Created in COMSOL Multiphysics 5.6

Cracking of a Notched Beam

Introduction

Cracking is an important failure mechanism in many materials, especially in brittle and quasi-brittle materials such as concrete. To model the failure of such materials require not only an appropriate material model, but also modeling techniques to properly describe the size effect in order to avoid mesh dependent results.

In this example, failure is examined by simulating a three-point bending fracture test of a notched concrete beam. The model is based on the experiments presented in Ref. 1. In that paper, several tests are presented on specimens of different sizes and configurations (different notch sizes or no notch) to study the size effect. In this example one of those specimens is modeled, although the model is parameterized such that all specimens from the reference can be reproduced. The results are compared to measured data.

Model Definition

The geometry of a general specimen from Ref. 1 is given in Figure 1. The parameter D_n is defined by

$$D_n = \frac{D_0}{2^{n-1}} \tag{1}$$

where D_0 = 400 mm and $n = (1, 2, 3, 4)$.

Furthermore, the notch is described by its depth $\alpha_m = (0.5, 0.2, 0)$. In this example, the specimen given by $n = 2$ and $\alpha_m = 0.2$ is modeled. Notice that the width of the notch is set to one mesh element. The depth of the beam is constant and equal to 50 mm. The assumption of plane stress is used. The beam is simply supported and loaded with a 'distributed' point load above the notch. As in the experimental set up described in Ref. 1, the load is controlled through the crack mouth opening displacement (CMOD) which is monotonically increased for all tested specimens. Although the deflection is also monotonically increased for $n = 2$ and $\alpha_m = 0.2$, this might not be the case for larger specimens.

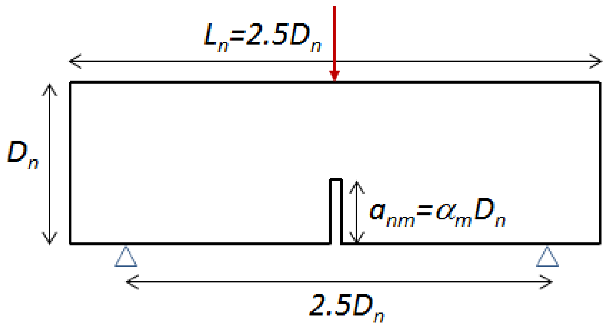


Figure 1: Geometry of the specimens, as specified in Ref. 1.

According to Ref. 1, measurements on the used concrete gave a Young's modulus of 37 GPa and a Poisson's ratio of 0.21. The measured compressive strength is 42.3 MPa and the tensile strength is 3.9 MPa. The material is described by a linear elastic material model with damage to account for tensile cracking. The damage model is set up to describe tensile material failure by using the Rankine equivalent strain definition. A damage evolution law with an exponential strain softening is used. With this option, the fracture energy enters as an additional input parameter, and is here set to 85 J/m^2. When modeling tensile cracking it is also necessary to add regularization to the damage model in order to ensure that a consistent amount of energy is dissipated during mesh refinement or for different discretization orders. Two techniques for this are available in COMSOL Multiphysics and both are evaluated in this model:

- The crack band method
- The implicit gradient method

The crack band method considers the current discretization and modifies the damage model locally at each material point based on the element size. A more refined approach is to use the implicit gradient method which enforces a predefined width of the damage zone through a localization limiter. This is done by adding a nonlocal strain variable and an internal length scale to the material model through an additional PDE that is solved simultaneously with the displacements. While the recommendation to use a linear displacement field when using the crack band method is used, the study with the implicit gradient method is set up to use the default discretization settings (that is, quadratic serendipity). For simplicity, the same mesh size is used for both cases. This can be considered as a bit unfair when comparing results; when using first order discretization a significantly finer mesh would be needed to obtain a similar accuracy in both methods.

