Created in COMSOL Multiphysics 6.1

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 compressive load is gradually applied and removed in a parametric study in order to predict the total interfacial damage in one load cycle.

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.

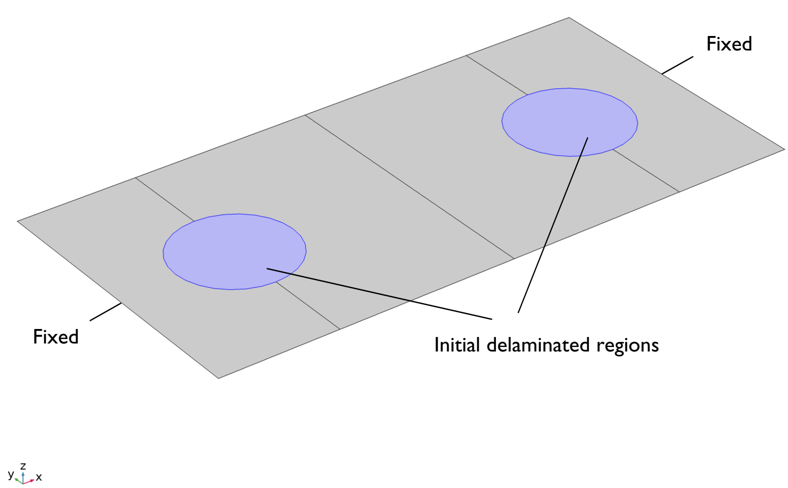

Figure 1: The geometry of a composite plate having two layers with an initial delaminated region at the interface.

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. 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 1.

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	ν_{YZ}	0.45
Poisson's ratio	ν_{XY}=ν_{XZ}	0.25
Shear modulus	G_{YZ}	3.7 GPa
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.

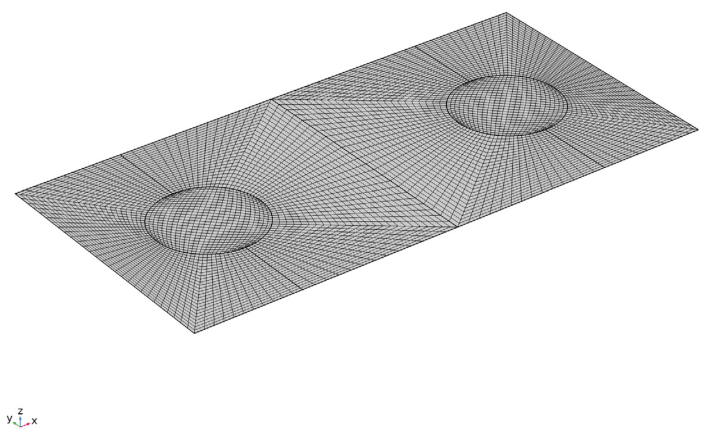

Figure 2: A mapped finite element mesh used to accurately model the delamination propagation in the composite plate.

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

Results and Discussion

The von Mises stress distribution in both layers of the composite plate when the applied load is having maximum value is shown in Figure 3. The corresponding von Mises 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.

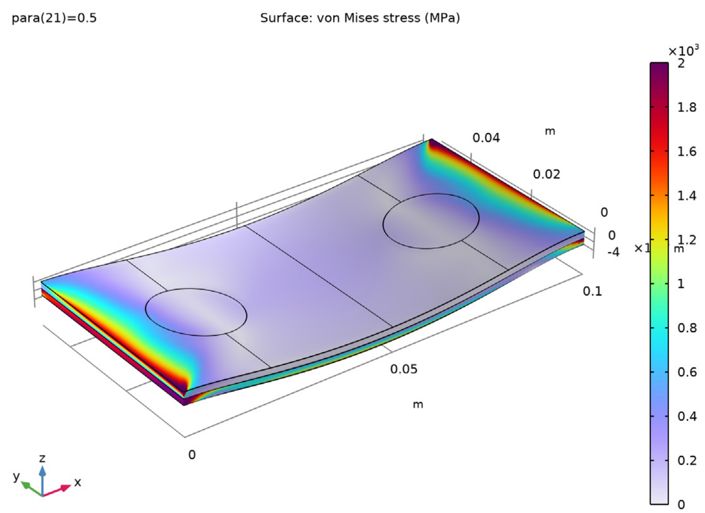

Figure 3: von Mises stress distribution in the composite plate for maximum applied force value.

