Mixed-Mode Debonding of a Laminated Composite

This model is licensed under the COMSOL Software License Agreement 5.6.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

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, i.e. 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.

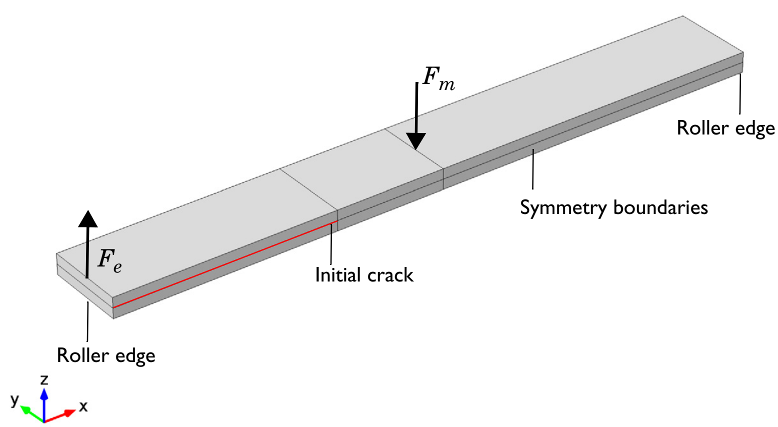

Figure 1: The geometry of the test specimen.

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

TABLE 2: LAMINATED COMPOSITE MATERIAL PROPERTIES.

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

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

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

$$l_\mathrm{r} = 8\left(\frac{6m_\mathrm{m} + \sqrt{3m_\mathrm{m}(1-m_\mathrm{m})}}{3 + 9m_\mathrm{m} + 8\sqrt{3m_\mathrm{m}(1-m_\mathrm{m})}}\right).$$

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.

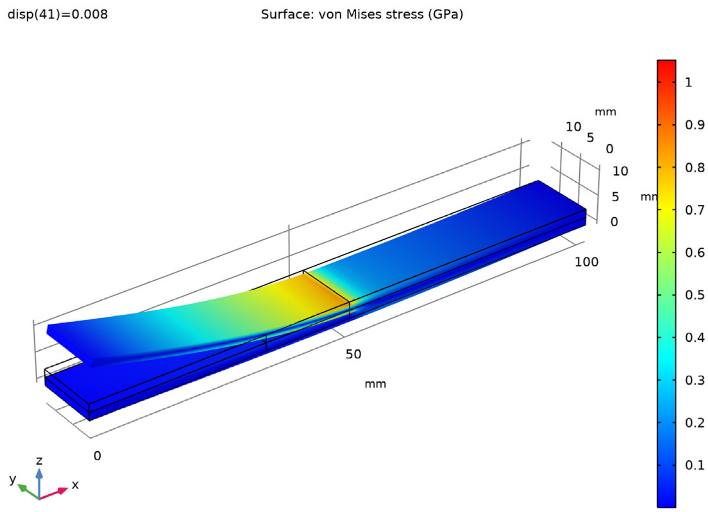

Figure 2: The Von Mises stress distribution at the last computation step.

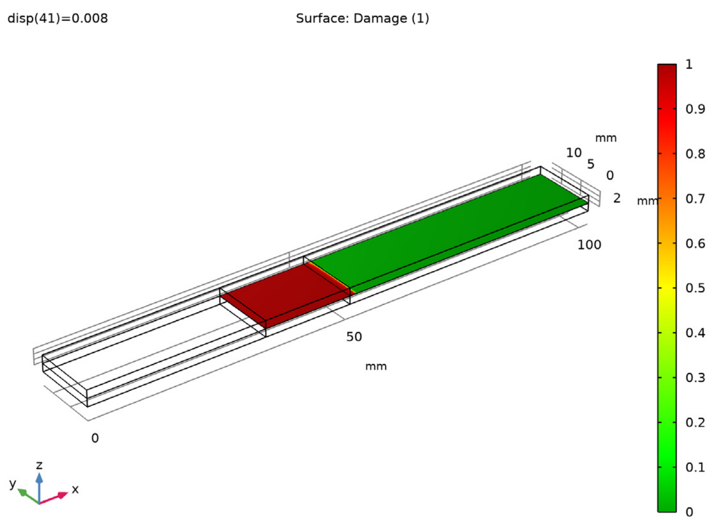

Figure 3: Plot showing the health of the laminate interface. The debonded part is shown in red, the intact part in green.