Results and Discussion

Two studies are defined to compare the two regularization techniques:

- The crack band method with a linear displacement field
- The implicit gradient method

The results from both studies are compared in Figure 2 showing the force versus CMOD curves as well as the measured data from Ref. 1. As an additional comparison, the force versus deflection curves are shown in Figure 3, despite no measured data being available. The crack band method in Study 1 is in very good agreement with the measured data at the peak load, while for the post peak behavior, the simulated force is underestimated for a given CMOD. For the implicit gradient method in Study 2, the peak load is overestimated by approximately 1 kN. Also for this case, the force is underestimated for the post peak response. However, both solutions can describe the overall failure behavior of the beam.

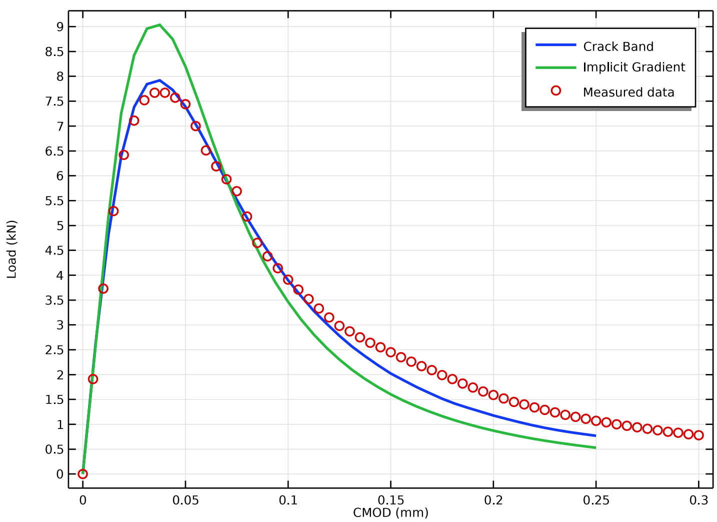


Figure 2: Load versus crack mouth opening displacement.

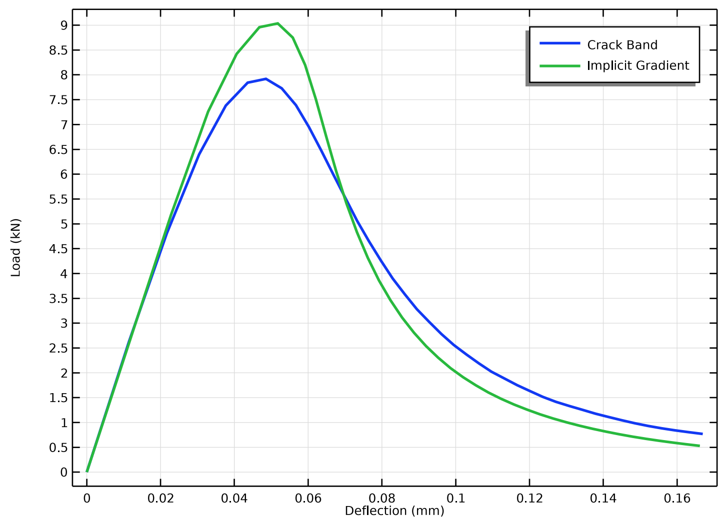


Figure 3: Load versus vertical deflection.

The results of Study 1 can be analyzed in more detail in Figure 4 and Figure 5, which show the damage and strain fields, respectively, at the final step of the solution. As expected for the crack band method, both the damage and the strains are localized in to a single row of elements. It can also be noticed that the crack almost propagates through the entire height of the beam.

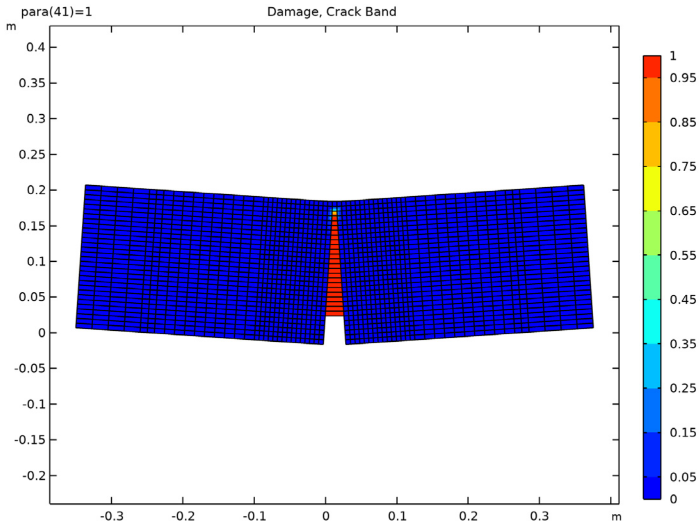


Figure 4: Damage distribution when using the crack band model.

