Model created in COMSOL Multiphysics 6.4

Brittle Damage in Uniaxial Tension

Introduction

Modeling crack formation in quasi-brittle materials such as concrete is associated with the phenomenon of strain localization due to material softening. In a finite element model, this can cause the solution to be mesh dependent, which is an undesirable property. This tutorial model shows how to avoid this problem by using two different regularization techniques available in COMSOL Multiphysics.

The example describes the axial stretching of a bar, where a damage model is used to account for tensile cracking.

Model Definition

The geometry of the model consists of a 10 cm long bar with height and thickness equal to 2 mm. The analysis is done under plane stress conditions. Due to symmetry, only half the height is modeled.

The bar is considered to be made of a brittle material with the following properties.

- Young's modulus is 30 GPa.
- Poisson's ratio is 0.2.
- The tensile strength is 2 MPa.
- The fracture energy is 60 J/m^2. This is the energy dissipated during the creation of a single crack. The cracking process is modeled using an isotropic damage model with a single damage variable that only considers the tensile failure of the material.

One of the ends of the bar is subjected to an incrementally increasing displacement, while a roller boundary condition is applied at the other end. A roller boundary condition is also applied to the bottom edge of the bar where symmetry is assumed.

To avoid unwanted mesh dependency on the solution during cracking, the damage model needs to be regularized to ensure that a consistent amount of energy is dissipated during mesh refinement or for different discretization orders. Two techniques are available in COMSOL Multiphysics, and these are exemplified 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 damage model.

A mapped mesh with 101x1 elements is used, either with linear or quadratic shape functions for the displacement field. To force a strain localization, a weakness is introduced in the middle of the bar by reducing the tensile strength by 2.5%.

Results and Discussion

Four studies are set up to model different combinations of discretization orders and regularization methods:

1. Quadratic displacement field with the crack band method;
2. Quadratic displacement field with the implicit gradient method;
3. Linear displacement field with the crack band method;
4. Linear displacement field with the implicit gradient method.

Contour plots of the damage variable are shown in Figure 1 for the simulations using the crack band method and in Figure 2 for those using the implicit gradient method. The difference between the two regularization methods is clearly visible by comparing the two figures. For the crack band method, damage is only nonzero in a single element. On the contrary, for the implicit gradient method the damage is distributed over several elements. The effect of this is also seen in Figure 3, where the force versus average strain curves from each study are compared. For studies using the crack band method (Study 1 and 3), the force-deformation curve has the same shape as the exponential strain-softening law prescribed by the damage model. On the other hand, for studies 2 and 4 where the implicit gradient method is used, the shape of the force-deformation curves differs, which is a consequence of the evolution of the damage zone during the incremental stretching of the bar. There are also differences between the quadratic and linear solutions; these discrepancies are due to how well the respective discretization can resolve strains in the damage zone.

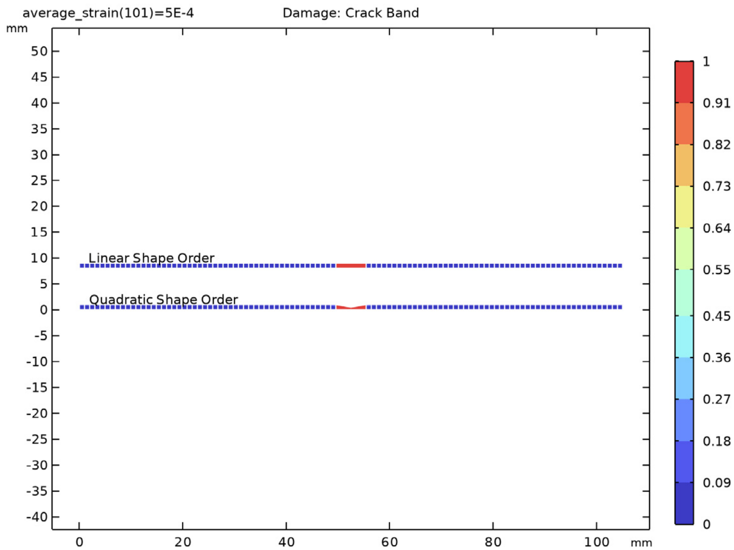

Figure 1: Damage distribution using crack-band regularization.