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

The force $\boldsymbol{F}_{\mathrm{lp}}$ at the load point of the lever can be determined from the load applied to the cracked edge in the model $\boldsymbol{F}_{\mathrm{e}}$ and the lengths of the test beam l_{b} and load lever l_{l}:

$$\boldsymbol{F}_{\mathrm{lp}} = \boldsymbol{F}_{\mathrm{e}}\frac{l_{\mathrm{b}}/2}{l_{\mathrm{l}}}.$$

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

$$l_{\mathrm{l}} = \frac{(l_{\mathrm{b}}/2)\left(\frac{1}{2}\sqrt{3\frac{1-m_{\mathrm{m}}}{m_{\mathrm{m}}}}+1\right)}{3-\frac{1}{2}\sqrt{3\frac{1-m_{\mathrm{m}}}{m_{\mathrm{m}}}}}.$$

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

$$u_{\mathrm{lp}} = \left(\frac{3l_{\mathrm{l}}-l_{\mathrm{b}}/2}{4l_{\mathrm{b}}/2}\right)u_{\mathrm{Ie}}+\left(\frac{l_{\mathrm{l}}+l_{\mathrm{b}}/2}{l_{\mathrm{b}}/2}\right)(-w_{\mathrm{c}}+u_{\mathrm{Ie}}/4).$$

The resulting load-displacement curve is shown in Figure 4. 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.

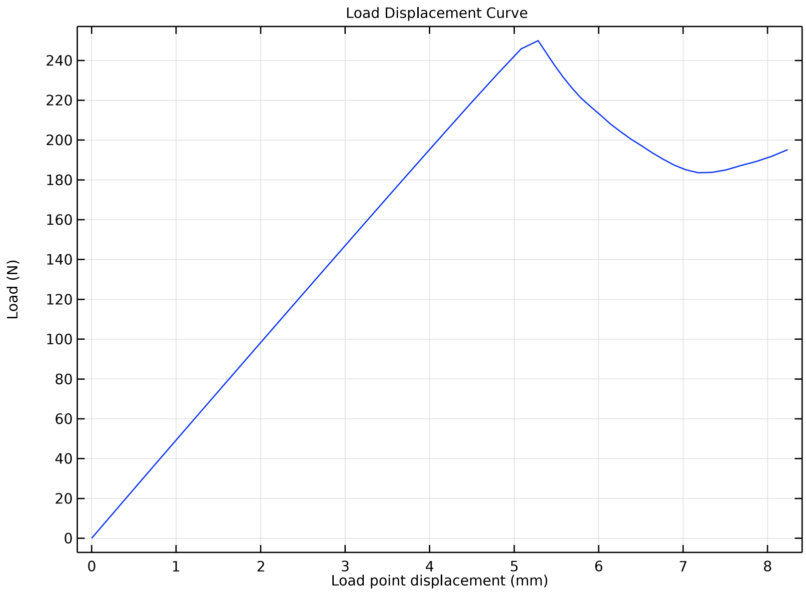

Figure 4: Load-displacement curve of the MMB test at 50% mixed-mode loading.

Notes About the COMSOL Implementation

To implement a cohesive zone model in COMSOL Multiphysics, use the **Contact** node with an **Adhesion** subnode and a **Decohesion** subnode. Notice that **Decohesion** requires an active **Adhesion** subnode.

Since no large relative deformations of the laminates are expected, the contact search can be made in the initial configuration. Setting the **Mapping method** to **Initial configuration** in the **Contact Pair** node significantly improves the performance of the model.

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.

The 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 three layered blocks.

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/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/2`.

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

Union 1 (uni1)

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

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

Copy 1 (copy1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Copy**.

2 Select the object **uni1** only.

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

4 In the **z** text field, type `hb/2`.

Form Union (fin)

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

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 From the **Pair type** list, choose **Contact pair**.

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

DEFINITIONS

Load variables for the load point from files.

Load Point Variables

1 In the **Home** toolbar, click **Variables** and choose **Local Variables**.

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 17 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 26 only.

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

Contact Pair 1 (ap1)

Remove the boundaries that correspond to the initial crack from the contact pair.

