Model created in COMSOL Multiphysics 6.4

Progressive Delamination in a Laminated Shell

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 composites using the Layered Shell interface. The capabilities of the CZM to predict mixed-mode softening and delamination propagation are demonstrated in the model.

The example illustrates the delamination initiation and propagation in a composite plate having two layers with an initial delaminated region at the interface. A bending load is gradually applied and removed in a parametric study in order to predict the total interfacial damage in one load cycle.

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

Model Definition

The geometry of a composite plate is shown in Figure 1. The composite plate consists of two layers where each layer has a thickness of 1.5 mm with [0/45] stacking sequence.

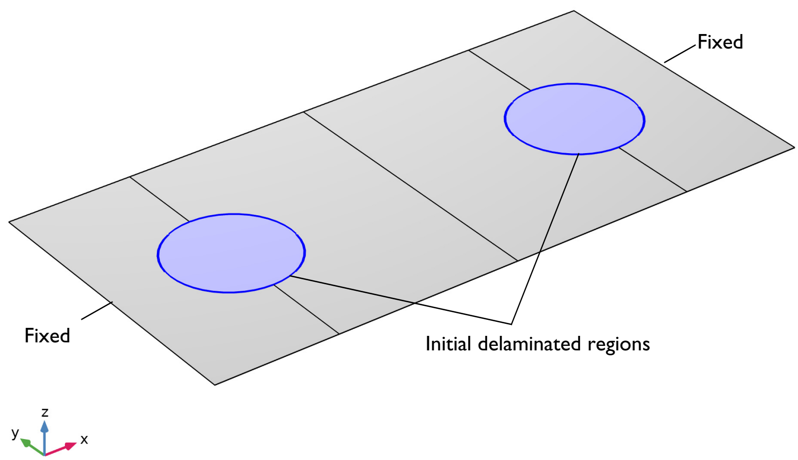

Figure 1: The geometry of a composite plate having two layers with an initial delaminated region at the interface.

The geometry consists of circular regions where the interface between the two layers is in delaminated or debonded state.

MATERIAL PROPERTIES

The material properties are those of AS4/PEEK unidirectional laminates and are listed in Table 1. 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 1: 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	$\nu_{XY}=\nu_{XZ}$	0.25

TABLE 1: LAMINATED COMPOSITE MATERIAL PROPERTIES.

PROPERTY	SYMBOL	VALUE
Poisson's ratio	ν_{YZ}	0.45
Shear modulus	G_{XY}=G_{XZ}	5.5 GPa

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 under different loading. The material properties needed for this constitutive model are summarized in Table 2.

TABLE 2: 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_{Ic}	969 J/m^2
Critical energy release rate, shear	G_{IIc}	1719 J/m^2
Exponent of Benzeggagh and Kenane (B-K) criterion	η	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.

BOUNDARY CONDITIONS

- Face load with a total maximum value of 28 kN is applied at the top surface of the composite plate in negative z-direction. The load is parametrically increased and then decreased to zero using a sinusoidal function.
- Fixed constraints are used on the exterior edges of the plate which are parallel to y-axis.

FINITE ELEMENT MESH

A rather fine mapped mesh is used in the geometry in order to accurately predict the initiation and propagation of delamination in the structure.

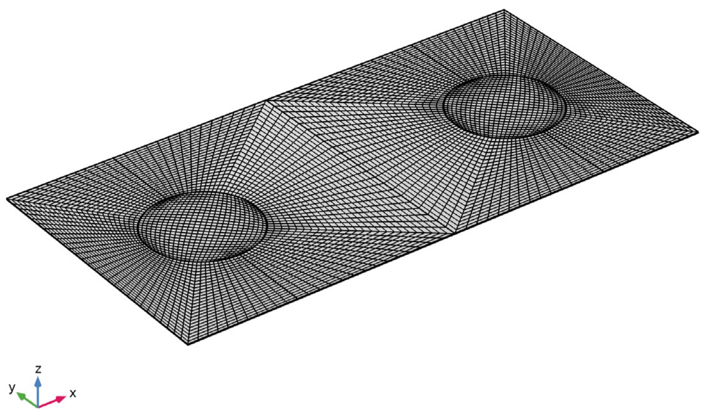

Figure 2: A mapped finite element mesh used to accurately model the delamination propagation in the composite plate.