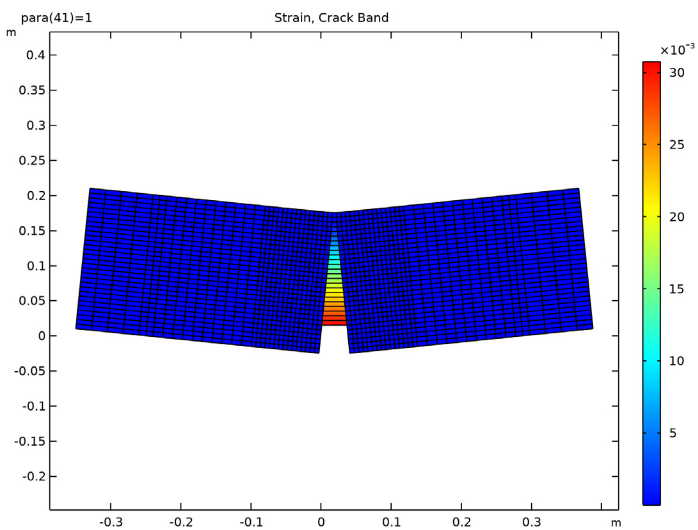


Figure 5: Distribution of equivalent strain when using the crack band model.

The same plots are shown for the results from Study 2 in Figure 6 and Figure 7. For this modeling technique, a clearly distributed zone of damage over several elements is obtained. Also, the strains are much less localized as compared to the crack band method. In Figure 6, it can be noticed that the damage field stretches below the tip of the notch. This is a nonphysical behavior that results from the nonlocal modeling approach. This could explain the overestimation of the peak load seen in Figure 2 and is a general issue of nonlocal regularization methods for problems where a crack initiates at non-trivial physical boundaries, such as a notch. Finally, the stress versus strain response of the models are compared in Figure 8 at four points located at different heights above the notch.

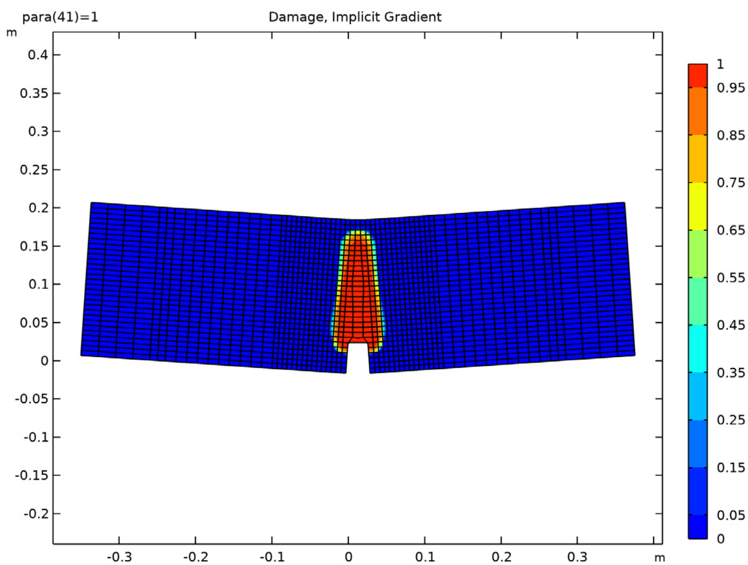


Figure 6: Damage distribution when using the implicit gradient model.

