



Model created in COMSOL Multiphysics 6.4

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 requires 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

$$D_n = \frac{D_0}{2^{n-1}} \quad (1)$$

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. Plane stress conditions are used. The beam is simply supported and loaded with a ‘distributed’ point load above the notch. As in the experimental setup 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.

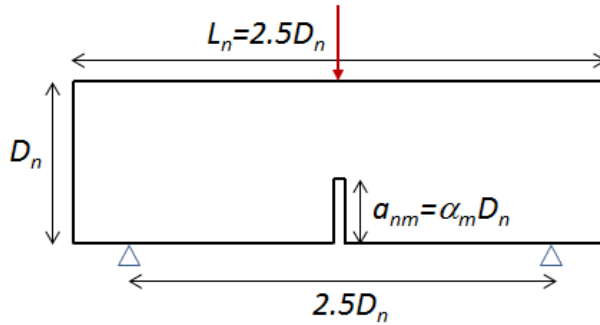


Figure 1: Geometry of the specimens, as specified in Ref. 1.

According to Ref. 1, measurements on the concrete used 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 exponential strain softening is used. With this option, the fracture energy enters as an additional input parameter, which 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 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 in this case, the force is underestimated for the post-peak response. However, both solutions can describe the overall failure behavior of the beam.

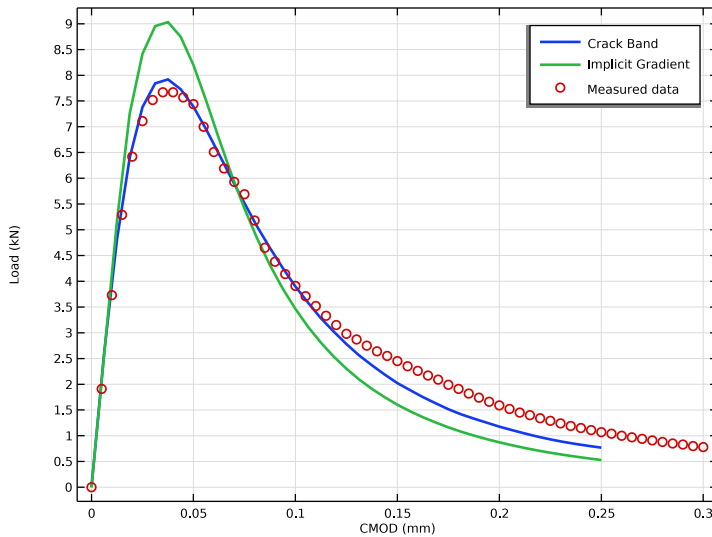


Figure 2: Load versus crack mouth opening displacement.

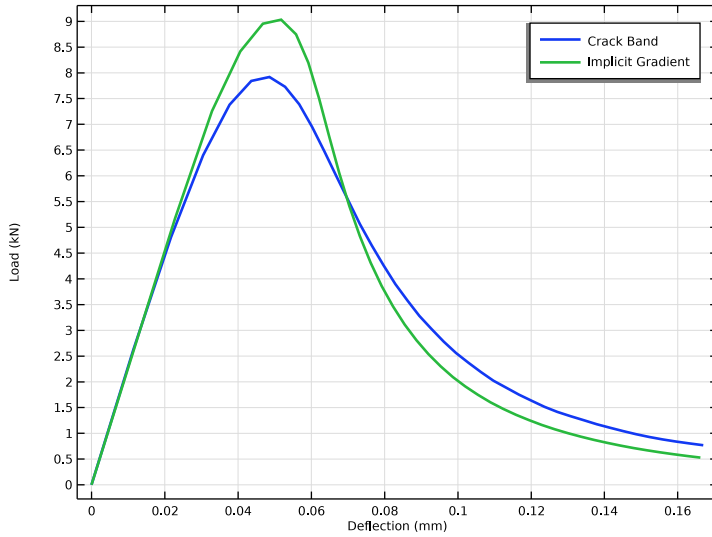


Figure 3: Load versus vertical deflection.

