Created in COMSOL Multiphysics 5.6

Brittle Fracture of a Holed Plate

Introduction

This example studies the brittle fracture of a holed plate made of a cement mortar. The set up of the model, including dimensions and material properties, is based on the experimental data reported in Ref. 1. As the plate is loaded, a mixed-mode fracture is induced with a crack propagating from the predefined notch to the unsymmetrically placed hole in the center of the plate.

Fracture is modeled using a damage model that regularizes the sharp geometry of the crack by the phase field approximation. This means that the crack is described in the domain material, so the non local phase field makes the crack path independent of the mesh elements. The example shows how to define an efficient and stable solver configuration for the phase field damage method, which is often unstable for this class of problems.

Model Definition

The geometry of the specimen (Ref. 1) is shown in Figure 1. The overall dimensions of the plate equal to 65 mm in width and 120 mm in height. The thickness of the plate is constant and equal to 1 mm. A notch is placed on the left boundary, 65 mm from the bottom of the plate, in order to control where the crack initiates during loading. A mixed-mode fracture is induced by offsetting the large hole and the notch from the center of the plate. Loading is applied through displacement controlled metal pins inserted into the two smaller holes. The model assumes a plane stress condition.

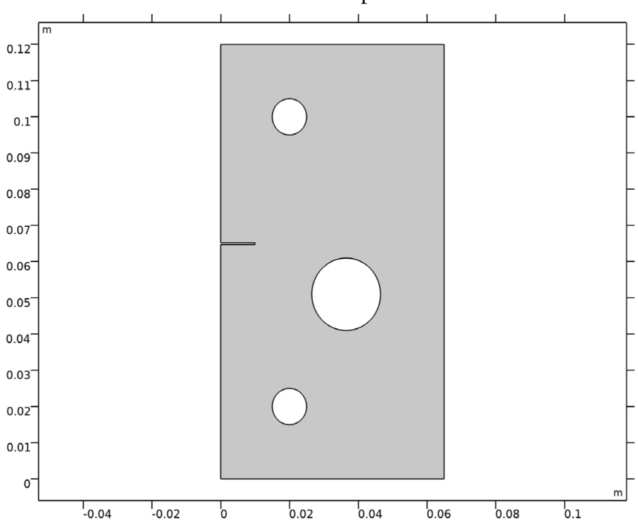

Figure 1: Geometry of the holed plate model.

The plate is made of a cement mortar composed of 22% cement, 66% sand with a grain size smaller than 1 mm, and 12% water. Material properties (from Ref. 1) are given in Table 1. In order to account for tensile cracking, the mortar is described as a linear elastic material model with damage. A phase field approximation is made after the sharp crack geometry, and cracking is incorporated in to the domain material. The set up of the phase field damage model does not include a damage threshold, meaning that all material points subjected to tension are damaged. The only material input to the damage model is the *critical energy release rate* G_c (sometimes called fracture energy), whereas the tensile strength is determined by the damage evolution function and the evolution of the phase field. The latter can be modified by the *internal length scale* l_{int} that controls the width of the localized phase field. In this example, the parameter l_{int} is set to 0.25 mm.

TABLE 1: MATERIAL PROPERTIES OF THE CEMENT MORTAR.

MATERIAL PROPERTY	VALUE
Young's modulus	6 GPa
Poisson's ratio	0.22
Critical energy release rate	2280 J/m^2

To properly resolve the phase field and to achieve a stable material behavior, a high mesh density is required in the vicinity of the propagating crack. Since the expected crack trajectory is known in advance, the mesh is locally refined with a maximum mesh element size equal to l_{int}. This length can be considered as being close to the upper limit of the appropriate mesh size. If extra computational cost is acceptable, a finer mesh size would resolve the phase field better and also yield a more stable material behavior.

The two metal pins in the model are represented by two rigid connectors attached to the boundaries of the two smaller holes. Loading is applied by prescribed displacements to the two rigid connectors while they are free to rotate. No other loads nor constraints are considered.