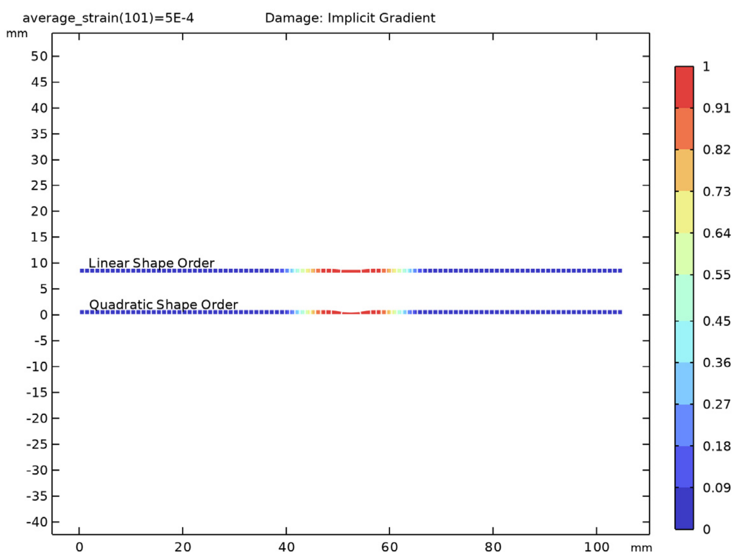

Figure 2: Damage distribution using implicit regularization.

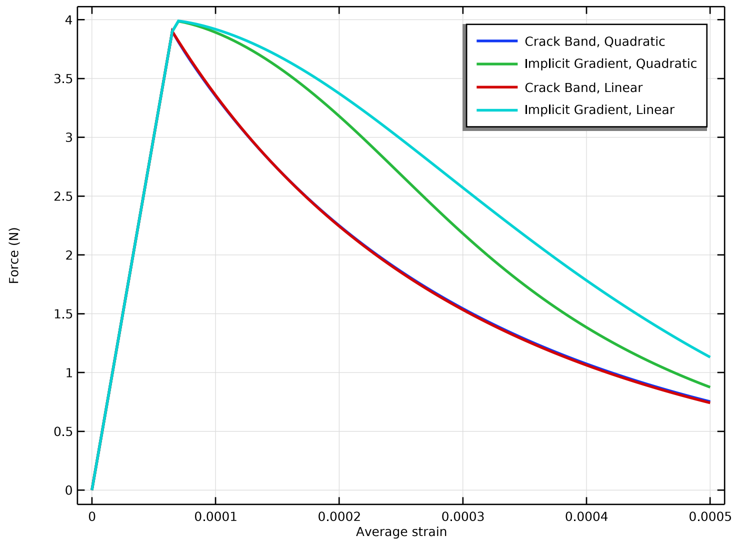

Figure 3: Force versus prescribed end displacement.

Figure 4 shows the stress versus strain and the evolution of the damage variable in the centroid of the middle element. Even though the model is globally supplied with the same fracture energy, the different regularization methods result in significantly different stress versus strain curves. The same conclusion can be drawn from studying the damage evolution curves in Figure 5, where the damage grows significantly faster for the models using the crack band model.

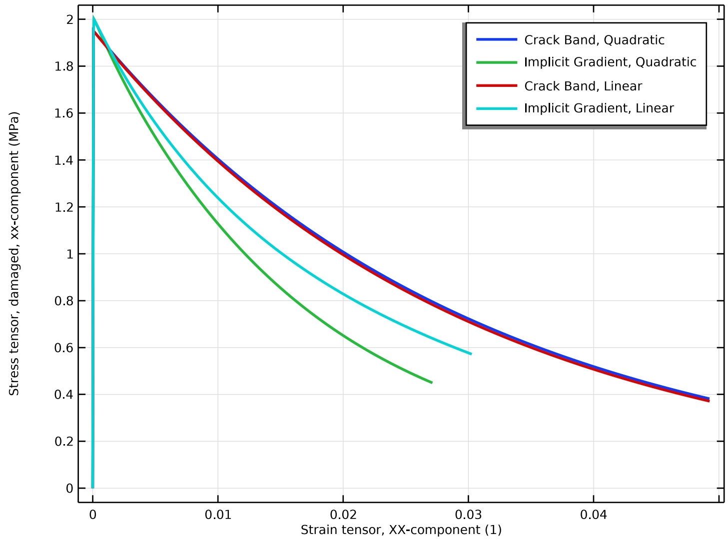

Figure 4: Stress versus strain at the center of the damaged region.

