Model created in COMSOL Multiphysics 6.4

Mixed-Mode Debonding of a Laminated Composite

Introduction

Interfacial failure by delamination or debonding is one of the main failure modes of laminate structures. Such failures 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 how to model debonding using the decohesion model in COMSOL Multiphysics. The capabilities of the CZM to predict mixed-mode softening and delamination propagation are demonstrated in a model of the mixed-mode bending of a composite beam.

Model Definition

COHESIVE ZONE MODEL (CZM)

The CZM is in this example defined using the displacement-based damage model available in the **Decohesion** node under **Contact**. 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 the 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.

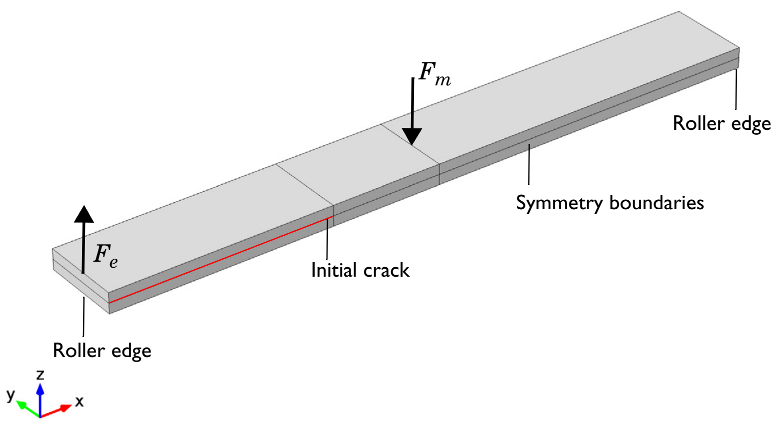

Figure 1: The geometry of the test specimen.

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	ν_{XY}=ν_{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 $\boldsymbol{F}_e$ 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. At this step the initial crack has propagated along the interface as shown in Figure 3. The crack now extends beyond the center of the beam.

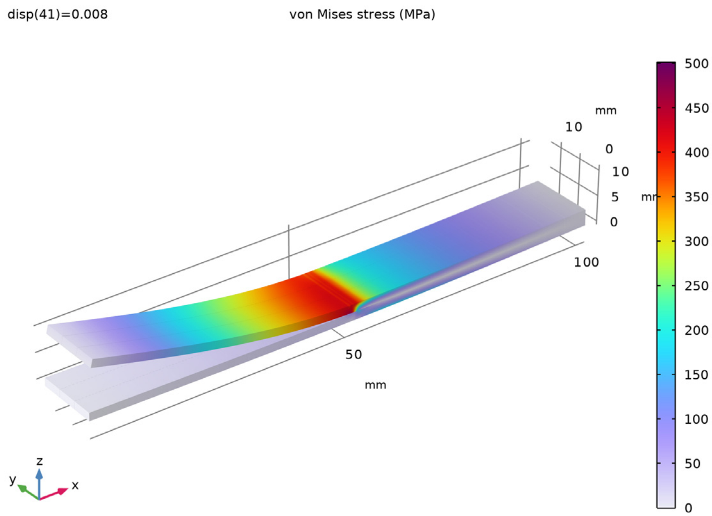

Figure 2: The von Mises stress distribution at the last computation step.