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 into a single row of elements. It can also be noticed that the crack almost propagates through the entire height of the beam.

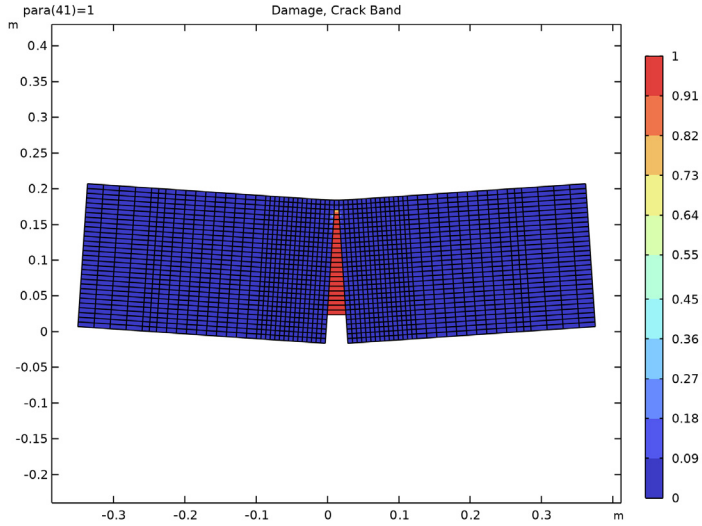


Figure 4: Damage distribution when using the crack band model.

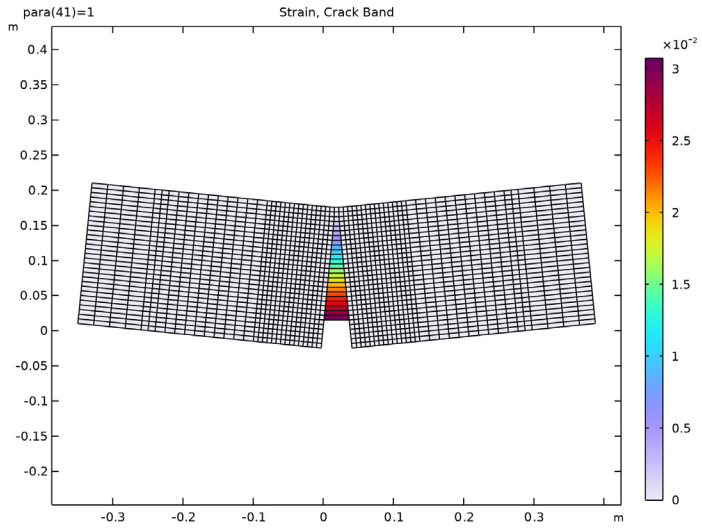


Figure 5: Equivalent strain distribution when using the crack band model.

The same plots are shown for the implicit gradient method from Study 2 in [Figure 6](#) and [Figure 7](#). For this modeling technique, a 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 an unphysical 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 nontrivial physical boundaries, such as a notch. Finally, the stress versus strain response of the models is compared in [Figure 8](#) at four points located at different heights above the notch.

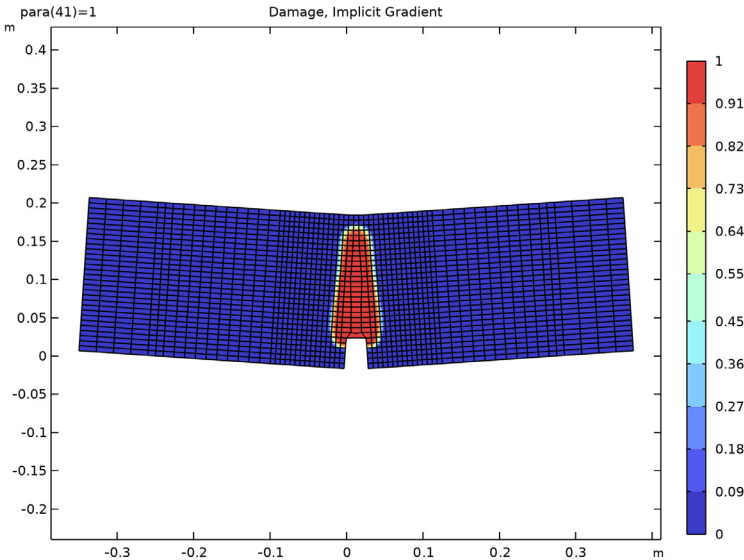


Figure 6: Damage distribution when using the implicit gradient model.