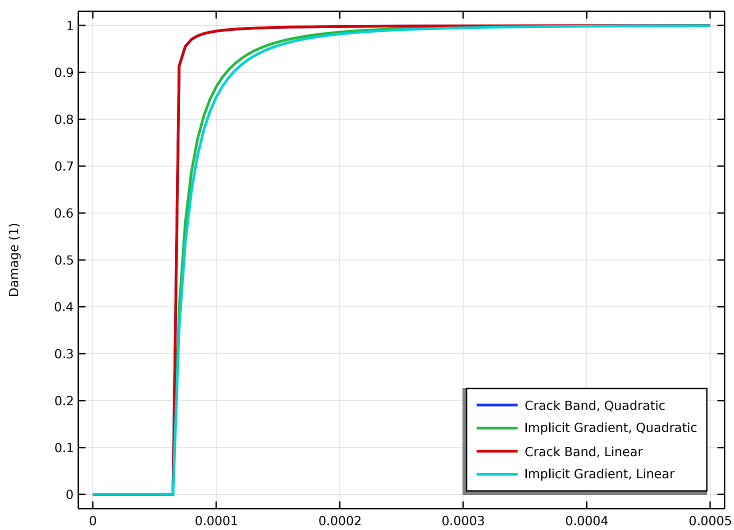

Figure 5: Evolution of the damage variable at the center of the damaged region.

To study the strain localization phenomenon in more detail, the equivalent strain and the damage variable are plotted along the bar in Figure 6 for studies 1 and 2 (quadratic displacement field), and in Figure 7 for studies 3 and 4 (linear displacement field). When the crack band method is used, all damage and deformation is concentrated into a single element. Note, however, that especially for the quadratic displacement field, some unwanted damage also appeared in the adjacent elements. To minimize this unwanted effect, it is in general recommended to use linear interpolation for the displacements when using the crack band method. When comparing this localization behavior to that of the implicit gradient method, the latter exhibits a distributed damage zone. However, it can also be noticed that the strain field is much more narrow. This is in fact also visible in Figure 2 where the middle elements are much more stretched than those toward the edge of the damaged zone. When using the implicit gradient method, the distribution of damage and the localization of strains are clearly much less mesh dependent than when using the crack band method. This can be further investigated by doing a mesh refinement study.

Finally, an important consequence of the different methods is the properties of the displacement field. In Figure 8, the horizontal displacement component is plotted along the middle part of the bar for all four studies. It can be noticed that for both studies 1 and 3 (crack band method), most of the horizontal displacements are localized in the middle element and its derivative is not well defined (see also Figure 6 and Figure 7). However, for studies 2 and 4, where the implicit gradient method is used, both the displacements and their derivatives remain continuous and well defined across the crack.

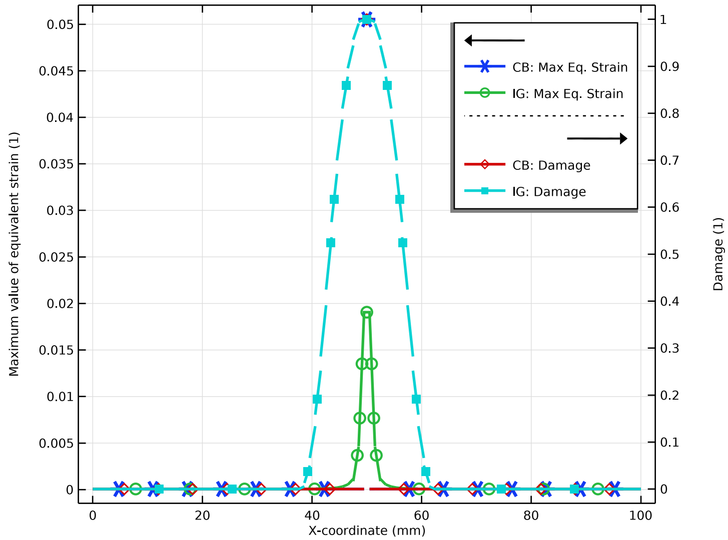

Figure 6: Distribution along the bar of equivalent strain and damage when using quadratic shape functions.