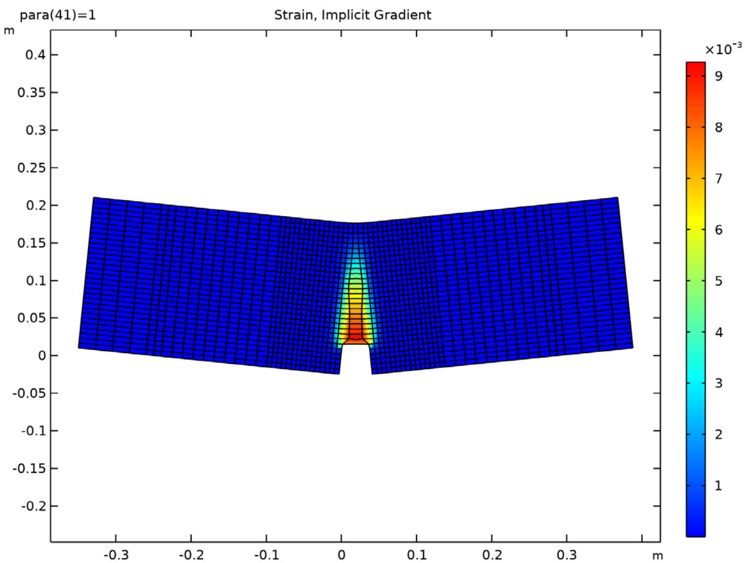


Figure 7: Distribution of equivalent strain when using the implicit gradient model.

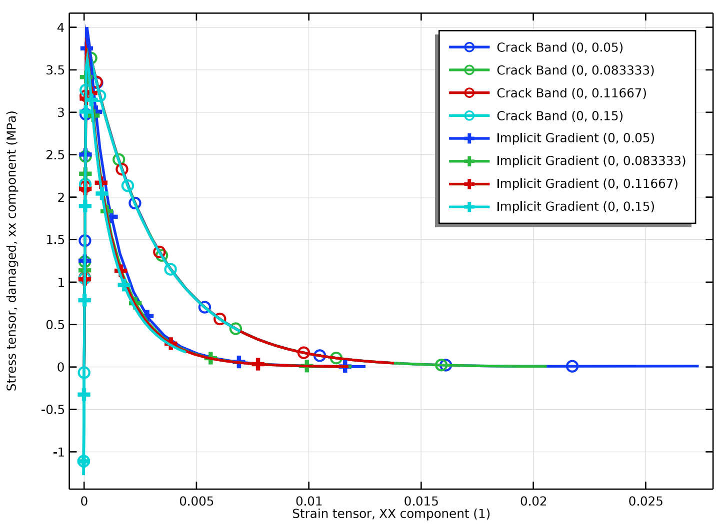


Figure 8: Horizontal stress versus horizontal strain at four points above the notch.

Notes about the COMSOL implementation

Apart from being prone to tensile cracking, concrete is also highly nonlinear in compression. Accounting for this can improve the force versus CMOD curves. This can for example be done by using the Mazars damage model for concrete available in the Geomechanics module.

The loading of the beam is controlled through the CMOD. A suitable modeling technique to accomplish this is to use an algebraic equation that controls the applied force so that the model reaches the desired CMOD. This is implemented using a **Global Equation**, and the parametric solver steps up the desired CMOD.

Although a single specimen is analyzed, the model is parameterized so that all set-ups presented in Ref. 1 can be described. For example, the size effect can be examined by running a parametric sweep over n. However, if a specimen without a notch is to be modeled by setting $i = 0$, some minor modifications to the geometry and load control are necessary:

- Remove the part of the geometry that creates the notch.
- Redefine the position of the CMOD coupling operators so that the measuring points are further from the midsection.

For some configurations, the nonlinear solver also have to be modified to obtain a converging solution. In particular, this is the case for the specimens without a notch. For example, better convergence is often obtained if in the **Fully Coupled** node the **Nonlinear method** is set to **Constant (Newton)** with **Jacobian update** on every iteration.

Reference

1. D. Grégoire, L.B. Rojas-Solano, and G. Pijaudier-Cabot, "Failure and size effect for notched and unnotched concrete beams," *Int. J. Numer. Anal. Meth. Geomech.*, vol. 37, no. 10, pp. 1434–1452, 2013.