In the thickness direction, each layer has only one mesh element in order to reduce the overall computation time.

Results and Discussion

The stress distribution in the fiber direction for both layers of the composite plate when the applied load is having maximum value is shown in Figure 3. The corresponding stress distribution at the midplane of two layers is shown in Figure 4. In this figure, the bottom layer undergoes higher stresses than the top layer due to the bending effects and fiber orientation.

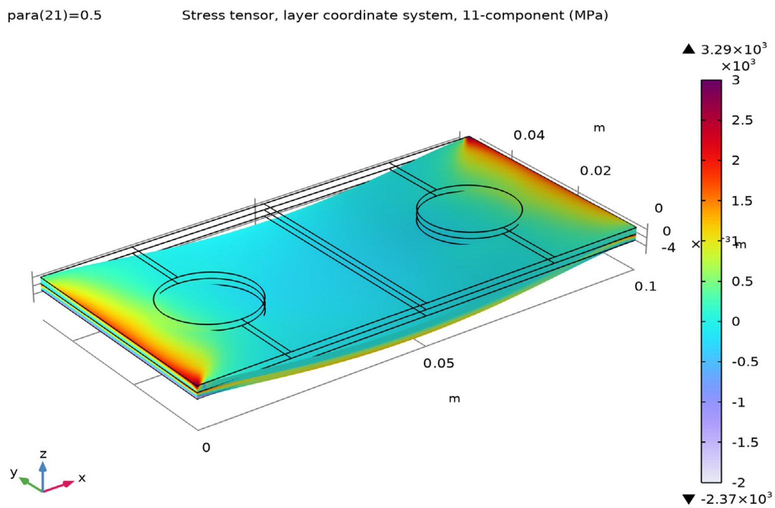

Figure 3: The stress distribution in the composite plate for maximum applied force value.

The state of delaminated region when the applied load is having maximum value is shown in Figure 5, where debonded part is shown in red and bonded part is shown in green color. It can be seen the delamination starts near the comparatively weaker regions or high stress regions. The two such locations in the plate are the boundaries of initially delaminated region and the region near fixed edges.

The adhesive stress in the first tangent direction when the applied load is having maximum value is shown in Figure 6. Figure 7 illustrates the variation of applied load and total damage area as a function of parameter. It can be seen that the interfacial damage is irreversible and it stays permanently in the structure even if the load is removed.

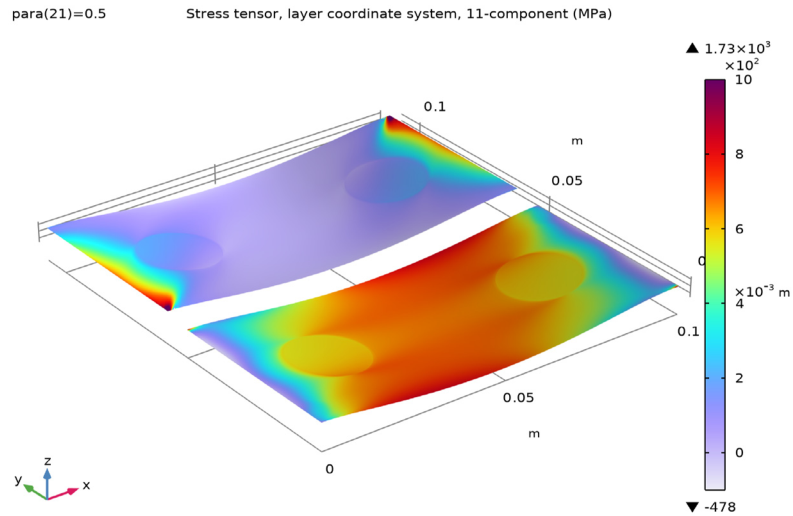

Figure 4: The stress distribution at layer midplanes for maximum applied force value.