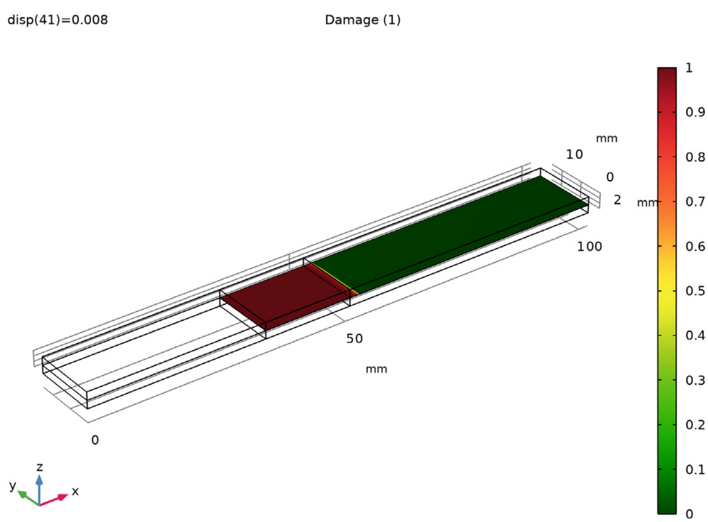

Figure 3: Plot showing the health of the laminate interface. The debonded part is shown in red, the intact part in green.

If, however, you zoom to the crack front region in the plot showing the interface health, you will notice that the color surfaces do not look perfectly regular. The reason is that the damage function has sharp or even discontinuous gradients within some elements. If you want examine the damage state in detail, then it is more relevant to look at the results in the individual integration points, since it is those values that are actually used during the solution. In Figure 4, such a plot is shown. Gauss point plots are suited for examining local details in the solution, but not for providing a general overview.

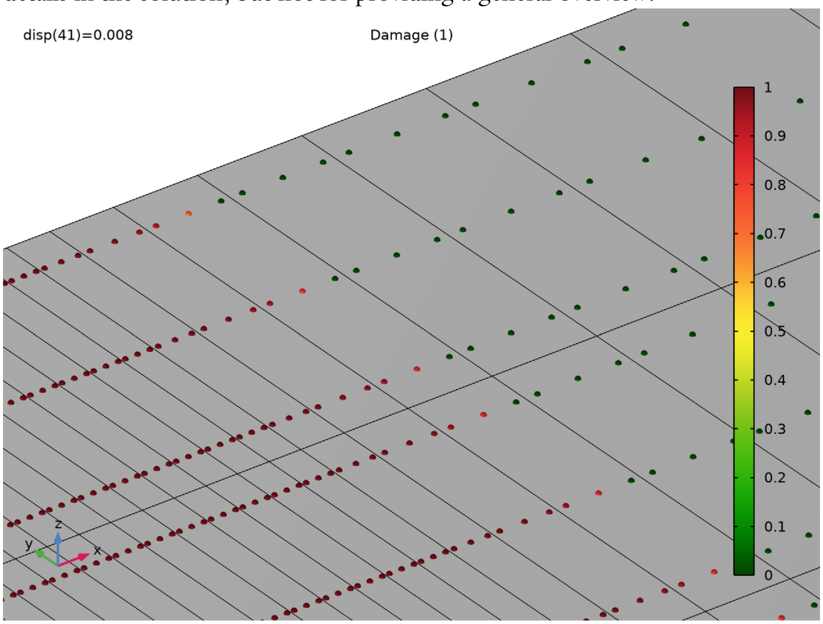

Figure 4: Interface health visualized in the integration points.

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}:

$$F_{\mathrm{lp}} = F_{\mathrm{e}}\frac{l_{\mathrm{b}}/2}{l_{\mathrm{l}}}.$$

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

$$l_1 = \frac{(l_b/2)\left(\frac{1}{2}\sqrt{3\frac{1-m_m}{m_m}}+1\right)}{3-\frac{1}{2}\sqrt{3\frac{1-m_m}{m_m}}}.$$

l_1 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

$$u_{lp} = \left(\frac{3l_1 - l_b/2}{4l_b/2}\right)u_{Ie} + \left(\frac{l_1 + l_b/2}{l_b/2}\right)(-w_c + u_{Ie}/4).$$

The resulting load-displacement curve is shown in Figure 5. The curve confirms what Figure 3 displayed. The maximum load that the beam with the initial crack can carry is exceeded and delamination occurs. After exceeding a 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.