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. In the plot there are two damage area indices plotted; one is the overall damage area index and the other is the damage area index where the interface has broken by 90% or more.

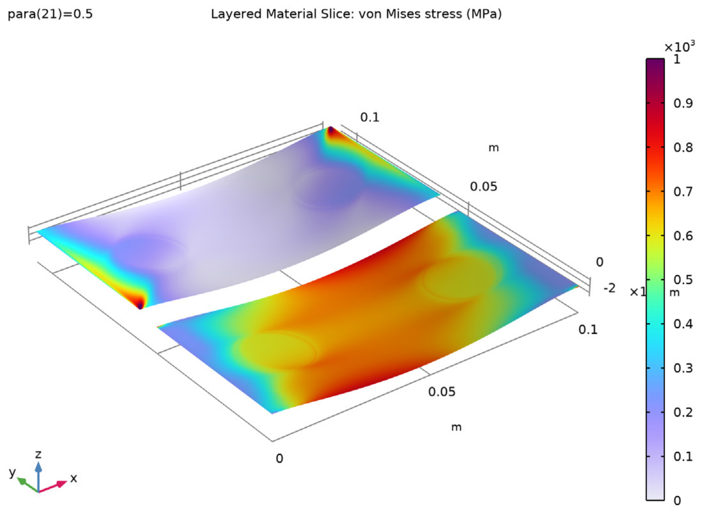

Figure 4: von Mises stress distribution at layer midplanes for maximum applied force value.

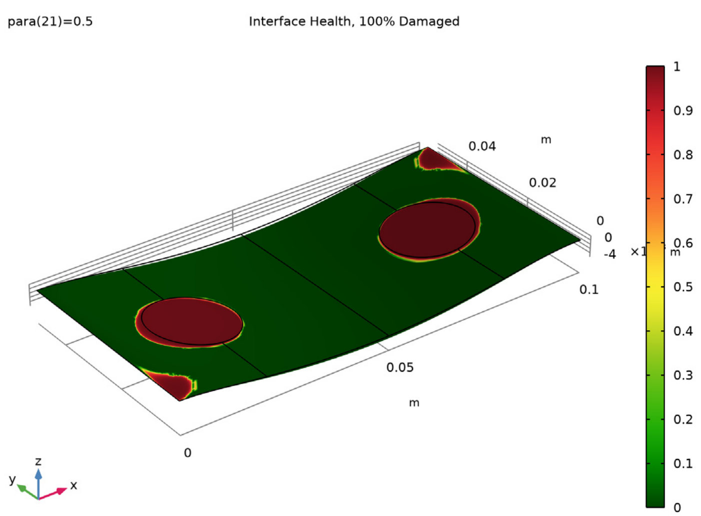

Figure 5: Plot showing the health of the laminate interface for maximum applied force value. The debonded part is shown in red, the intact part in green.

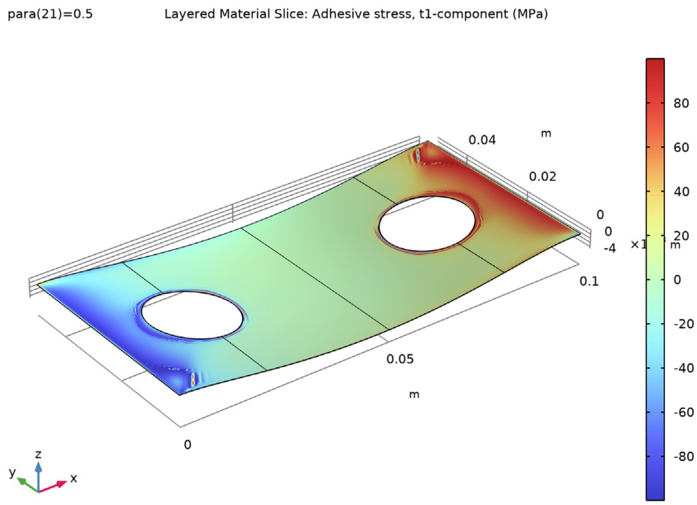

Figure 6: Adhesive stress in first tangent direction for maximum applied force value.