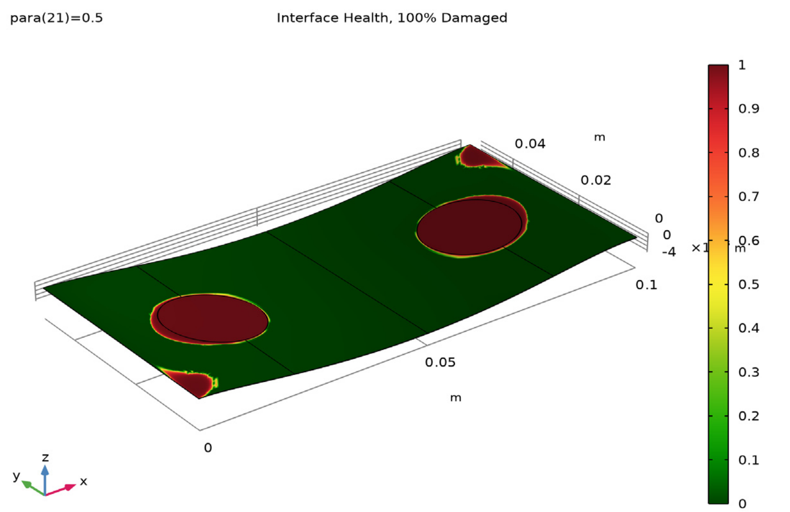

Figure 5: Plot showing the health of the laminate interface for maximum applied force value. The debonded part is shown in red, the intact part in green.

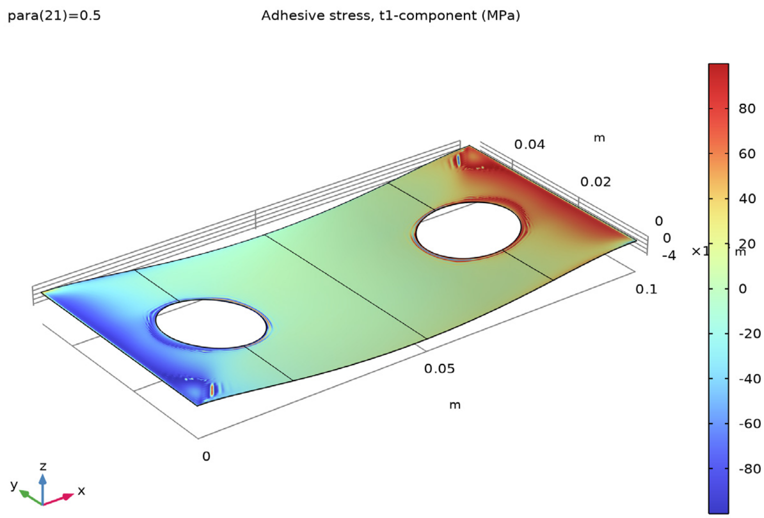

Figure 6: Adhesive stress in first tangent direction for maximum applied force value.