Results and Discussion

Figure 2 shows the reaction force in the upper rigid connector versus its prescribed displacement. A first peak in the curve can be identified at a load of 0.63 kN and a lateral displacement of 0.33 mm. As shown in Figure 3, this peak corresponds to the formation of a crack a the tip of the notch. Actually, the crack starts propagating slightly before the observed peak, see the first green circle in Figure 2 and leftmost plot in Figure 3. Once in the post-peak regime of the curve, the crack propagates rapidly and the load capacity of the plate reduces significantly. As the crack approaches the large hole, its diverts from the initially straight path; and a new stable configuration is obtained once it reaches the hole.

At this point the load can be increased, but with a much lower stiffness than prior to the formation of the crack.

A second peak is reached at a load of 0.15 kN and 1.7 mm displacement. A second crack propagates toward the right boundary of the plate, and it would eventually split the specimen in two pieces. However, due to local compressive stress fields and residual stiffness in the damaged mesh elements in the model, the crack propagation is slowed down before reaching the exterior boundary. By increasing the prescribed displacement of the rigid connector; the crack would eventually reach the exterior also in the simulation when the bending action of the plate changes to pure tensile loading.

The distribution of the phase field at the last step of the simulation is shown in Figure 4. The phase field has a smooth variation, which indicates that the mesh resolution is sufficiently fine. For the second crack, the phase field is slightly wider close the hole, since there it localized without the aid of a geometric notch. Localization of strains is an unstable event, which explains the small bleeding of the phase field observed in the second crack.

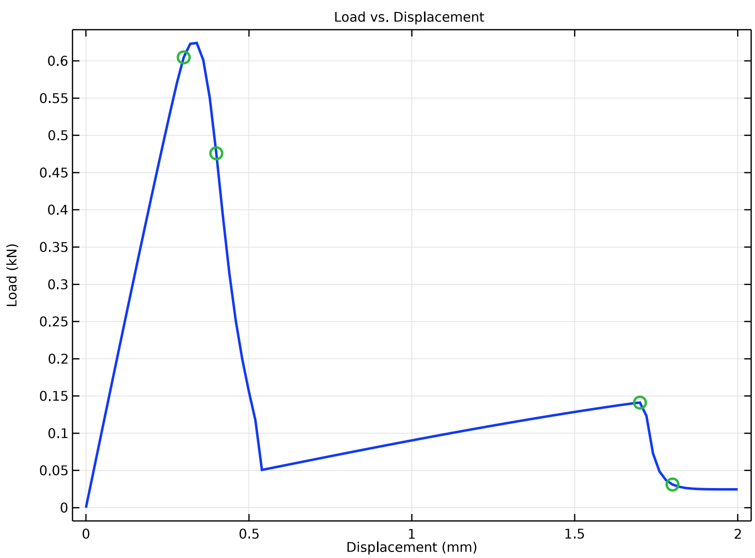

Figure 2: Load versus displacement curve with green circles indicating the snapshots presented in Figure 3.

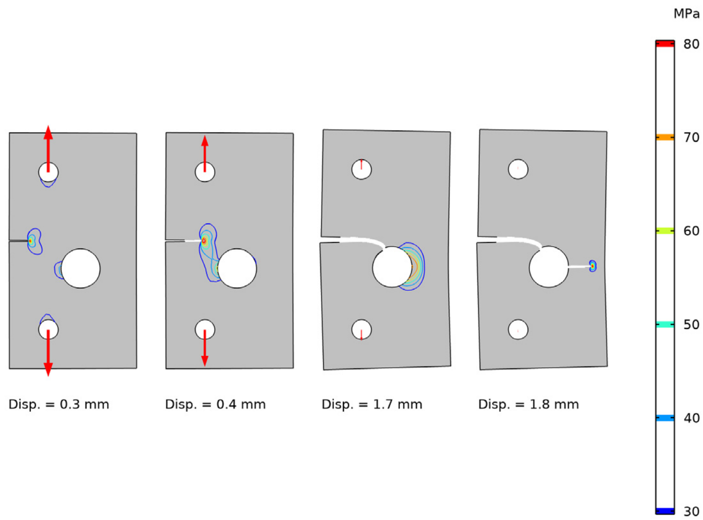

Figure 3: Crack trajectory and maximum principal stress field for different parameter steps. The red arrows indicate the relative size of the reaction force in the rigid connectors.