Application Library path: `Nonlinear_Structural_Materials_Module/Damage/notched_beam_damage`

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 Physics**.

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 `notched_beam_damage_parameters.txt`.

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 `3.5*Dn`.

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

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 In the **y** text field, type `Dn/2`.

7 Click to expand the **Layers** section. Add a number of subdivisions to the beam in order to facilitate application of loads and constraints, as well as mesh control.

8 Select the **Layers to the left** check box.

9 Clear the **Layers on bottom** check box.

10 In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`0.5*Dn-loadSurf/2`
`Layer 2`	`loadSurf`
`Layer 3`	`0.75*Dn-loadSurf/2`
`Layer 4`	`0.5*Dn-loadSurf/2`
`Layer 5`	`loadSurf`
`Layer 6`	`0.5*Dn-loadSurf/2`
`Layer 7`	`0.75*Dn-loadSurf/2`
`Layer 8`	`loadSurf`

Parameter Check 1 (pch1)

1 In the **Geometry** toolbar, click **Programming** and choose **Parameter Check**.

2 In the **Settings** window for **Parameter Check**, locate the **Parameter Check** section.

3 In the **Condition** text field, type `am<=0`.

4 In the **Error message** text field, type `Notch size must be larger than zero`.

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 `eSize`.

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

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 In the **y** text field, type `am/2`.

Difference 1 (dif1)

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

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

3 Select the object **r1** only.

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

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

6 Select the object **r2** only.

7 Click **Build Selected**.

Polygon 1 (pol1)

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

Add a help line for easy mapped meshing.

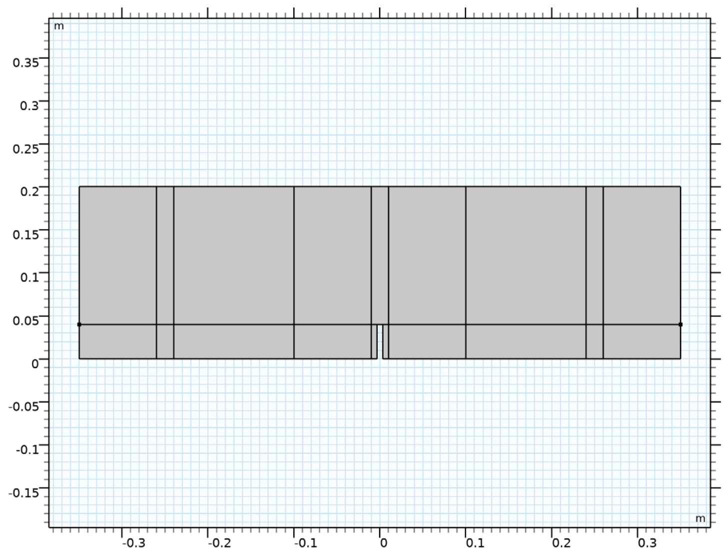
2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 From the **Data source** list, choose **Vectors**.

4 In the **x** text field, type `-Dn*1.75 Dn*1.75`.

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

6 Click **Build All Objects**.



DEFINITIONS

Create a variable measuring the crack mouth opening displacement (CMOD).

Crack Mouth Opening Displacment, Right

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

2 In the **Settings** window for **Average**, type `CMOD_right` in the **Operator name** text field.

3 In the **Label** text field, type `Crack Mouth Opening Displacment, Right`.

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

5 Select Point 18 only.

Crack Mouth Opening Displacment, Left

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

2 In the **Settings** window for **Average**, type `Crack Mouth Opening Displacment, Left` in the **Label** text field.

3 In the **Operator name** text field, type `CMOD_left`.

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

5 Select Point 16 only.

Variables 1

1 In the **Model Builder** window, 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
CMOD	CMOD_right(u)-CMOD_left(u)	m	

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 `depth`.