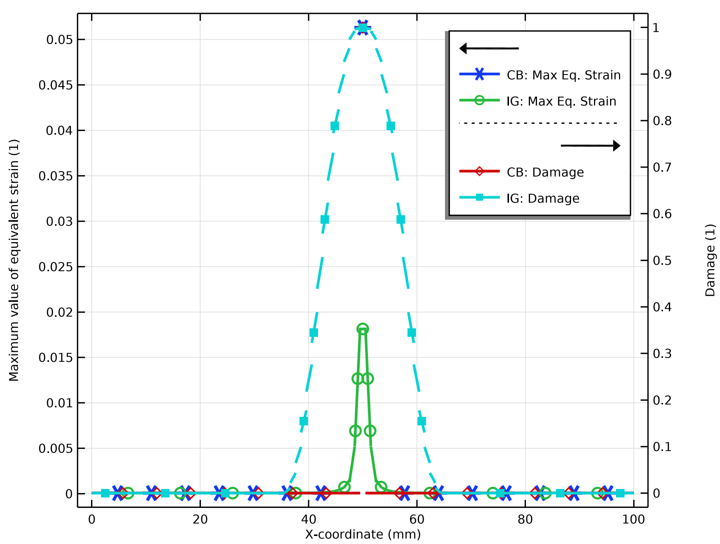

Figure 7: Distribution along the bar of equivalent strain and damage when using linear shape functions.

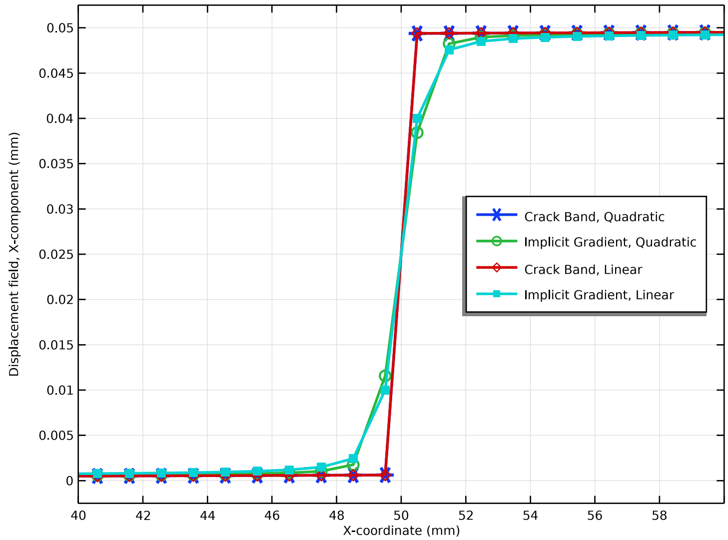

Figure 8: Displacement along the bar.

Notes About the COMSOL implementation

Because the material softens, it would not be possible to perform this analysis with a prescribed load. Using a prescribed displacement, however, solves the problem.

In this example, the material is made slightly weaker at the center of the bar. It can be extremely difficult to make a model like this to converge without this trick. In this particular example, the damage had appeared in the entire bar based on numerical roundoff, and the solution would jump back and forth between iterations. Homogeneous stress states are notoriously difficult to handle with softening material models. Fortunately, this is seldom the case in real life structures. One approach, with some physical interpretation is to modify the material data with a function with a random spatial distribution.

Application Library path: `Nonlinear_Structural_Materials_Module/Damage/damage_test_bar`

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
L	0.1[m]	0.1 m	Bar length
H	L/50	0.002 m	Bar thickness
max_average_strain	5e-4	5E-4	Maximum average strain
average_strain	0	0	Average strain (loading parameter)
lscale	0.001[m]	0.001 m	Length scale

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2 In the **Settings** window for **Geometry**, locate the **Units** section.
3 From the **Length unit** list, choose **mm**.

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 `L`.

4 In the **Height** text field, type `H/2`.

5 Click **Build All Objects**.

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 `H`.

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.

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Damage: Implicit Gradient

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

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`.