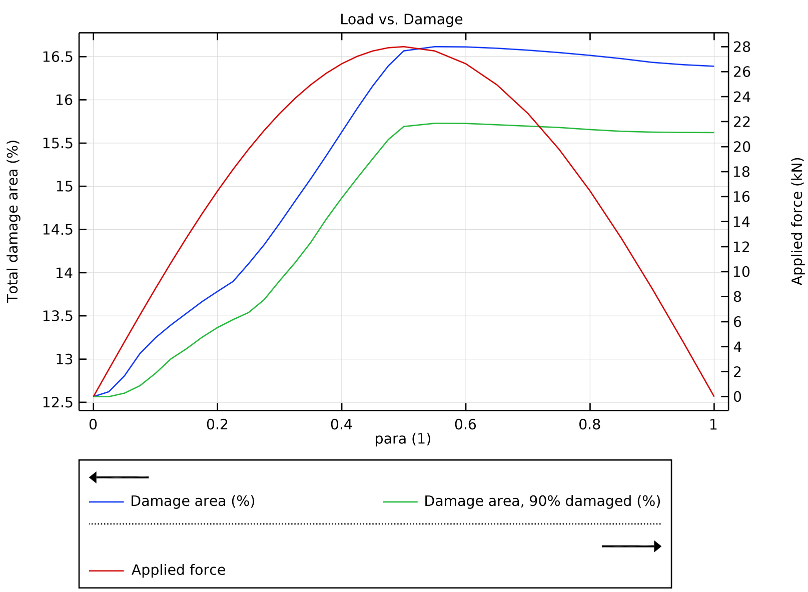

Figure 7: Load vs. damage curve. The overall damage area as well as area where 90% or more damage has occurred are shown.

Notes About the COMSOL Implementation

- 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.
- Modeling a composite laminated shell requires a surface geometry (2D), in general called a base surface, and a **Layered Material** node which adds an extra dimension (1D) to the base surface geometry in the surface normal direction. You can use the **Layered Material** functionality to model several layers stacked on top of each other having different thicknesses, material properties, and fiber orientations. You can also optionally specify the interface materials between the layers and control mesh elements in each layer.

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`.

AS4/PEEK

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

2 In the **Settings** window for **Material**, type `AS4/PEEK` in the **Label** text field.

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

Layered Material: [0/45]

1 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 (deg)	Thickness	Mesh elements
Layer 1	AS4/PEEK (mat1)	0	hb	1

4 Click **Add**.

5 In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness	Mesh elements
Layer 2	AS4/PEEK (mat1)	45	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)

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/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** check box.

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**.

MATERIALS

Layered Material Link 1 (llmat1)

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

GLOBAL DEFINITIONS

AS4/PEEK (mat1)

1 In the **Settings** window for **Material**, locate the **Material Contents** section.

2 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	{Evector1, Evector2, Evector3}	`{122.7e9, 10.1e9, 10.1e9}`	Pa	Orthotropic
Poisson's ratio	{nuvector1, nuvector2, nuvector3}	`{0.25, 0.45, 0.25}`	1	Orthotropic
Shear modulus	{Gvector1, Gvector2, Gvector3}	`{5.5e9, 3.7e9, 5.5e9}`	N/m²	Orthotropic
Density	rho	`1570`	kg/m³	Basic

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 **Model Builder** window, under **Component 1 (comp1)** right-click **Layered Shell (lshell)** and choose **Material Models>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_t text field, type `N_strength`.

7 In the σ_s 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 **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 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** check box.
4. Click to expand 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
para (Load parameter)	`range(0,0.025,0.5) range(0.55,0.05,1)`	

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**.