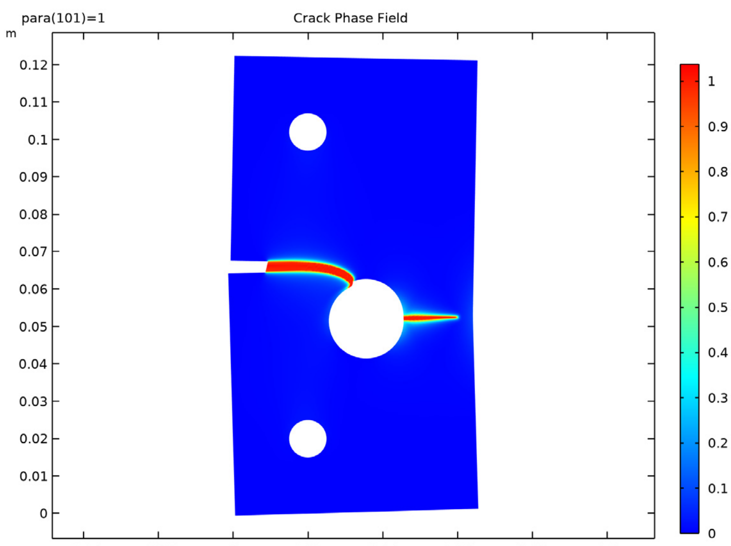

Figure 4: Crack phase field at the last parameter step.

Notes About the COMSOL Implementation

The loading of the specimen is done by adding a **Prescribed Displacement** node to each **Rigid Connector**. In this way the loading is displacement-controlled, which is necessary to track the peak loads in the global response. The simulation would have stopped before reaching the first peak if used a load-controlled set up.

While it is possible to solve this type of brittle fracture problem using a fully coupled strategy, the phase field damage model can often exhibit poor or slow convergence. In this example it is therefore shown how to use a segregated strategy to improve the convergence and stability of the numerical solution; by splitting the evolution of the crack phase field and the displacement field into two groups. This type of algorithmic operator split for phase field fracture models was originally suggested in Ref. 2, and it can conceptually be summarized as follows for step $n + 1$:

1. *Initialization*. At step n, the crack phase field, the displacement field, and other state variables are known.
2. *Update state variables*. Update internal state variables with the values from step n.
3. *Solve for the Crack Phase Field*. Compute the crack phase field variable in a Newton step, with the displacement field frozen at step n.
4. *Solve for the Displacement field*. Compute the displacement field variable in a Newton step with the updated crack phase field.

These steps lead to a single-pass algorithm that is accurate only for sufficiently small parameter steps. An improvement to this scheme is to add a multi-pass correction, by iterating in each increment over steps 3 and 4; until either convergence is achieved or for a predefined number of iterations. Such a strategy is used in this example by setting the **Number of iteration** to 3 in the **Segregated** subnode under the **Stationary Solver**.

Notice that the suggested solver configuration does not require convergence of the outer problem, and the solution is always accepted after 3 segregated iterations. However, each sub group locally fulfills the defined convergence criteria. Hence, the displacement field can be considered as a converged solution given the current crack phase field.

The accuracy of the scheme can be improved if the extra computational cost of increasing the number of iterations (or the required convergence of the outer problem) is acceptable.

References

1. M. Ambati, T. Gerasimov, and L. De Lorenzis, “A review on phase-field models of brittle fracture and a new fast hybrid formulation,” *Comput. Mech.*, vol. 55, pp. 383–405, 2015.

2. C. Miehe, M. Hofhacker, and F. Welschinger, “A phase field model for rate-independent crack propagation: Robust algorithmic implementation based on operator splits,” *Comput. Methods Appl. Mech. Eng.*, vol. 199, pp. 2765–2778, 2010.

Application Library path: `Geomechanics_Module/Damage/holed_plate_fracture`

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 **2D**.