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	30[GPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.2	1	Young's modulus and Poisson's ratio
Density	rho	2400	kg/m³	Basic
Peak strength	sigmap	2[MPa]	N/m²	Scalar damage
Fracture energy per area	Gf	60	J/m²	Scalar damage

Make the tensile strength somewhat lower in the center of the bar, so that you can control where the damage is initialized.

Piecewise: Weakening

1 In the **Model Builder** window, expand the **Material 1 (mat1)** node.

2 Right-click **Component 1 (comp1)** > **Materials** > **Material 1 (mat1)** > **Scalar damage (sdmg)** and choose **Functions** > **Piecewise**.

3 In the **Settings** window for **Piecewise**, type `Piecewise: Weakening` in the **Label** text field.

4 In the **Function name** text field, type `weak`.

5 Locate the **Definition** section. Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
0	0.0495	1
0.0495	0.0505	0.975
0.0505	.1	1

6 Locate the **Units** section. In the **Arguments** text field, type `m`.

7 In the **Function** text field, type `1`.

Material 1 (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** > **Material 1 (mat1)** click **Scalar damage (sdmg)**.

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

3 In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Peak strength	sigmap	`2[MPa]*weak(X)`	N/m²	1x1

SOLID MECHANICS (SOLID)

Roller 1

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

2 Select Boundaries 1 and 2 only.

Prescribed Displacement 1

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

2 Select Boundary 4 only.

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

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

5 In the u_{0x} text field, type `L*average_strain`.

MESH 1

Mapped 1

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

Distribution 1

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

2 Select Boundary 2 only.

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

4 In the **Number of elements** text field, type `101`.

CRACK BAND, QUADRATIC

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

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

Step 1: Stationary

1 In the **Model Builder** window, under **Crack Band, Quadratic** click **Step 1: Stationary**.

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

3 Select the **Auxiliary sweep** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
average_strain (Average strain (loading parameter))	range(0,0.01,1)*max_average_strain	

6 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.

7 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Linear Elastic Material 1** > **Damage: Implicit Gradient**.

8 Right-click and choose **Disable**.

Solution 1 (sol1)

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

Allow smaller steps if it is difficult to find a solution.

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

3 In the **Model Builder** window, expand the **Crack Band, Quadratic** > **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** checkbox.

6 In the **Minimum step size** text field, type `1e-5*max_average_strain`.

7 In the **Initial step size** text field, type `0.01*max_average_strain`.

8 In the **Maximum step size** text field, type `0.01*max_average_strain`.

9 In the **Model Builder** window, under **Crack Band, Quadratic** > **Solver Configurations** > **Solution 1 (sol1)** click **Dependent Variables 1**.

10 In the **Settings** window for **Dependent Variables**, locate the **Scaling** section.

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

12 In the **Scale** text field, type `1.0e-4`.

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/m^2)** 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 **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 **Crack Band, Quadratic/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.

Surface 1

1 In the **Model Builder** window, expand the **Damage: Crack Band** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click to expand the **Quality** section.

3 From the **Resolution** list, choose **Custom**.

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** 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 From the **Wireframe color** list, choose **White**.

5 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**.

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
average_strain (Average strain (loading parameter))	range(0,0.01,1)*max_average_strain	

Solution 2 (sol2)

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2** > **Solver Configurations** > **Solution 2 (sol2)** > **Stationary Solver 1** node, then click **Parametric 1**.

4 In the **Settings** window for **Parametric**, locate the **Continuation** section.

5 Select the **Tuning of step size** checkbox.

6 In the **Minimum step size** text field, type `1e-5*max_average_strain`.

7 In the **Initial step size** text field, type `0.01*max_average_strain`.

8 In the **Maximum step size** text field, type `0.01*max_average_strain`.

9 In the **Model Builder** window, under **Study 2** > **Solver Configurations** > **Solution 2 (sol2)** click **Dependent Variables 1**.

10 In the **Settings** window for **Dependent Variables**, locate the **Scaling** section.

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

12 In the **Scale** text field, type `1.0e-4`.

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

14 In the **Settings** window for **Study**, locate the **Study Settings** section.

15 Clear the **Generate default plots** checkbox.

16 In the **Label** text field, type `Implicit Gradient, Quadratic`.

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

Repeat the two studies, but with linear shape order for the elements.

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

19 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics** > **Advanced Physics Options**.