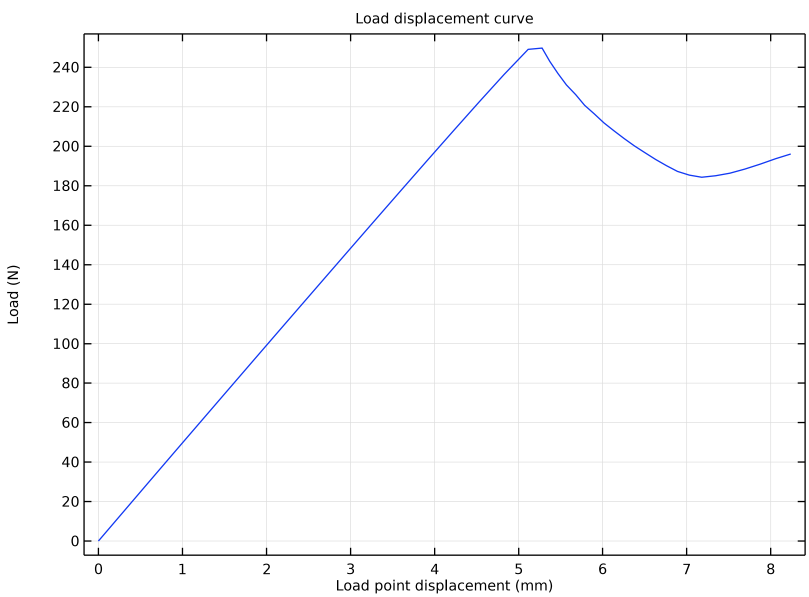

Figure 5: Load-displacement curve of the MMB test at 50% mixed-mode loading.

Notes About the COMSOL Implementation

Since no large relative deformations of the laminates are expected, the **Interior Contact** node is utilized to model the behavior between laminates. This is an alternative way to model contact without requiring an assembly.

To implement a cohesive zone model in COMSOL Multiphysics, add an **Adhesion** subnode with a **Decohesion** subnode to the **Interior Contact** node.

To trace the solution after the peak load, the simulation must be displacement controlled. This is obtained by using a global equation, in which the distributed load is controlled by the sweep parameter, which is the displacement at the free edge.

To accurately resolve the decohesion at the interface, a dense mesh is used in the zone where delamination is expected. If the mesh elements were too large, the crack could jump several elements in one parameter step, resulting in an unstable solution.

Reference

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* 37.16 (2003): 1415–1438.

2. J.R. Reeder, and J.R. Crews Jr., "Mixed-mode bending method for delamination testing," *AiAA Journal* 28.7 (1990): 1270–1276.

Application Library path: `Structural_Mechanics_Module/Contact_and_Friction/cohesive_zone_debonding`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Structural Mechanics** > **Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies** > **Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Load all model parameters from a file containing parameters for the geometry, material properties and boundary conditions.

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 `cohesive_zone_debonding_parameters.txt`.

GEOMETRY 1

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

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

3 From the **Length unit** list, choose **mm**.

Now draw the model geometry with two layered blocks that are overlapping.

Block 1 (blk1)

1 In the **Geometry** toolbar, click **Block**.

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

3 In the **Width** text field, type `lb`.

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

5 In the **Height** text field, type `hb`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	hb/2

Block 2 (blk2)

1 In the **Geometry** toolbar, click **Block**.

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

3 In the **Width** text field, type `lb/2-cl`.

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

5 In the **Height** text field, type `hb`.

6 Locate the **Position** section. In the **x** text field, type `cl`.

7 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	hb/2

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

DEFINITIONS

Boundary System 1 (sys1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** node, then 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**.

Load Point Variables

1 In the **Definitions** toolbar, click **Local Variables**.

Load variables for the load point from files.

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

3 Locate the **Variables** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `cohesive_zone_debonding_load_point_variables.txt`.

Integration Edge

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

The following nonlocal integration couplings make values of the selected points globally available.

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 3 only.

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

Integration Center

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

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

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

4 Select Point 15 only.

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

GLOBAL DEFINITIONS

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.