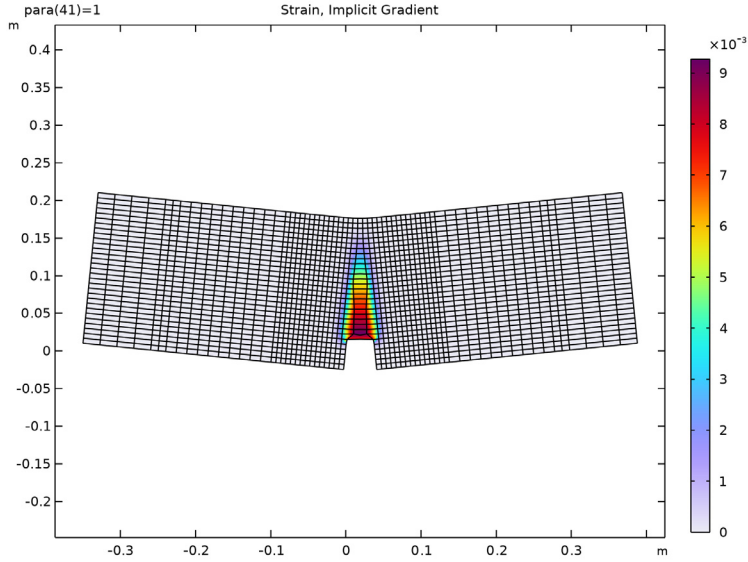


Figure 7: Equivalent strain distribution when using the implicit gradient model.

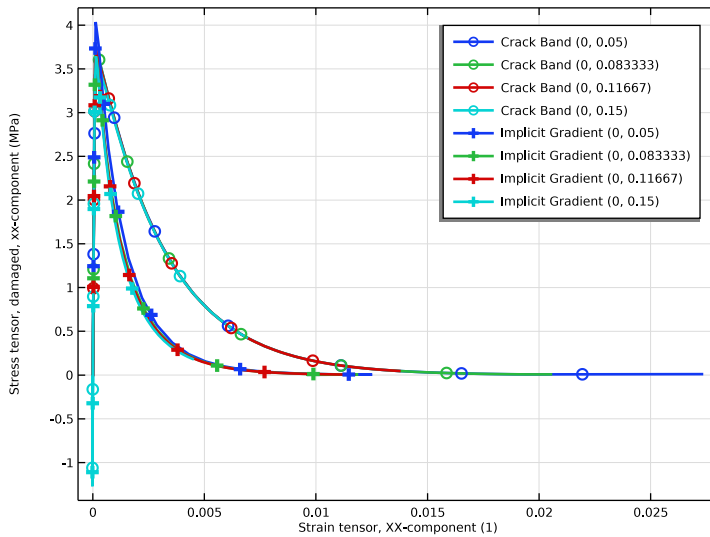


Figure 8: Horizontal stress versus horizontal strain at four points above the notch.

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 with 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 has 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 Right-click and choose **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 $3.5 \cdot D_n$.
- 4 In the **Height** text field, type D_n .
- 5 Locate the **Position** section. From the **Base** list, choose **Center**.
- 6 In the **y** text field, type $D_n/2$.
- 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** checkbox.
- 9 Clear the **Layers on bottom** checkbox.