20 Click **OK**.

SOLID MECHANICS (SOLID)

Discretization Linear

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

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

3 From the **Displacement field** list, choose **Linear**.

4 In the **Label** text field, type `Discretization Linear`.

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 3

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
average_strain (Average strain (loading parameter))	`range(0,0.01,1)*max_average_strain`	

5 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.

6 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Linear Elastic Material 1** > **Damage: Implicit Gradient**.

7 Right-click and choose **Disable**.

8 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)**.

9 From the **Discretization** list, choose **Discretization Linear**.

Solution 3 (sol3)

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

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node.

3 In the **Model Builder** window, expand the **Study 3** > **Solver Configurations** > **Solution 3 (sol3)** > **Stationary Solver 1** node, then click **Parametric 1**.

4 In the **Settings** window for **Parametric**, locate the **Continuation** section.

5 Select the **Tuning of step size** checkbox.

6 In the **Minimum step size** text field, type `1e-5*max_average_strain`.

7 In the **Initial step size** text field, type `0.01*max_average_strain`.

8 In the **Maximum step size** text field, type `0.01*max_average_strain`.

9 In the **Model Builder** window, under **Study 3** > **Solver Configurations** > **Solution 3 (sol3)** click **Dependent Variables 1**.

10 In the **Settings** window for **Dependent Variables**, locate the **Scaling** section.

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

12 In the **Scale** text field, type `1.0e-4`.

13 In the **Model Builder** window, click **Study 3**.

14 In the **Settings** window for **Study**, locate the **Study Settings** section.

15 Clear the **Generate default plots** checkbox.

16 In the **Label** text field, type `Crack Band, Linear`.

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

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 4

Step 1: Stationary

1 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

2 Select the **Modify model configuration for study step** checkbox.

3 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)**.

4 From the **Discretization** list, choose **Discretization Linear**.

5 Locate the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.

6 Click **Add**.

7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
average_strain (Average strain (loading parameter))	range(0,0.01,1)* max_average_strain	

Solution 4 (sol4)

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

2 In the **Model Builder** window, expand the **Solution 4 (sol4)** node.

3 In the **Model Builder** window, expand the **Study 4** > **Solver Configurations** > **Solution 4 (sol4)** > **Stationary Solver 1** node, then click **Parametric 1**.

4 In the **Settings** window for **Parametric**, locate the **Continuation** section.

5 Select the **Tuning of step size** checkbox.

6 In the **Minimum step size** text field, type `1e-5*max_average_strain`.

7 In the **Initial step size** text field, type `0.01*max_average_strain`.

8 In the **Maximum step size** text field, type `0.01*max_average_strain`.

9 In the **Model Builder** window, under **Study 4** > **Solver Configurations** > **Solution 4 (sol4)** click **Dependent Variables 1**.

10 In the **Settings** window for **Dependent Variables**, locate the **Scaling** section.

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

12 In the **Scale** text field, type `1.0e-4`.

13 In the **Model Builder** window, click **Study 4**.

14 In the **Settings** window for **Study**, locate the **Study Settings** section.

15 Clear the **Generate default plots** checkbox.

16 In the **Label** text field, type `Implicit Gradient, Linear`.

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

RESULTS

Annotation 1

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

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

3 In the **Text** text field, type `Quadratic Shape Order`.

4 Locate the **Position** section. In the **Y** text field, type `2*H`.

5 Locate the **Coloring and Style** section. Clear the **Show point** checkbox.

6 In the **Damage: Crack Band** toolbar, click **Plot**.

Annotation 1, Mesh 1, Surface 1

1 In the **Model Builder** window, under **Results** > **Damage: Crack Band**, Ctrl-click to select **Surface 1**, **Mesh 1**, and **Annotation 1**.

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

Surface 2

1 In the **Settings** window for **Surface**, locate the **Data** section.

2 From the **Dataset** list, choose **Crack Band, Linear/Solution 3 (sol3)**.

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

Deformation 1

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

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

3 In the **Y-component** text field, type `v+4*H/100`.

Mesh 2

1 In the **Model Builder** window, under **Results** > **Damage: Crack Band** click **Mesh 2**.

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

3 From the **Dataset** list, choose **Crack Band, Linear/Solution 3 (sol3)**.

Deformation 1

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

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

3 In the **Y-component** text field, type `v+4*H/100`.

Annotation 2

1 In the **Model Builder** window, under **Results** > **Damage: Crack Band** click **Annotation 2**.

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

3 In the **Text** text field, type `Linear Shape Order`.

4 Locate the **Position** section. In the **Y** text field, type `6*H`.

5 In the **Damage: Crack Band** toolbar, click **Plot**.

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

Damage: Implicit Gradient

1 In the **Model Builder** window, right-click **Damage: Crack Band** 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 **Implicit Gradient, Quadratic/ Solution 2 (sol2)**.