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

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 In the table, enter the following settings:

Name	Expression	Value	Description
height	120[mm]	0.12 m	Plate height
width	65[mm]	0.065 m	Plate width
notchHeight	0.5[mm]	5E-4 m	Notch height
notchWidth	10[mm]	0.01 m	Notch width
notchLocation	65[mm]	0.065 m	Notch location
holeRadius	10[mm]	0.01 m	Hole radius
holeX	36.5[mm]	0.0365 m	Hole center, x-coordinate
holeY	51[mm]	0.051 m	Hole center, y-coordinate
elemSize	0.25[mm]	2.5E-4 m	Mesh element size
para	0	0	Load parameter

GEOMETRY 1

Rectangle 1 (r1)

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

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

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

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

Circle 1 (c1)

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

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

3 In the **Radius** text field, type `holeRadius`.

4 Locate the **Position** section. In the **x** text field, type `holeX`.

5 In the **y** text field, type `holeY`.

Circle 2 (c2)

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

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

3 In the **Radius** text field, type `5[mm]`.

4 Locate the **Position** section. In the **x** text field, type `20[mm]`.

5 In the **y** text field, type `20[mm]`.

Circle 3 (c3)

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

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

3 In the **Radius** text field, type `5[mm]`.

4 Locate the **Position** section. In the **x** text field, type `20[mm]`.

5 In the **y** text field, type `height-20[mm]`.

Rectangle 2 (r2)

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

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

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

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

5 Locate the **Position** section. In the **y** text field, type `notchLocation-notchHeight/2`.

Difference 1 (dif1)

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

2 Select the object **r1** only.

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

4 Find the **Objects to subtract** subsection. Select the **Activate Selection** toggle button.

5 Select the objects **c1**, **c2**, **c3**, and **r2** only.

Partition the plate to facilitate an increased mesh density around the expected crack path.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **x** text field, type `width`.

5 In the **y** text field, type `holeY+elemSize*12`.

6 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

7 In the **x** text field, type `holeX`.

8 In the **y** text field, type `holeY+elemSize*5`.

Line Segment 2 (ls2)

1 Right-click **Line Segment 1 (ls1)** and choose **Duplicate**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 In the **y** text field, type `holeY-elemSize*12`.

4 Locate the **Endpoint** section. In the **y** text field, type `holeY-elemSize*5`.

Polygon 1 (pol1)

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

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

3 From the **Type** list, choose **Open curve**.

4 Locate the **Coordinates** section. In the table, enter the following settings:

x (m)	y (m)
holeX	holeY
holeX	notchLocation+elemSize*12
notchWidth*3/4	notchLocation+elemSize*12
notchWidth*3/4	notchLocation+notchHeight/2

Polygon 2 (pol2)

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

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

3 From the **Type** list, choose **Open curve**.

4 Locate the **Coordinates** section. In the table, enter the following settings:

x (m)	y (m)
notchWidth*3/4	notchLocation-notchHeight/2
notchWidth*3/4	notchLocation-elemSize*5
holeX+holeRadius	holeY

Union 1 (uni1)

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

2 Click the **Select All** button in the **Graphics** toolbar.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 On the object **uni1**, select Boundaries 14–20, 22, and 23 only.

Mesh Control Domains 1 (mcd1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Mesh Control Domains**.

2 On the object **fin**, select Domains 3 and 4 only.

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

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

MATERIALS

Material 1 (mat1)

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

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	`6[GPa]`	Pa	Basic
Poisson's ratio	nu	`0.22`	1	Basic
Density	rho	2000	kg/m³	Basic

SOLID MECHANICS (SOLID)

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

2 In the **Settings** window for **Solid Mechanics**, locate the **2D Approximation** section.

3 From the list, choose **Plane stress**.

4 Locate the **Thickness** section. In the d text field, type `1[mm]`.

Linear Elastic Material 1

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

Damage 1

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

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

3 From the **Damage model** list, choose **Phase field damage**.

4 In the G_c text field, type `2280[J/m^2]`.

5 In the l_{int} text field, type `0.25[mm]`.

Rigid Connector 1

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

2 Select Boundaries 9, 10, 13, and 14 only.

3 In the **Settings** window for **Rigid Connector**, locate the **Prescribed Displacement at Center of Rotation** section.