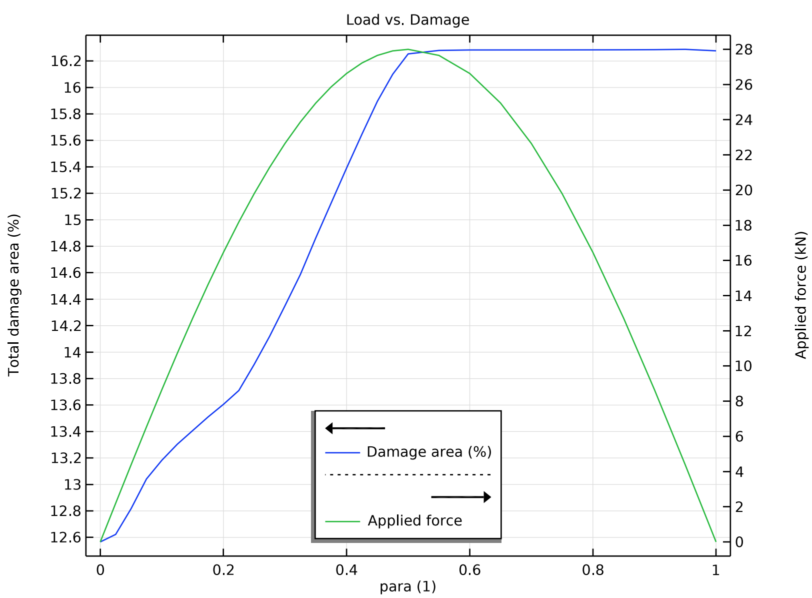

Figure 7: Load vs. damage curve. The overall damage area as well as area where 90% or more damage has occurred are shown.

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 **Layered Shell** interface, use the **Delamination** node which allows you to model adhesion, delamination and contact after delamination. There are two different ways to specify adhesive stiffness with default being taken from the interface material properties. Cohesive zone models are based on either displacement or energy in order to predict the interfacial separation. The contact after delamination is modeled by 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 which is not delaminated set the initial state to *bonded*. The **Delamination** node is only applicable to the internal interfaces of composite laminates.

Application Library path: `Composite_Materials_Module/Delamination/progressive_delamination_in_a_laminated_shell`

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** > **Layered Shell (lshell)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies** > **Stationary**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `progressive_delamination_in_a_laminated_shell_parameters.txt`.

ADD MATERIAL

COMSOL Multiphysics is equipped with built-in material properties for a number of lamina materials. Select the needed materials from the **Composites** material folder in the built-in material library.

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.
2 Go to the **Add Material** window.
3 In the tree, select **Composites** > **Laminae** > **Unidirectional fiber lamina: AS4/APC2 carbon/PEEK thermoplastic [fiber volume fraction 50%]**.
4 Right-click and choose **Add to Global Materials**.
5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

GLOBAL DEFINITIONS

Add a **Layered Material** node and assign appropriate thickness and rotation angles to each ply.

Layered Material: [0/45]

1. In the **Model Builder** window, under **Global Definitions** right-click **Materials** and choose **Layered Material**.
2. In the **Settings** window for **Layered Material**, type `Layered Material: [0/45]` in the **Label** text field.
3. Locate the **Layer Definition** section. 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 rad	hb	1

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

Layer	Material	Rotation	Value	Thickness	Mesh elements
`Layer 2`	Unidirectional fiber lamina: AS4/APC2 carbon/PEEK thermoplastic [fiber volume fraction 50%] (mat1)	45	0.7854 rad	hb	1

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 **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
F	Fmax*sin(pi*para)	N	Applied force

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 **Boundary System** to reference configuration.

Boundary System 1 (sys1)

1 In the **Model Builder** window, click **Boundary System 1 (sys1)**.

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

3 From the **Frame** list, choose **Reference configuration**.

4 Find the **Coordinate names** subsection. From the **Axis** list, choose **x**.

GEOMETRY 1

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 `lb/2`.

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

5 Click **Build Selected**.

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

Work Plane 1 (wp1) > Circle 1 (c1)

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

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

3 In the **Radius** text field, type `lb/10`.

4 Locate the **Position** section. In the **xw** text field, type `lb/5`.

5 In the **yw** 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/5`.

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

5 Click **Build Selected**.

Work Plane 1 (wp1) > Mirror 1 (mir1)

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

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

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

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

5 Locate the **Point on Line of Reflection** section. In the **xw** text field, type `lb/2`.

6 Click **Build Selected**.

Form Union (fin)

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

Ignore Edges 1 (ige1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Virtual Operations** > **Ignore Edges**.

2 On the object **fin**, select Edges 8 and 20 only.

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

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

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

MATERIALS

Layered Material Link 1 (llmat1)

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

LAYERED SHELL (LSHELL)

For the portion of interface which 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 Boundaries 2 and 5 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.

Delamination 2

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

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

2 Select Boundaries 1, 3, 4, and 6 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_{ts} text field, type N_strength.