ADD MATERIAL

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.

MATERIALS

Material Link 1 (matlnk1)

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

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

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

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

Add the CZM using a **Interior Contact** node.

Interior Contact 1

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

2 Select Boundaries 15 and 24 only.

3 In the **Settings** window for **Interior Contact**, locate the **Contact Pressure Penalty Factor** section.

4 From the **Penalty factor control** list, choose **User defined**.

5 In the p_{n} text field, type `pn`.

Adhesion 1

1 In the **Physics** toolbar, click **Attributes** and choose **Adhesion**.

2 In the **Settings** window for **Adhesion**, locate the **Adhesive Activation** section.

3 From the **Activation criterion** list, choose **Always active**.

4 Locate the **Adhesive Stiffness** section. In the n_{τ} text field, type `1`.

Decohesion 1

1 In the **Physics** toolbar, click **Attributes** and choose **Decohesion**.

2 In the **Settings** window for **Decohesion**, locate the **Decohesion** section.

3 In the σ_{ts} text field, type `sigmat`.

4 In the σ_{ss} text field, type `sigmas`.

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

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

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

8 In the α text field, type `alpha`.

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

10 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics** > **Equation Contributions**.

11 Click **OK**.

This is to make **Auxiliary Slit** and **Global Equations** accessible.

Auxiliary Slit 1

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

2 Select Boundary 6 only.

Symmetry 1

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

2 Select Boundaries 2, 5, 11, 14, 20, and 23 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 Select Edge 7 only.

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

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

0	x
0	y
force	z

Load on Middle Edge (Fm)

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

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

3 Select Edge 33 only.

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

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

0	x
0	y
-lr*force	z

Prescribed Displacement 1

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

2 Select Edges 2 and 41 only.

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

4 From the **Displacement in z direction** list, choose **Prescribed**.

Prescribed Displacement 2

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

2 Select Point 1 only.

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

4 From the **Displacement in x direction** list, choose **Prescribed**.

Add a global equation to control the applied load with a monotonically increasing parameter.

Global Equations 1 (ODE1)

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 (ut_0) (1/s)	Description
force	disp-u_Ie	0	0	

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.
5 In the **Physical Quantity** dialog, type `force` in the text field.
6 In the tree, select **General** > **Force (N)**.
7 Click **OK**.
8 In the **Settings** window for **Global Equations**, locate the **Units** section.
9 Click **Select Source Term Quantity**.
10 In the **Physical Quantity** dialog, type `Displacement` in the text field.
11 In the tree, select **General** > **Displacement (m)**.
12 Click **OK**.

MESH 1

Mapped 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.
2 Select Boundaries 2, 5, 11, 14, 20, and 23 only.

Distribution 1

1 In the **Mesh** toolbar, click **Distribution**.
2 Select Edges 3, 6, and 8 only.
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
4 In the **Number of elements** text field, type `10`.

Distribution 2

1 In the **Mesh** toolbar, click **Distribution**.
2 Select Edges 16, 19, and 21 only.

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

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

Distribution 3

1 In the **Mesh** toolbar, click **Distribution**.

2 Select Edges 29, 32, and 34 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 `10`.

Swept 1

In the **Mesh** toolbar, click **Swept**.

Distribution 1

1 In the **Mesh** toolbar, click **Distribution**.

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

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

4 Click **Build All**.

STUDY 1

Step 1: Stationary

Configure the solver to enable tracking of the post peak behavior of the beam.

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 From the **Tolerance** list, choose **User controlled**.

5 In the **Relative tolerance** text field, type `1e-4`.

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

7 Click **Add**.

8 In the table, enter the following settings:

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

9 Click to expand the **Results While Solving** section. Select the **Plot** checkbox.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.
2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.
3 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Dependent Variables 1** node, then click **Global Equations 1 (comp1.ODE1)**.
4 In the **Settings** window for **State**, locate the **Scaling** section.
5 From the **Method** list, choose **Manual**.
6 In the **Scale** text field, type 200.