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

5 Select the **Prescribed in y direction** check box.

6 Click to expand the **Reaction Force Settings** section. Select the **Evaluate reaction forces** check box.

Rigid Connector 2

1 Right-click **Rigid Connector 1** and choose **Duplicate**.

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

3 Click **Clear Selection**.

4 Select Boundaries 11, 12, 15, and 16 only.

5 Locate the **Prescribed Displacement at Center of Rotation** section. In the u_{0y} text field, type `para*2[mm]`.

MESH 1

Free Quad 1

In the **Mesh** toolbar, click **Free Quad**.

Size 1

1 Right-click **Free Quad 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 2 and 4 only.

5 Click to expand the **Element Size Parameters** section. Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated text field, type `elemSize`.

Size

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

2 In the **Settings** window for **Size**, click to expand the **Element Size Parameters** section.

3 In the **Maximum element size** text field, type `0.0075`.

4 In the **Maximum element growth rate** text field, type `2`.

5 Click **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**, 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	Parameter unit
para (Load parameter)	`range(0,0.01,1)`	

Set up a segregated solver sequence to improve the convergence when using the phase field fracture model.

Solution 1 (sol1)

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

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

Use a small parameter step to improve the accuracy and stability of the model.

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

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

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

6 In the **Maximum step size** text field, type `0.0025`.

7 In the **Initial step size** text field, type `0.0025`.

8 In the **Model Builder** window, right-click **Stationary Solver 1** and choose **Segregated**.

Restrict the number of outer iterations to three. Allowing more outer iterations improves the accuracy of the phase field, but comes at an additional computational cost.

9 In the **Settings** window for **Segregated**, locate the **General** section.

10 From the **Termination technique** list, choose **Iterations or tolerance**.

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

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

13 In the **Settings** window for **Segregated Step**, type `Displacement Field` in the **Label** text field.

14 Locate the **General** section. In the **Variables** list, select **Crack phase field (comp1.solid.phic)**.

15 Under **Variables**, click **Delete**.

16 Click to expand the **Method and Termination** section. From the **Termination technique** list, choose **Tolerance**.

17 In the **Tolerance factor** text field, type `1`.

18 In the **Model Builder** window, right-click **Segregated 1** and choose **Segregated Step**.

19 In the **Settings** window for **Segregated Step**, type `Crack Phase Field` in the **Label** text field.

20 Locate the **General** section. Under **Variables**, click **Add**.

21 In the **Add** dialog box, select **Crack phase field (comp1.solid.phic)** in the **Variables** list.

22 Click **OK**.

23 In the **Settings** window for **Segregated Step**, locate the **Method and Termination** section.

24 From the **Termination technique** list, choose **Tolerance**.

25 In the **Tolerance factor** text field, type `1`.

26 Right-click **Crack Phase Field** and choose **Move Up**.

Generate datasets and default plots.

27 In the **Study** toolbar, click **Get Initial Value**.

RESULTS

Crack Phase Field

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

2 In the **Settings** window for **2D Plot Group**, type `Crack Phase Field` 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. Clear the **Plot dataset edges** check box.

Surface 1

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

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

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

Deformation

1 In the **Model Builder** window, expand the **Surface 1** node, then click **Deformation**.

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

3 Select the **Scale factor** check box.

4 In the associated text field, type `1`.

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**, click to expand the **Results While Solving** section.

3 Select the **Plot** check box.

4 From the **Update at** list, choose **Steps taken by solver**.

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

RESULTS

Crack Trajectory

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Crack Trajectory` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Label**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

5 Locate the **Color Legend** section. Select the **Show units** check box.

Surface 1

1 Right-click **Crack Trajectory** and choose **Surface**.

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

3 In the **Expression** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Gray**.

Deformation 1

1 Right-click **Surface 1** and choose **Deformation**.

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

3 Select the **Scale factor** check box.

4 In the associated text field, type `1`.

Filter 1

1 In the **Model Builder** window, right-click **Surface 1** and choose **Filter**.

2 In the **Settings** window for **Filter**, locate the **Element Selection** section.

3 In the **Logical expression for inclusion** text field, type `solid.phic<0.9`.

4 From the **Element nodes to fulfill expression** list, choose **All**.

Line 1

1 In the **Model Builder** window, right-click **Crack Trajectory** and choose **Line**.

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

3 In the **Expression** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Black**.

6 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

7 Clear the **Color** check box.

8 Clear the **Color and data range** check box.

Deformation 1

Right-click **Line 1** and choose **Deformation**.