Linear Elastic Material 1

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

Damage: Crack Band

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

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

3 Find the **Damage evolution** subsection. In the G_{f} text field, type `85`.

4 In the **Label** text field, type `Damage: Crack Band`.

Damage: Implicit Gradient

1 Right-click **Damage: Crack Band** and choose **Duplicate**.

2 In the **Settings** window for **Damage**, type `Damage: Implicit Gradient` in the **Label** text field.

3 Locate the **Damage** section. Find the **Spatial regularization method** subsection. From the list, choose **Implicit gradient**.

4 In the l_{int} text field, type `lscale`.

5 In the h_{dmg} text field, type `3*lscale`.

Rigid Connector 1

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

2 Select Boundary 7 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.

Rigid Connector 2

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

2 Select Boundary 42 only.

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

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

Boundary Load 1

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

2 Select Boundary 25 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

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

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

0	x
-load	y

The applied load is controlled through the CMOD.

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

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

8 In the tree, select the check box for the node **Physics>Advanced Physics Options**.

9 Click **OK**.

Load Control

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

2 In the **Settings** window for **Global Equations**, type `Load Control` in the **Label** text field.

3 Locate the **Global Equations** section. 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)	Description
load	CMOD-para* maxCMOD	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 displ in the text field.

12 Click **Filter**.

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

14 Click **OK**.

Discretization, Linear

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

For the crack band method, linear shape order for the displacements is preferred.

2 In the **Settings** window for **Discretization**, type Discretization, Linear in the **Label** text field.

3 Locate the **Discretization** section. From the **Displacement field** list, choose **Linear**.

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	37[GPa]	Pa	Basic
Poisson's ratio	nu	0.21	1	Basic

Property	Variable	Value	Unit	Property group
Density	rho	2400	kg/m³	Basic
Tensile strength	sigmat	3.9[MPa]	Pa	Isotropic strength parameters

MESH 1

Mapped 1

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

Size

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

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

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type eSize.

Distribution 1

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

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

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

4 Select Boundaries 2 and 47 only.

Distribution 2

1 Right-click **Mapped 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 Select Boundaries 7 and 42 only.

Distribution 3

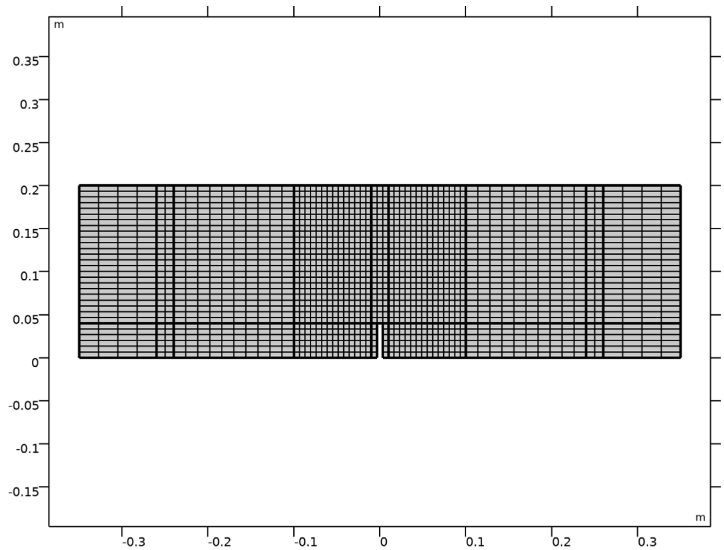
1 Right-click **Mapped 1** and choose **Distribution**.

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

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

4 Select Boundaries 12 and 37 only.

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 **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Linear Elastic Material 1>Damage: Implicit Gradient**.

5 Click **Disable**.

6 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid)**.

7 Click **Discretization, Linear**.

8 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

9 Click **Add**.

10 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Loading parameter)	`range(0,0.025,1)`	

11 In the **Model Builder** window, click **Study 1**.

12 In the **Settings** window for **Study**, type `Study: Crack Band` in the **Label** text field.

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

RESULTS

Damage, Crack Band

1 In the **Model Builder** window, under **Results** click **Damage (solid)**.

2 In the **Settings** window for **2D Plot Group**, type `Damage, Crack Band` 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.