Surface 2

1 In the **Model Builder** window, expand the **Damage: Implicit Gradient** node, then click **Surface 2**.

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

3 From the **Dataset** list, choose **Implicit Gradient, Linear/Solution 4 (sol4)**.

Mesh 2

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

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

3 From the **Dataset** list, choose **Implicit Gradient, Linear/Solution 4 (sol4)**.

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

Axial Force

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

2 In the **Settings** window for **1D Plot Group**, type `Axial Force` in the **Label** text field.

Global 1

1 In the **Axial Force** toolbar, click **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Reactions** > **Prescribed Displacement 1** > **Reaction force (spatial frame) - N** > **solid.disp1.RFsumx - Reaction force, x-component**.

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

Expression	Unit	Description
solid.disp1.RFsumx	N	Force

4 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

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

6 In the table, enter the following settings:

Legends
Crack Band, Quadratic

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 **Implicit Gradient, Quadratic/Solution 2 (sol2)**.

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

Legends
Implicit Gradient, Quadratic

Global 3

1 Right-click **Global 2** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Crack Band, Linear/Solution 3 (sol3)**.

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

Legends
Crack Band, Linear

Global 4

1 Right-click **Global 3** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Implicit Gradient, Linear/Solution 4 (sol4)**.

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

Legends
Implicit Gradient, Linear

Axial Force

1 In the **Model Builder** window, click **Axial Force**.

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.

5 Select the **x-axis label** checkbox. In the associated text field, type `Average strain`.

6 In the **Axial Force** toolbar, click **Plot**.

Cut Point: Study 1

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

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

3 In the **X** text field, type `L/2`.

4 In the **Y** text field, type `0`.

5 In the **Label** text field, type `Cut Point: Study 1`.

Cut Point: Study 2

1 Right-click **Cut Point: Study 1** and choose **Duplicate**.

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

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

Cut Point: Study 3

1 Right-click **Cut Point: Study 2** and choose **Duplicate**.

2 In the **Settings** window for **Cut Point 2D**, type `Cut Point: Study 3` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Crack Band, Linear/ Solution 3 (sol3)**.

Cut Point: Study 4

1 Right-click **Cut Point: Study 3** and choose **Duplicate**.

2 In the **Settings** window for **Cut Point 2D**, type `Cut Point: Study 4` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Implicit Gradient, Linear/ Solution 4 (sol4)**.

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.

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

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 Point: 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.sdGpxx - 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 In the **Damaged Stress vs. Strain** toolbar, click **Plot**.

8 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

9 Click to expand the **Legends** section. Select the **Show legends** checkbox.

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

11 In the table, enter the following settings:

Legends
Crack Band, Quadratic

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 Point: Study 2**.

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

Legends
Implicit Gradient, Quadratic

Point Graph 3

1 Right-click **Point Graph 2** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Cut Point: Study 3**.

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

Legends
Crack Band, Linear

Point Graph 4

1 Right-click **Point Graph 3** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Cut Point: Study 4**.

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

Legends
`Implicit Gradient, Linear`

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

Damage Evolution

1 In the **Model Builder** window, right-click **Damaged Stress vs. Strain** and choose **Duplicate**.

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

3 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Point Graph 1

1 In the **Model Builder** window, expand the **Damage Evolution** node, then click **Point Graph 1**.

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

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

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `solid.dmg`.

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

Point Graph 2

1 In the **Model Builder** window, click **Point Graph 2**.

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

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

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

Point Graph 3

1 In the **Model Builder** window, click **Point Graph 3**.

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

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

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

Point Graph 4

1 In the **Model Builder** window, click **Point Graph 4**.

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

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

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

5 In the **Damage Evolution** toolbar, click **Plot**.

Localization, Quadratic

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

2 In the **Settings** window for **1D Plot Group**, type `Localization, Quadratic` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (average_strain)** list, choose **Last**.