7 In the σ_{ss} text field, type S_strength.

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

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

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

11 In the α text field, type eta.

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

Fixed Constraint 1

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

2 Select Edges 1 and 23 only.

Face Load 1

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

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

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

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

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

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

0	x
0	y
-F	z

MESH 1

Mapped 1

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

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

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

Distribution 1

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

2 Select Edges 4–6, 8–10, 15–17, and 19–21 only.

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

4 In the **Number of elements** text field, type `25`.

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

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Study Settings** section.

3 Select the **Include geometric nonlinearity** checkbox.

4 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Load parameter)	`range(0,0.025,0.5) range(0.55,0.05,1)`	

7 In the table, click to select the cell at row number 1 and column number 3.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

Switch to an undamped Newton method.

3 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** node, then click **Fully Coupled 1**.

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

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

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

Set default units for result presentation.

RESULTS

Preferred Units 1

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

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

3 Click **Add Physical Quantity**.

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

5 Click **OK**.

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

7 In the table, enter the following settings:

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

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

9 Click **Apply**.

Stress (lshell)

1 In the **Model Builder** window, under **Results** click **Stress (lshell)**.

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

3 From the **Parameter value (para)** list, choose **0.5**.

4 Locate the **Color Legend** section. Select the **Show maximum and minimum values** checkbox.

Surface 1

1 In the **Model Builder** window, expand the **Stress (lshell)** node, then click **Surface 1**.
2 In the **Settings** window for **Surface**, click to expand the **Range** section.
3 Select the **Manual color range** checkbox.
4 In the **Minimum** text field, type `-2e3`.
5 In the **Maximum** text field, type `3e3`.

Stress (lshell)

In the **Model Builder** window, collapse the **Results** > **Stress (lshell)** node.

ROOT

In the **Model Builder** window, right-click the root node and choose **Plot**.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.
2 Go to the **Result Templates** window.
3 In the tree, select **Study 1/Solution 1 (sol1)** > **Layered Shell** > **Stress, Slice (lshell)**.
4 Click the **Add Result Template** button in the window toolbar.
5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Stress, Slice (lshell)

1 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
2 From the **Parameter value (para)** list, choose **0.5**.
3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.
4 Locate the **Color Legend** section. Select the **Show maximum and minimum values** checkbox.

Layered Material Slice 1

1 In the **Model Builder** window, expand the **Stress, Slice (lshell)** node, then click **Layered Material Slice 1**.
2 In the **Settings** window for **Layered Material Slice**, locate the **Through-Thickness Location** section.
3 From the **Location definition** list, choose **Layer midplanes**.
4 Locate the **Layout** section. From the **Displacement** list, choose **Linear**.
5 From the **Orientation** list, choose **y**.

6 Click to expand the **Range** section. Select the **Manual color range** checkbox.

7 In the **Minimum** text field, type `-1e2`.

8 In the **Maximum** text field, type `1e3`.

Stress, Slice (lshell)

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

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

3 From the **View** list, choose **New view**.

4 In the **Stress, Slice (lshell)** toolbar, click **Plot**.

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

Interface Health, 100% Damaged

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

2 In the **Settings** window for **3D Plot Group**, type `Interface Health, 100% Damaged` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (para)** list, choose **0.5**.

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

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

Layered Material Slice 1

1 In the **Interface Health, 100% Damaged** 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**.

Deformation 1

1 Right-click **Layered Material Slice 1** and choose **Deformation**.

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

3 Select the **Scale factor** checkbox. In the associated text field, type `1`.

4 In the **Interface Health, 100% Damaged** toolbar, click **Plot**.

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

Adhesive Stress, t1 Direction

1 In the **Results** toolbar, click **3D Plot Group**.
2 In the **Settings** window for **3D Plot Group**, type `Adhesive Stress, t1 Direction` in the **Label** text field.
3 Locate the **Data** section. From the **Parameter value (para)** list, choose **0.5**.
4 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

Layered Material Slice 1

1 In the **Adhesive Stress, t1 Direction** toolbar, click **More Plots** and choose **Layered Material Slice**.
2 In the **Settings** window for **Layered Material Slice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Layered Shell** > **Delamination** > **Adhesive stress (spatial frame) - N/m²** > **lshell.fst1 - Adhesive stress, t1-component**.
3 Locate the **Through-Thickness Location** section. From the **Location definition** list, choose **Interfaces**.
4 Locate the **Coloring and Style** section. From the **Color table** list, choose **RainbowLight**.
5 From the **Scale** list, choose **Linear symmetric**.