Filter 1

1 In the **Model Builder** window, right-click **Line 1** and choose **Filter**.

2 In the **Settings** window for **Filter**, locate the **Element Selection** section.

3 In the **Logical expression for inclusion** text field, type `solid.phic<0.9`.

4 From the **Element nodes to fulfill expression** list, choose **All**.

Contour 1

1 In the **Model Builder** window, right-click **Crack Trajectory** and choose **Contour**.

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

3 In the **Expression** text field, type `solid.sdp1Gp`.

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

5 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

6 In the **Levels** text field, type `range(30,10,80)`.

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

8 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

9 Clear the **Color** check box.

10 Clear the **Color and data range** check box.

Deformation 1

Right-click **Contour 1** and choose **Deformation**.

Filter 1

1 In the **Model Builder** window, right-click **Contour 1** and choose **Filter**.

2 In the **Settings** window for **Filter**, locate the **Element Selection** section.

3 In the **Logical expression for inclusion** text field, type `solid.phic<0.9`.

4 From the **Element nodes to fulfill expression** list, choose **All**.

Crack Trajectory

In the **Model Builder** window, click **Crack Trajectory**.

Point Trajectories 1

1 In the **Crack Trajectory** toolbar, click **More Plots** and choose **Point Trajectories**.

2 In the **Settings** window for **Point Trajectories**, click **Replace Expression** in the upper-right corner of the **Trajectory Data** section. From the menu, choose **Component 1 (comp1)> Solid Mechanics>Rigid connectors>Rigid Connector 1>solid.xcx_rig1,solid.xcy_rig1 - Global coordinates of center of rotation (spatial frame)**.

3 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **None**.

4 Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

5 From the menu, choose **Component 1 (comp1)>Solid Mechanics>Rigid connectors> Rigid Connector 1>solid.rig1.RFx,solid.rig1.RFy - Reaction force (spatial frame)**.

6 Select the **Scale factor** check box.

7 In the associated text field, type `4E-5`.

8 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Deformation 1

1 Right-click **Point Trajectories 1** and choose **Deformation**.

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

3 In the **X component** text field, type `solid.rig1.u`.

4 In the **Y component** text field, type `solid.rig1.v`.

Point Trajectories 2

1 In the **Model Builder** window, under **Results>Crack Trajectory** right-click **Point Trajectories 1** and choose **Duplicate**.

2 In the **Settings** window for **Point Trajectories**, locate the **Trajectory Data** section.

3 In the **X-expression** text field, type `solid.xcx_rig2`.

4 In the **Y-expression** text field, type `solid.xcy_rig2`.

5 Locate the **Coloring and Style** section. Find the **Point style** subsection. In the **Arrow, X component** text field, type `solid.rig2.RFx`.

6 In the **Arrow, Y component** text field, type `solid.rig2.RFy`.

7 Locate the **Inherit Style** section. From the **Plot** list, choose **Point Trajectories 1**.

Deformation 1

1 In the **Model Builder** window, expand the **Point Trajectories 2** node, then click **Deformation 1**.

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

3 In the **X component** text field, type `solid.rig2.u`.

4 In the **Y component** text field, type `solid.rig2.v`.

5 In the **Crack Trajectory** toolbar, click **Plot**.

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

Animation 1

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

2 In the **Settings** window for **Animation**, locate the **Frames** section.

3 From the **Frame selection** list, choose **All**.

4 Right-click **Animation 1** and choose **Play**.

Load vs. Displacement

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` in the **Label** text field.

3 In the **Label** text field, type `Load vs. Displacement`.

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

5 Locate the **Plot Settings** section. Select the **x-axis label** check box.

6 In the associated text field, type `Displacement (mm)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Load (kN)`.

Global 1

1 Right-click **Load vs. Displacement** 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
`solid.rig2.RFy`	`N`	`Reaction force, y component`

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

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

Expression	Unit	Description
`solid.rig2.RFy`	`kN`	`Reaction force, y component`

6 Locate the **x-Axis Data** section. In the **Expression** text field, type `solid.rig2.v`.

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

8 Click to expand the **Coloring and Style** section. In the **Width** text field, type `2`.

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