1 In the **Model Builder** window, click **Contact Pair 1 (ap1)**.

2 In the **Settings** window for **Pair**, locate the **Pair Type** section.

3 Select the **Manual control of selections and pair type** check box.

4 Locate the **Source Boundaries** section. Select the **Activate Selection** toggle button.

5 In the list, select **4**.

6 Click **Remove from Selection**.

7 Select Boundaries 9 and 14 only.

8 Locate the **Destination Boundaries** section. Select the **Activate Selection** toggle button.

9 In the list, select **19**.

10 Click **Remove from Selection**.

11 Select Boundaries 24 and 29 only.

Since no large relative deformations of the laminates are expected, the contact search can be made in the initial configuration. This setting results in a significant speed up of the problem.

12 Locate the **Advanced** section. From the **Mapping method** list, choose **Initial configuration**.

13 In the **Extrapolation tolerance** text field, type `1e-2`.

MATERIALS

Create a new material containing the orthotropic linear elastic properties of AS4/PEEK.

AS4/PEEK

1 In the **Materials** toolbar, click **Blank Material**.

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

3 Click to expand the **Material Properties** section. In the **Material properties** tree, select **Solid Mechanics>Linear Elastic Material>Orthotropic>Young's modulus (Evector)**.

4 Click **Add to Material**.

5 Locate the **Material Contents** section. 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	{nuvector 1, nuvector2 , nuvector3 }	`{0.25, 0.45, 0.25}`	1	Orthotropic
Shear modulus	{Gvector 1, Gvector2, Gvector3 }	`{5.5e9, 3.7e9, 5.5e9}`	N/m²	Orthotropic
Density	rho	`1570`	kg/m³	Basic

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

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

3 From the **Solid model** list, choose **Orthotropic**.

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

Contact 1

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

2 In the **Settings** window for **Contact**, locate the **Pair Selection** section.

3 Under **Pairs**, click **Add**.

4 In the **Add** dialog box, select **Contact Pair 1 (ap1)** in the **Pairs** list.

5 Click **OK**.

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

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

8 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.

Contact 1

In the **Model Builder** window, click **Contact 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_t text field, type `sigmat`.

4 In the σ_s 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`.

Symmetry 1

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

2 Select Boundaries 2, 7, 12, 18, 23, and 28 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 32 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 48 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 In the **Settings** window for **Prescribed Displacement**, locate the **Edge Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `2, 26` in the **Selection** text field.

5 Click **OK**.

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

7 Select the **Prescribed in z direction** check box.

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 Select the **Prescribed in x direction** check box.

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

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

7 Click **OK**.

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

Global Equations 1

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.

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

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)
`force`	`disp-u_Ie`	`0`	`0`

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

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

6 Click **Filter**.

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 box, type `length` in the text field.

12 Click **Filter**.

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

14 Click **OK**.

MESH 1

Mapped 1

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

2 Select Boundaries 4, 9, 14, 19, 24, and 29 only.

Distribution 1

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

2 Select Edges 4 and 30 only.

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

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

Distribution 2

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

2 Select Edges 5 and 31 only.

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

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

Distribution 3

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

2 Select Edges 13 and 39 only.

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

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

Distribution 4

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

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

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

4 Select Edges 21, 24, 47, and 51 only.

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

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

Swept 1

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

Distribution 1

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

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

3 In the **Number of elements** text field, type 2.

4 Click **Build All**.

STUDY 1

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

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

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

6 Click to expand the **Results While Solving** section. Select the **Plot** check box.

Solution 1 (sol1)

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

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

3 In the **Settings** window for **Stationary Solver**, locate the **General** section.

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

5 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** node, then click **State variable force (comp1.ODE1)**.

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

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

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

Use a linear predictor.

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

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

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

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

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

Switch to an undamped Newton method.

14 In the **Model Builder** window, click **Fully Coupled 1**.

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

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

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

RESULTS

Surface 1

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

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

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

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

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

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**.

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**.

The following plot corresponds to the load-displacement output of the MMB test.

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 **Label**.

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

5 In the **Expression** text field, type `u_lp`.

6 From the **Unit** list, choose **mm**.

7 Click to expand the **Legends** section. Clear the **Show legends** check box.

Load Displacement Curve

1 In the **Model Builder** window, click **Load Displacement Curve**.

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

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

Global Evaluation 1

1 In the **Results** 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 **Operation** list, choose **Maximum**.

5 Click **Evaluate**.