10 In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	$0.5 \cdot D_n - \text{loadSurf} / 2$
Layer 2	loadSurf
Layer 3	$0.75 \cdot D_n - \text{loadSurf} / 2$
Layer 4	$0.5 \cdot D_n - \text{loadSurf} / 2$
Layer 5	loadSurf
Layer 6	$0.5 \cdot D_n - \text{loadSurf} / 2$
Layer 7	$0.75 \cdot D_n - \text{loadSurf} / 2$
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 for message** text field, type $am \leq 0$.
- 4 In the **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 **Model Builder** window, click **Difference 1 (dif1)**.
- 5 In the **Settings** window for **Difference**, locate the **Difference** section.
- 6 Click to select the  **Activate Selection** toggle button for **Objects to subtract**.
- 7 Select the object **r2** only.
- 8 Click  **Build Selected**.

Polygon 1 (pol1)

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

Add a help line to simplify mapped meshing.

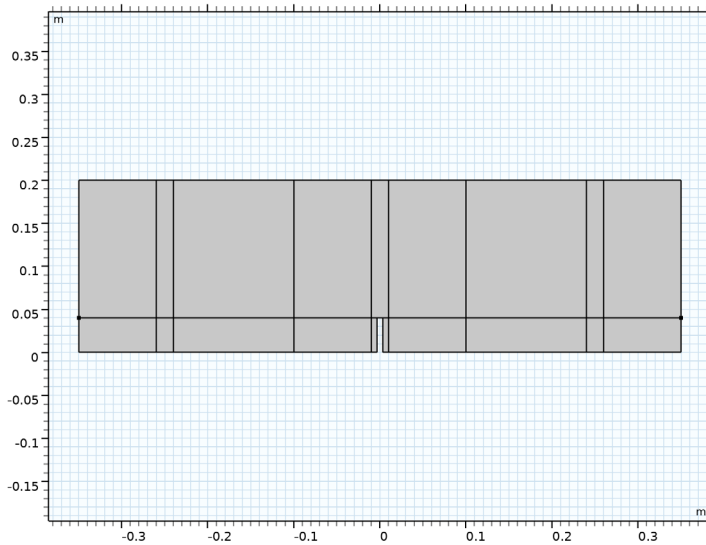
2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

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**.



Most internal edges and vertices are only needed to facilitate a mapped mesh. Use **Virtual Operations** to only make them visible during meshing.

Mesh Control Edges 1 (mce1)

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


2 On the object **fin**, select Boundaries 4, 6, 8, 9, 11, 13, 14, 16, 18, 19, 21, 23, 24, 27, 30, 31, 33, 34, 36, 38, 39, 41, 43, 44, 46, 48, and 49 only.

3 In the **Settings** window for **Mesh Control Edges**, locate the **Input** section.

4 Clear the **Include adjacent vertices** checkbox.

Mesh Control Vertices 1 (mcv1)


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

- 2 On the object **mce1**, select Points 2, 5, 6, 8–14, 20, 21, 23–25, 27, 28, 30, 31, and 33 only.
- 3 In the **Geometry** toolbar, click  **Build All**.


DEFINITIONS

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

Crack Mouth Opening Displacement, 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 Displacement, Right**.
- 4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Point**.
- 5 Select Point 8 only.

Crack Mouth Opening Displacement, Left

- 1 In the **Definitions** toolbar, click  **Nonlocal Couplings** and choose **Average**.
- 2 In the **Settings** window for **Average**, type **Crack Mouth Opening Displacement, 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 6 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	Crack mouth opening displacement


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**, type Damage: Crack Band in the **Label** text field.


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 1scale .
- 5 In the h_{dmg} text field, type $3*1\text{scale}$.


Rigid Connector 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Rigid Connector**.
- 2 Select Boundary 4 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** checkbox.
- 5 Select the **Prescribed in y direction** checkbox.

Rigid Connector 2

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Rigid Connector**.
- 2 Select Boundary 12 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** checkbox.


Boundary Load 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Boundary Load**.
- 2 Select Boundary 6 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 using a **Global Equation**.

