_n	τ_1
p_n	τ_2
p_n	n

- 6 Locate the **Delamination** section. In the σ_{ts} text field, type sigmat .
- 7 In the σ_{ss} text field, type sigmas .
- 8 In the G_{ct} text field, type G_{ct} .
- 9 In the G_{cs} text field, type G_{cs} .
- 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}_{tot} vector as


0	x
0	y
-1r*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. From the **Displacement in z direction** list, choose **Prescribed**.


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. From the **Displacement in x direction** list, choose **Prescribed**.
- 6 Click the  **Show More Options** button in the **Model Builder** toolbar.
- 7 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics > Equation 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 I (ODEI)

- 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	$f(u, ut, utt, t)$ (l)	Initial value (u_0) (l)	Initial value (ut_0) (l/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, type **force** in the text field.
- 7 In the tree, select **General > Force (N)**.
- 8 Click **OK**.
- 9 In the **Settings** window for **Global Equations**, locate the **Units** section.
- 10 Click  **Select Source Term Quantity**.
- 11 In the **Physical Quantity** dialog, type **length** in the text field.
- 12 In the tree, select **General > Length (m)**.
- 13 Click **OK**.

Use the same meshing as for the solid model.

MESH 2

Copy 1



- 1 In the **Mesh** toolbar, click  **Copy** and choose **Copy**.
- 2 In the **Settings** window for **Copy**, locate the **Source Mesh** section.
- 3 From the **Mesh** list, choose **Mesh 1**.
- 4 Select Boundaries 7, 16, and 25 only.
- 5 Locate the **Destination Entities** section. Click to select the **Activate Selection** toggle button.
- 6 From the **Selection** list, choose **All boundaries**.
- 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 **Solve for** column of the table, under **Component [Layered Shell] (comp2)**, clear the checkbox for **Layered Shell (Ishell)**.
Add a new study for the **Layered Shell** interface and disable the **Solid Mechanics** interface.

ADD STUDY

- 1 In the **Study** 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 **Study** 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.


- 1 In the **Model Builder** window, under **Study [Layered Shell]** click **Step 1: Stationary**.
- 2 In the **Settings** window for **Stationary**, locate the **Study Settings** section.
- 3 Select the **Include geometric nonlinearity** checkbox.

- 4 Locate the **Physics and Variables Selection** section. In the **Solve for** column of the table, under **Component [Solid Mechanics] (comp1)**, clear the checkbox for **Solid Mechanics (solid)**.
- 5 Locate the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.
- 6 Click **+ Add**.
- 7 In the table, enter the following settings:

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

- 8 In the table, click to select the cell at row number 1 and column number 3.
- 9 Locate the **Study Settings** section. From the **Tolerance** list, choose **User controlled**.
- 10 In the **Relative tolerance** text field, type $1e-4$.

Solution 2 (sol2)

- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.
- 3 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)**.
- 4 In the **Settings** window for **Field**, locate the **Scaling** section.
- 5 From the **Method** list, choose **Manual**.
- 6 In the **Scale** text field, type $1e-3$.
- 7 In the **Model Builder** window, under **Study [Layered Shell] > Solver Configurations > Solution 2 (sol2) > Dependent Variables 1** click **Global Equations 1 (comp2.ODE1)**.
- 8 In the **Settings** window for **State**, locate the **Scaling** section.
- 9 From the **Method** list, choose **Manual**.
- 10 In the **Scale** text field, type 200.
Use a linear predictor.
- 11 In the **Model Builder** window, expand the **Study [Layered Shell] > Solver Configurations > Solution 2 (sol2) > Stationary Solver 1** node, then click **Parametric 1**.
- 12 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.
- 13 Select the **Tuning of step size** checkbox.
- 14 In the **Minimum step size** text field, type $1e-6$.


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

Switch to an undamped Newton method.

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

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

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

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

RESULTS

Surface 1

Update the default plot expression to von Mises stress to allow comparison with results from the **Solid Mechanics** interface.

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

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


3 In the **Expression** text field, type `Ishell.misesGp`.

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


5 Clear the **Description** checkbox.

Stress (Ishell)

1 In the **Model Builder** window, click **Stress (Ishell)**.

2 In the **Stress (Ishell)** toolbar, click  **Plot**.

Interface Health (Ishell)


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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study [Layered Shell]/ Solution 2 (3) (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 (Ishell)** 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. From the **Color table** list, choose **Traffic**.

Interface Health (Ishell)

- 1 In the **Model Builder** window, click **Interface Health (Ishell)**.
- 2 In the **Interface Health (Ishell)** toolbar, click  **Plot**.

Interface Health (Ishell), Stress (Ishell)

- 1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Stress (Ishell)** and **Interface Health (Ishell)**.
- 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 **ID Plot Group**, locate the **Plot Settings** section.
- 3 Select the **x-axis label** checkbox. In the associated text field, type Point Displacement (m).
- 4 Select the **y-axis label** checkbox. In the associated text field, type Point Load (N).

Global I

- 1 In the **Model Builder** window, expand the **Load Displacement Curve** node, then click **Global I**.
- 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** checkbox.
- 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 (3) (sol2)**.
- 4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
2*comp2.F_1pL		Load


- 5 Locate the **x-Axis Data** section. In the **Expression** text field, type `comp2.u_1pL`.
- 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**.

Load (Ishell)


- 1 In the **Results** toolbar, click  **Evaluation Group**.
- 2 In the **Settings** window for **Evaluation Group**, type `Load (Ishell)` in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Study [Layered Shell]/Solution 2 (3) (sol2)**.

Global Evaluation 1




- 1 Right-click **Load (Ishell)** and choose **Global Evaluation**.
- 2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.
- 3 In the table, enter the following settings:

Expression	Unit	Description
2*comp2.F_1pL		Load

- 4 Locate the **Data Series Operation** section. From the **Transformation** list, choose **Maximum**.

5 In the **Load (Ishell)** toolbar, 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 (Ishell)**.
- 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.