RESULTS

Stress (lshell)

1. In the **Settings** window for **3D Plot Group**, locate the **Data** section.
2. From the **Parameter value (para)** list, choose **0.5**.

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 to expand the **Range** section. Select the **Manual color range** check box.

5 In the **Minimum** text field, type `0`.

6 In the **Maximum** text field, type `2e3`.

Stress (lshell)

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

ROOT

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

2 In the **Home** toolbar, click **Add Predefined Plot**.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

2 In the tree, select **Study 1/Solution 1 (sol1)>Layered Shell>Stress, Slice (lshell)**.

3 Click **Add Plot** in the window toolbar.

4 In the **Home** toolbar, click **Add Predefined Plot**.

RESULTS

Stress, Slice (lshell)

1 In the **Model Builder** window, under **Results** click **Stress, Slice (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 **Plot Settings** section. Clear the **Plot dataset edges** check box.

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 Locate the **Expression** section. From the **Unit** list, choose **MPa**.

7 Click to expand the **Range** section. Select the **Manual color range** check box.

8 In the **Minimum** text field, type `0`.

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

Interface Health, 100% Damaged

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, 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. Click **Change Color Table**.

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

7 Click **OK**.

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** check box. In the associated text field, type `1`.

Interface Health, 100% Damaged

In the **Model Builder** window, collapse the **Results>Interface Health, 100% Damaged** node.

ROOT

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

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

RESULTS

Interface Health, 90% Damaged

1 In the **Model Builder** window, right-click **Interface Health, 100% Damaged** and choose **Duplicate**.
2 In the **Settings** window for **3D Plot Group**, type `Interface Health, 90% Damaged` in the **Label** text field.

Layered Material Slice 1

1 In the **Model Builder** window, expand the **Interface Health, 90% Damaged** node, then click **Layered Material Slice 1**.
2 In the **Settings** window for **Layered Material Slice**, locate the **Expression** section.
3 In the **Expression** text field, type `lshell.idmg>=0.9`.

Interface Health, 90% Damaged

1 In the **Model Builder** window, collapse the **Results>Interface Health, 90% Damaged** node.
2 In the **Model Builder** window, click **Interface Health, 90% Damaged**.
3 In the **Interface Health, 90% Damaged** toolbar, click **Plot**.

Adhesive Stress, t1 Direction

1 In the **Home** toolbar, click **Add Plot Group** and choose **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 **Expression** section. From the **Unit** list, choose **MPa**.
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 **Rainbow>RainbowLight** in the tree.

7 Click **OK**.

8 In the **Settings** window for **Layered Material Slice**, locate the **Coloring and Style** section.

9 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** check box. In the associated text field, type `1`.

Adhesive Stress, t1 Direction

1 In the **Model Builder** window, collapse the **Results>Adhesive Stress, t1 Direction** node.

2 In the **Model Builder** window, click **Adhesive Stress, t1 Direction**.

3 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**.

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

Surface Integration 1

1 In the **Results** toolbar, click **More Derived Values** and choose **Integration>Surface Integration**.

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

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

4 Locate the **Selection** section. From the **Selection** list, choose **All boundaries**.

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

Expression	Unit	Description
`lshell.idmg/(lb*wb)`	`%`	`Damage area`
`(lshell.idmg>=0.9)/(lb*wb)`	`%`	`Damage area, 90% damaged`

6 Click **Evaluate**.

Load vs. Damage

1 In the **Results** toolbar, click **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** check box. In the associated text field, type `para (1)`.

6 Select the **y-axis label** check box. In the associated text field, type `Total damage area (%)`.

7 Select the **Two y-axes** check box.

8 Click to collapse the **Axis** section. Locate the **Legend** section. From the **Layout** list, choose **Outside graph axis area**.

9 From the **Position** list, choose **Bottom**.

Table Graph 1

1 Right-click **Load vs. Damage** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, click to expand the **Legends** section.

3 Select the **Show legends** check box.

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** check box.

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**.