- 6 Click the  **Show More Options** button in the **Model Builder** toolbar.
- 7 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics > Equation Contributions**.
- 8 In the tree, select the checkbox 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	$f(u, ut, utt, t)$ (l)	Initial value (u_0) (l)	Initial value (ut_0) (l/s)	Description
load	CMOD - para* maxCMOD	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 displ in the text field.
- 11 In the tree, select **General > Displacement (m)**.
- 12 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	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.21	1	Young's modulus and Poisson's ratio
Density	rho	2400	kg/m ³	Basic
Peak strength	sigmap	3.9 [MPa]	N/m ²	Scalar damage
Fracture energy per area	Gf	85	J/m ²	Scalar damage

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.

Mapped 1

- 1 In the **Model Builder** window, click **Mapped 1**.
- 2 In the **Settings** window for **Mapped**, click to expand the **Control Entities** section.
- 3 From the **Smooth across removed control entities** list, choose **Off**.

Distribution 1

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

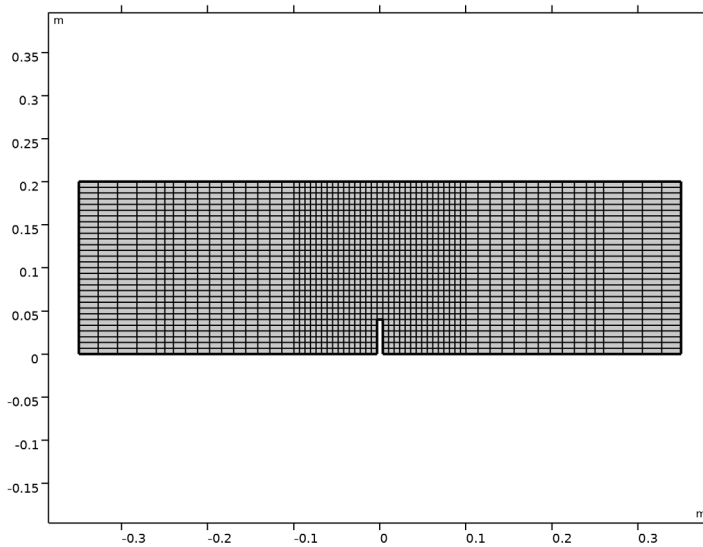
- 2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 3 In the **Number of elements** text field, type 4.
- 4 Select Boundaries 2 and 13 only.

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 4 and 12 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 10 and 20 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** checkbox.
- 4 In the tree, select **Component 1 (comp1) > Solid Mechanics (solid) > Linear Elastic Material 1 > Damage: Implicit Gradient**.
- 5 Right-click and choose **Disable**.
- 6 In the tree, select **Component 1 (comp1) > Solid Mechanics (solid)**.
- 7 From the **Discretization** list, choose **Discretization, Linear**.
- 8 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.
- 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 **Study** toolbar, click **= Compute**.
- Set default units for result presentation.

RESULTS

Preferred Units 1

- 1 In the **Results** toolbar, click  **Configurations** and choose **Preferred Units**.
 - 2 In the **Settings** window for **Preferred Units**, locate the **Units** section.
 - 3 Click **+ Add Physical Quantity**.
 - 4 In the **Physical Quantity** dialog, select **General > Displacement (m)** in the tree.
 - 5 Click **OK**.
 - 6 In the **Settings** window for **Preferred Units**, locate the **Units** section.
 - 7 In the table, enter the following settings:
- | Quantity | Unit | Preferred unit |
|--------------|------|----------------|
| Displacement | m | mm |
- 8 Click **+ Add Physical Quantity**.
 - 9 In the **Physical Quantity** dialog, select **Solid Mechanics > Stress tensor (N/m²)** in the tree.

10 Click **OK**.

11 In the **Settings** window for **Preferred Units**, locate the **Units** section.

12 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Stress tensor	N/m ²	MPa

13 Click  **Add Physical Quantity**.

14 In the **Physical Quantity** dialog, select **General > Force (N)** in the tree.

15 Click **OK**.

16 In the **Settings** window for **Preferred Units**, locate the **Units** section.

17 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Force	N	kN

18 Click  **Apply**.

RESULT TEMPLATES

1 In the **Results** toolbar, click  **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study: Crack Band/Solution 1 (sol1) > Solid Mechanics > Damage (solid)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click  **Result Templates** to close the **Result Templates** window.

