Failure of a Concrete Beam Using Coupled Damage–Plasticity

Introduction

Failure of reinforced concrete structures involves advanced material behavior and interaction between materials. This example demonstrates how to model various stages of the failure of a reinforced concrete beam by including a coupled damage–plasticity material model for the concrete, metal plasticity for the rebars, and a nonlinear bond-slip law for the interaction between concrete and the rebar.

Model Definition

The example aims to model a simply supported reinforced concrete beam subjected to two distributed loads. The total length of the beam is 4.5 m, and the cross section is 200 mm wide and 300 mm high. A geometrical reinforcement content equal to 4.5% is used, with the rebar placed 50 mm from the bottom of the beam

The geometry is defined in 2D with an assumption of plane stress, and a symmetry plane is utilized so that only half of the beam is modeled. Figure 3 shows the geometry and a schematic description of the boundary conditions. Since a 2D geometry is used, the entire reinforcement content is assumed to be lumped in the a single edge, where truss elements are inserted with the appropriate cross-section area.

Figure 1: Model geometry and boundary conditions.

Properties of the concrete material are given in Table 1. All other parameters necessary to define the coupled damage–plasticity model are set to their default. The reinforcement is defined as an elastoplastic material with linear isotropic hardening. Table 2 lists its material properties.

TABLE 1: MATERIAL PROPERTIES OF THE CONCRETE.

Material property	Symbol	Value
Young's modulus	E	25 GPa
Poisson's ratio	ν	0.2
Compressive strength	σ_{uc}	30 MPa
Tensile strength	σ_{ut}	2 MPa
Fracture energy	G_{ft}	220 J/m^2

TABLE 2: MATERIAL PROPERTIES OF THE REINFORCEMENT.

Material property	Symbol	Value
Young's modulus	E	208 GPa
Poisson's ratio	ν	0.3
Initial yield stress	σ_{ys}	500 MPa
Isotropic hardening modulus	E_{Tiso}	1 GPa

To accurately describe the failure process of reinforced concrete structures, it is important to also consider the interaction between concrete and rebar, often referred to as a bond-slip law. The interaction can be severely nonlinear and have a large impact on how cracks form and on the overall failure mechanism. In this example, the interaction is modeled by inserting distributed springs between the truss mesh elements of the rebar and the solid mesh elements of the concrete. In order to include the bond-slip law, a friction model is added to the otherwise elastic springs in the axial direction of the rebar. Any bond-slip law can here be included by adding a user-defined hardening model to the cohesion that determines when slip occurs. Here a hardening model according to Figure 3 is

implemented, which is similar to what can be found in many model and design codes for concrete structures.
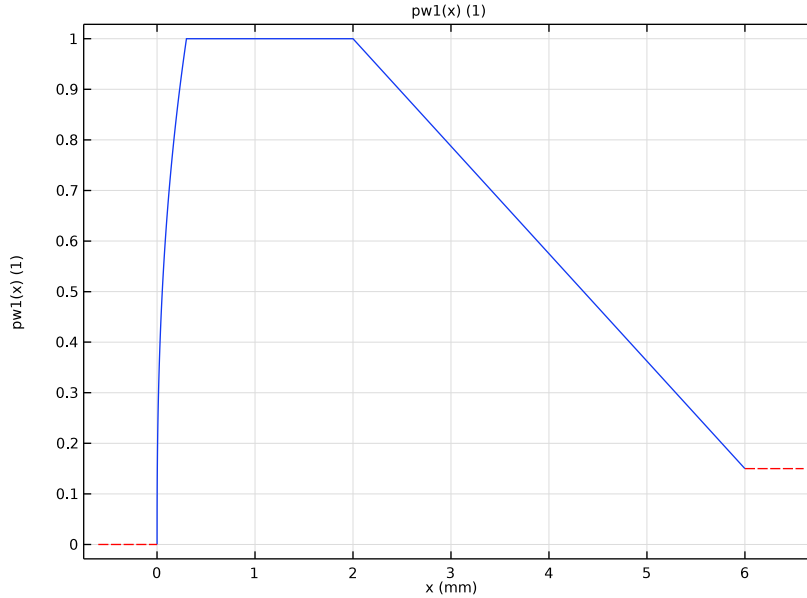
Figure 2: Hardening model used for the bond-slip law. Note that the y-axis is a normalized measure of the bond strength.