4 Locate the **Title** section. From the **Title type** list, choose **None**.

Line Graph 1

1 Right-click **Localization, Quadratic** and choose **Line Graph**.

2 Select Boundary 2 only.

3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Damage** > **solid.kappadmgGp - Maximum value of equivalent strain - 1**.

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

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

6 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

8 From the **Positioning** list, choose **Interpolated**.

9 In the **Number** text field, type `15`.

10 Click to expand the **Quality** section. From the **Evaluation settings** list, choose **Manual**.

11 From the **Resolution** list, choose **No refinement**.

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

13 Click to expand the **Legends** section. Select the **Show legends** checkbox.

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

15 In the table, enter the following settings:

Legends
`CB: Max Eq. Strain`

16 In the **Localization, Quadratic** toolbar, click **Plot**.

Line Graph 2

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

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

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

4 From the **Parameter selection (average_strain)** list, choose **Last**.

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

Legends
IG: Max Eq. Strain

6 In the **Localization, Quadratic** toolbar, click **Plot**.

Line Graph 1, Line Graph 2

1 In the **Model Builder** window, under **Results** > **Localization, Quadratic**, Ctrl-click to select **Line Graph 1** and **Line Graph 2**.

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

Line Graph 3

1 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Damage** > **solid.dmgGp - Damage - 1**.

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

Legends
CB: Damage

Line Graph 4

1 In the **Model Builder** window, click **Line Graph 4**.

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

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

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

Legends
IG: Damage

Localization, Quadratic

1 In the **Model Builder** window, click **Localization, Quadratic**.

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

3 Select the **Two y-axes** checkbox.

4 In the table, select the **Plot on secondary y-axis** checkboxes for **Line Graph 3** and **Line Graph 4**.

5 In the **Localization, Quadratic** toolbar, click **Plot**.

Localization, Linear

1 Right-click **Localization, Quadratic** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Localization, Linear` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Crack Band, Linear/ Solution 3 (sol3)**.

Line Graph 2

1 In the **Model Builder** window, expand the **Localization, Linear** node, then click **Line Graph 2**.

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

3 From the **Dataset** list, choose **Implicit Gradient, Linear/Solution 4 (sol4)**.

Line Graph 4

1 In the **Model Builder** window, click **Line Graph 4**.

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

3 From the **Dataset** list, choose **Implicit Gradient, Linear/Solution 4 (sol4)**.

4 In the **Localization, Linear** toolbar, click **Plot**.

Horizontal Displacement

1 In the **Model Builder** window, right-click **Localization, Quadratic** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Horizontal Displacement` in the **Label** text field.

3 Locate the **Plot Settings** section. Clear the **Two y-axes** checkbox.

Line Graph 1

1 In the **Model Builder** window, expand the **Horizontal Displacement** node, then click **Line Graph 1**.

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

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

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

Legends
`Crack Band, Quadratic`

5 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Positioning** list, choose **In evaluation points**.

Line Graph 2

1 In the **Model Builder** window, click **Line Graph 2**.

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

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

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

Legends
`Implicit Gradient, Quadratic`

5 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Positioning** list, choose **In evaluation points**.

Line Graph 3

1 In the **Model Builder** window, click **Line Graph 3**.

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

3 From the **Dataset** list, choose **Crack Band, Linear/Solution 3 (sol3)**.

4 From the **Parameter selection (average_strain)** list, choose **Last**.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `u`.

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

Legends
`Crack Band, Linear`

7 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Positioning** list, choose **In evaluation points**.

Line Graph 4

1 In the **Model Builder** window, click **Line Graph 4**.

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

3 From the **Dataset** list, choose **Implicit Gradient, Linear/Solution 4 (sol4)**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `u`.

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

Legends
`Implicit Gradient, Linear`

6 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Positioning** list, choose **In evaluation points**.

Horizontal Displacement

1 In the **Model Builder** window, click **Horizontal Displacement**.

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

3 Select the **Manual axis limits** checkbox.

4 In the **x minimum** text field, type 40.

5 In the **x maximum** text field, type 60.

6 Locate the **Legend** section. From the **Position** list, choose **Middle right**.

7 In the **Horizontal Displacement** toolbar, click **Plot**.