Deformation 1

1 In the **Model Builder** window, expand the **Damage, Crack Band** node.

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

Deformation 1

1 In the **Model Builder** window, expand the **Results>Damage, Crack Band>Contour 1** node, then click **Deformation 1**.

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 `100`.

Mesh 1

1 In the **Model Builder** window, right-click **Damage, Crack Band** and choose **Mesh**.

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

3 From the **Element color** list, choose **None**.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Contour 1**.

Deformation 1

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

2 In the **Damage, Crack Band** toolbar, click **Plot**.

Strain, Crack Band

1 In the **Model Builder** window, right-click **Stress (solid)** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Strain, Crack Band` 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, expand the **Strain, Crack Band** node, then click **Surface 1**.

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>Damage>solid.kappadmg - Maximum value of equivalent strain**.

3 Click to expand the **Quality** section. From the **Resolution** list, choose **No refinement**.

4 From the **Smoothing** list, choose **None**.

Mesh 1

1 In the **Model Builder** window, right-click **Strain, Crack Band** and choose **Mesh**.

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

3 From the **Element color** list, choose **None**.

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

Deformation 1

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

2 In the **Strain, Crack Band** toolbar, click **Plot**.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

4 Click **Add Study**.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Stationary

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

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Loading parameter)	range(0,0.025,1)	

5 In the **Model Builder** window, click **Study 2**.

6 In the **Settings** window for **Study**, type `Study: Implicit Gradient` in the **Label** text field.

7 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

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

RESULTS

Damage, Crack Band, Strain, Crack Band

In the **Model Builder** window, under **Results**, Ctrl-click to select **Damage, Crack Band** and **Strain, Crack Band**.

Damage, Implicit Gradient

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

2 In the **Settings** window for **2D Plot Group**, type `Damage, Implicit Gradient` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Implicit Gradient/ Solution 2 (sol2)**.

4 In the **Damage, Implicit Gradient** toolbar, click **Plot**.

Strain, Implicit Gradient

1 In the **Model Builder** window, under **Results** click **Strain, Crack Band 1**.

2 In the **Settings** window for **2D Plot Group**, type `Strain, Implicit Gradient` in the **Label** text field.

3 In the **Strain, Implicit Gradient** toolbar, click **Plot**.

4 Locate the **Data** section. From the **Dataset** list, choose **Study: Implicit Gradient/ Solution 2 (sol2)**.

5 In the **Strain, Implicit Gradient** toolbar, click **Plot**.

Load vs. Deflection

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. Deflection` in the **Label** text field.

Global 1

1 Right-click **Load vs. Deflection** 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
`load`	`kN`	`State variable load`

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

5 In the **Expression** text field, type `-CMOD_left(v)`.

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

7 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
`Crack Band`

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

10 In the **Load vs. Deflection** toolbar, click **Plot**.

Global 2

1 Right-click **Global 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study: Implicit Gradient/Solution 2 (sol2)**.

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

Legends
`Implicit Gradient`

5 In the **Load vs. Deflection** toolbar, click **Plot**.

Load vs. Deflection

1 In the **Model Builder** window, click **Load vs. Deflection**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

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

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

5 In the associated text field, type `Deflection (mm)`.

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

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

8 In the **Load vs. Deflection** toolbar, click **Plot**.

Load vs. CMOD

1 Right-click **Load vs. Deflection** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Load vs. CMOD` in the **Label** text field.

3 Locate the **Plot Settings** section. In the **x-axis label** text field, type `CMOD (mm)`.

Global 1

1 In the **Model Builder** window, expand the **Load vs. CMOD** node, then click **Global 1**.

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

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

Global 2

1 In the **Model Builder** window, click **Global 2**.

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

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

Table 1

1 In the **Results** toolbar, click **Table**.

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

3 Click **Import**.

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

TABLE

1 Go to the **Table** window.

2 Click **Table Graph** in the window toolbar.

RESULTS

Table Graph 1

In the **Model Builder** window, under **Results>1D Plot Group 9** right-click **Table Graph 1** and choose **Copy**.