RESULTS

Damage, Crack Band

1 In the **Settings** window for **2D Plot Group**, type **Damage, Crack Band** in the **Label** text field.

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

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

Deformation I

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

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


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

4 Select the **Scale factor** checkbox. 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 **Surface 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 Locate the **Title** section. From the **Title type** list, choose **Label**.
- 4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.


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.kappadmgGp - Maximum value of equivalent strain - 1**.
- 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 Locate 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 Right-click and choose **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** checkbox.
- 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** checkbox.
- 8 In the **Study** toolbar, click  **Compute**.

RESULTS



Damage, Crack Band, Strain, Crack Band

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


Damage, Implicit Gradient

- 1 In the **Settings** window for **2D Plot Group**, type Damage, Implicit Gradient in the **Label** text field.
- 2 Locate the **Data** section. From the **Dataset** list, choose **Study: Implicit Gradient/ Solution 2 (sol2)**.
- 3 In the **Damage, Implicit Gradient** toolbar, click  **Plot**.

Strain, Implicit Gradient

- 1 In the **Model Builder** window, under **Results** click **Strain, Crack Band I**.
- 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 **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID 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. From the **Width** list, choose **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 **ID Plot Group**, click to expand the **Title** section.

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

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** checkbox. In the associated text field, type Deflection (mm).

6 Select the **y-axis label** checkbox. In the associated text field, type Load (kN).

7 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 **ID 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 I

- 1 Go to the **Table I** window.
- 2 Click the **Table Graph** button in the window toolbar.

RESULTS


Table Graph 1

Right-click **Results > ID Plot Group 8 > 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 Click to expand the **Legends** section. Select the **Show legends** checkbox.
- 6 From the **Legends** list, choose **Manual**.
- 7 In the table, enter the following settings:


Legends
Measured data

- 8 In the **Load vs. CMOD** toolbar, click  **Plot**.

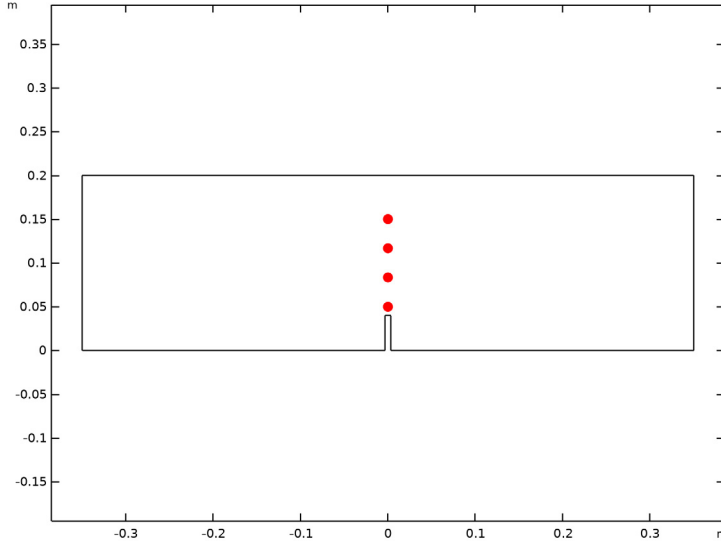
ID Plot Group 8

In the **Model Builder** window, under **Results** right-click **ID Plot Group 8** 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 $\{3 \cdot eSize/2 \ 13 \cdot eSize/2 \ 23 \cdot eSize/2 \ 33 \cdot eSize/2\} + am$.


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  **ID Plot Group**.
- 2 In the **Settings** window for **ID 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.sdGp_{xx} - Stress tensor, damaged, xx-component**.
- 5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

- 6 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**.
- 7 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.
- 8 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.
- 9 From the **Positioning** list, choose **Interpolated**.
- 10 Click to expand the **Legends** section. Select the **Show legends** checkbox.
- 11 Find the **Prefix and suffix** 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 **Prefix and suffix** 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 **ID Plot Group**, locate the **Title** section.
- 3 From the **Title type** list, choose **None**.