Use a linear predictor.

7 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** node, then click **Parametric 1**.
8 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.
9 Select the **Tuning of step size** checkbox.
10 In the **Minimum step size** text field, type 1e-6.
11 From the **Predictor** list, choose **Linear**.

Switch to an undamped Newton method.

12 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** click **Fully Coupled 1**.
13 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.
14 From the **Nonlinear method** list, choose **Constant (Newton)**.
15 In the **Study** toolbar, click **Get Initial Value**.

RESULTS

Volume 1

1 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Volume 1**.
2 In the **Settings** window for **Volume**, locate the **Expression** section.
3 From the **Unit** list, choose **MPa**.

The Internal Contact and Auxiliary Slit conditions will enforce different displacements on the top and the bottom sides of the same boundary. This will cause some plots with deformation to look strange. To remedy this, do the following:

4 Click to expand the **Shrink Elements** section. In the **Element scale factor** text field, type `0.999`.

Stress (solid)

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

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

3 Clear the **Plot dataset edges** checkbox.

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

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

Now, add a plot showing the state of debonding at the laminate interface on the undeformed geometry.

Interface Health

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` in the **Label** text field.

Surface 1

1 Right-click **Interface Health** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Contact** > **Adhesion** > **solid.bdmg - Damage - 1**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **Traffic**.

This choice plots the debonded part in red while the healthy part remains green.

4 In the **Interface Health** toolbar, click **Plot**.

Since the changes in damage level are rather abrupt, a surface plot can be difficult to interpret. The actual evaluation of damage states during solution is done at the integration point level. Add a plot which displays the values in each Gauss point.

Interface Health: Gauss Points

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

2 In the **Settings** window for **3D Plot Group**, type `Interface Health: Gauss Points` in the **Label** text field.

Point 1

1 In the **Interface Health: Gauss Points** toolbar, click **More Plots** and choose **Point**.

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

3 In the **Expression** text field, type `solid.bdmg`.

4 Locate the **Evaluation** section. From the **Geometry level** list, choose **Surface**.

5 From the **Placement** list, choose **Gauss points**.

6 In the **Gauss-point order** text field, type `4`.

7 Locate the **Coloring and Style** section. From the **Color table** list, choose **Traffic**.

8 In the **Interface Health: Gauss Points** toolbar, click **Plot**.

9 Select the **Radius scale factor** checkbox. In the associated text field, type `0.03`.

10 In the **Interface Health: Gauss Points** toolbar, click **Plot**.

Interface Health: Gauss Points

1 In the **Model Builder** window, click **Interface Health: Gauss Points**.

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

3 Clear the **Plot dataset edges** checkbox.

Mesh 1

1 Right-click **Interface Health: Gauss Points** and choose **Mesh**.

2 In the **Settings** window for **Mesh**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

Selection 1

1 Right-click **Mesh 1** and choose **Selection**.

2 Select Boundaries 15 and 24 only.

Mesh 1

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

2 In the **Settings** window for **Mesh**, locate the **Coloring and Style** section.

3 From the **Element color** list, choose **Gray**.

4 In the **Interface Health: Gauss Points** toolbar, click **Plot**.

Load Displacement Curve

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 Displacement Curve` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Load displacement curve`.

Global 1

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

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

3 In the table, enter the following settings:

Expression	Unit	Description
2*F_lp	N	Load

The factor 2 is needed to compensate for the model symmetry.

4 Click to expand the **Legends** section. Clear the **Show legends** checkbox.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type u_lp.

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

Finally, evaluate the maximal load that this beam can carry under this loading condition.

Load (solid)

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type Load (solid) in the **Label** text field.

Global Evaluation 1

1 In the **Load (solid)** toolbar, click **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*F_lp	N	Load

Again, the factor 2 is due to model symmetry.

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

5 In the **Load (solid)** toolbar, click **Evaluate**.