Table Graph 1

1 In the **Model Builder** window, right-click **Load vs. CMOD** and choose **Paste Table Graph**.

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

3 Find the **Line style** subsection. From the **Line** list, choose **None**.

4 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

5 From the **Positioning** list, choose **In data points**.

6 Click to expand the **Legends** section. Select the **Show legends** check box.

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

8 In the table, enter the following settings:

Legends
`Measured data`

9 In the **Load vs. CMOD** toolbar, click **Plot**.

1D Plot Group 9

In the **Model Builder** window, right-click **1D Plot Group 9** and choose **Delete**.

Cut Points, Study 1

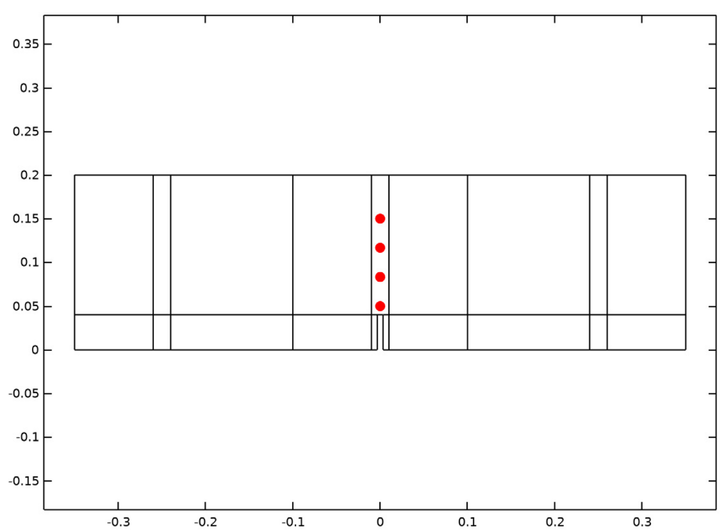
1 In the **Results** toolbar, click **Cut Point 2D**.

2 In the **Settings** window for **Cut Point 2D**, type `Cut Points, Study 1` in the **Label** text field.

3 Locate the **Point Data** section. In the **X** text field, type `0`.

4 In the **Y** text field, type `{3*eSize/2 13*eSize/2 23*eSize/2 33*eSize/2}+am`.

5 Click **Plot**.



Cut Points, Study 2

1 Right-click **Cut Points, Study 1** and choose **Duplicate**.

2 In the **Settings** window for **Cut Point 2D**, type `Cut Points, Study 2` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Implicit Gradient/ Solution 2 (sol2)**.

Damaged Stress vs. Strain

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

2 In the **Settings** window for **1D Plot Group**, type `Damaged Stress vs. Strain` in the **Label** text field.

Point Graph 1

1 Right-click **Damaged Stress vs. Strain** and choose **Point Graph**.

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

3 From the **Dataset** list, choose **Cut Points, Study 1**.

4 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Damage>Stress tensor, damaged (spatial frame) - N/m²>solid.sdxx - Stress tensor, damaged, xx component**.

5 Locate the **y-Axis Data** section. From the **Unit** list, choose **MPa**.

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

7 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Strain> Strain tensor (material and geometry frames)>solid.eXX - Strain tensor, XX component**.

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

9 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

10 Click to expand the **Legends** section. Select the **Show legends** check box.

11 Find the **Include** subsection. In the **Prefix** text field, type `Crack Band` .

12 In the **Damaged Stress vs. Strain** toolbar, click **Plot**.

Point Graph 2

1 Right-click **Point Graph 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Cut Points, Study 2**.

4 Locate the **Legends** section. Find the **Include** subsection. In the **Prefix** text field, type `Implicit Gradient` .

5 Locate the **Coloring and Style** section. From the **Color** list, choose **Cycle (reset)**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Plus sign**.

Damaged Stress vs. Strain

1 In the **Model Builder** window, click **Damaged Stress vs. Strain**.

2 In the **Settings** window for **1D Plot Group**, locate the **Title** section.

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