Deformation 1

1 Right-click **Layered Material Slice 1** and choose **Deformation**.
2 In the **Settings** window for **Deformation**, locate the **Scale** section.
3 Select the **Scale factor** checkbox. In the associated text field, type `1`.

Adhesive Stress, t1 Direction

1 In the **Model Builder** window, under **Results** click **Adhesive Stress, t1 Direction**.
2 In the **Adhesive Stress, t1 Direction** toolbar, click **Plot**.

Layered Material (Interfaces)

1 In the **Results** toolbar, click **More Datasets** and choose **Layered Material**.
2 In the **Settings** window for **Layered Material**, type `Layered Material (Interfaces)` in the **Label** text field.
3 Locate the **Layers** section. From the **Evaluate in** list, choose **Interfaces**.

Damaged Area

1 In the **Results** toolbar, click **Evaluation Group**.
2 In the **Settings** window for **Evaluation Group**, type `Damaged Area` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Layered Material (Interfaces)**.

Surface Integration 1

1. Right-click **Damaged Area** and choose **Integration** > **Surface Integration**.
2. In the **Settings** window for **Surface Integration**, locate the **Selection** section.
3. From the **Selection** list, choose **All boundaries**.
4. Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`gpeval(4,lshell.idmg)/(lb*wb)`	`%`	`Damage area`

5. In the **Damaged Area** toolbar, click **Evaluate**.

Load vs. Damage

1. In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Load vs. Damage` in the **Label** text field.
3. Click to expand the **Title** section. From the **Title type** list, choose **Label**.
4. Locate the **Plot Settings** section.
5. Select the **x-axis label** checkbox. In the associated text field, type `para (1)`.
6. Select the **y-axis label** checkbox. In the associated text field, type `Total damage area (%)`.
7. Select the **Two y-axes** checkbox.
8. Click to collapse the **Axis** section. Locate the **Legend** section. From the **Position** list, choose **Lower middle**.

Table Graph 1

1. Right-click **Load vs. Damage** and choose **Table Graph**.
2. In the **Settings** window for **Table Graph**, locate the **Data** section.
3. From the **Source** list, choose **Evaluation group**.
4. From the **Evaluation group** list, choose **Damaged Area**.
5. Click to expand the **Legends** section. Select the **Show legends** checkbox.

Global 1

1. In the **Model Builder** window, right-click **Load vs. Damage** and choose **Global**.
2. In the **Settings** window for **Global**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.
4. Locate the **y-Axis** section. Select the **Plot on secondary y-axis** checkbox.

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

Expression	Unit	Description
F	kN	Applied force

Load vs. Damage

1 In the **Model Builder** window, collapse the **Results** > **Load vs. Damage** node.

2 In the **Model Builder** window, click **Load vs. Damage**.

3 In the **Load vs. Damage** toolbar, click **Plot**.

Animation: Stress

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

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

3 Locate the **Frames** section. From the **Frame selection** list, choose **All**.

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

Animation: Interface Health

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

2 In the **Settings** window for **Animation**, type `Animation: Interface Health` in the **Label** text field.

3 Locate the **Scene** section. From the **Subject** list, choose **Interface Health, 100% Damaged**.

Animation: Adhesive Stress

1 Right-click **Animation: Interface Health** and choose **Duplicate**.

2 In the **Settings** window for **Animation**, type `Animation: Adhesive Stress` in the **Label** text field.

3 Locate the **Scene** section. From the **Subject** list, choose **Adhesive Stress, t1 Direction**.