The analysis is set up as a parametric sweep such that the load is incrementally increased to capture the entire failure process.

Results and Discussion

The total load applied to the beam is monitored by a **Probe** that stores its value at every step taken by the solver to show extra detail. Figure 3 shows the resulting curve. Here one can clearly distinguish three different stages:

- Up to approximately 10 kN, the beam basically behaves linearly.

- Above 10 kN, the stiffness of the beam changes, which is an indication that cracks have started to propagate. In fact, each new crack can be identified by the small dip or change in slope in the load-displacement curve; in total there are seven cracks.

- When the load reaches around 22 kN, the stiffness of the beam changes dramatically. This corresponds to yielding of the reinforcement. A small increase of the load now leads to a large increase of the vertical displacement of the beam. If one were to continue

the analysis the beam would eventually fail, either to crushing of the concrete in the upper part of the beam or rupture of the rebars if the model included such a failure mechanism.
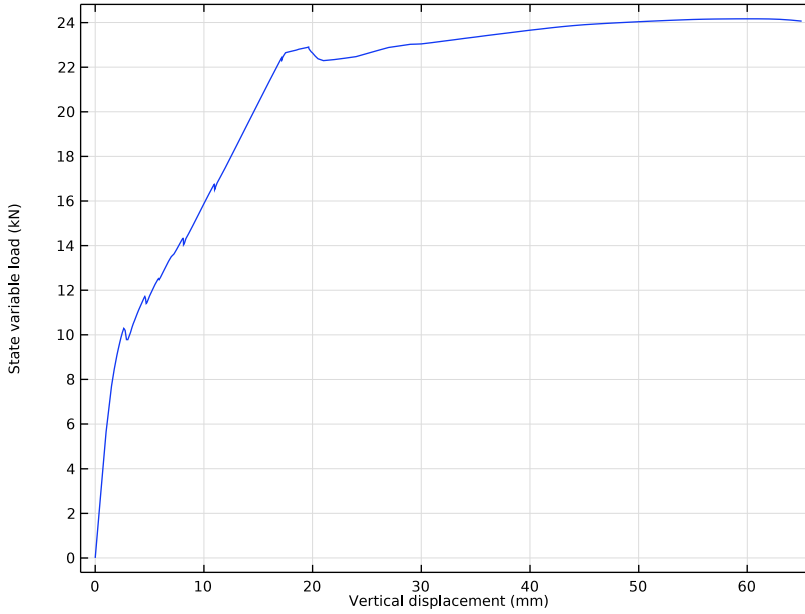
Figure 3: Load versus displacement curve of the reinforced concrete beam.

The seven cracks previously mentioned can be seen in Figure 4, which shows an estimate of the crack width at the last step of the analysis. Here one can clearly see that most of the deformation is localized to the two cracks near the midsection, where the crack width is close to 5 mm. Observe, however, that the stiffness is lost in all cracks as indicated by Figure 5, which shows the damage. In Figure 5, it can also be observed that (compressive) damage has started to develop in the upper part of the beam, and the beam is probably close to a complete failure at this point.

The reason why most deformation is localized in two of the cracks is due to yielding of the reinforcement. As shown in Figure 6, the plastic deformation is concentrated in two bands close to the midsection. Here, an effect of the bond-slip model is seen since the plastic bands clearly extend to more than the single mesh element of the crack. The computed slip is shown in Figure 7.

disp(66)=65 mm   Surface: Crack Width (mm)
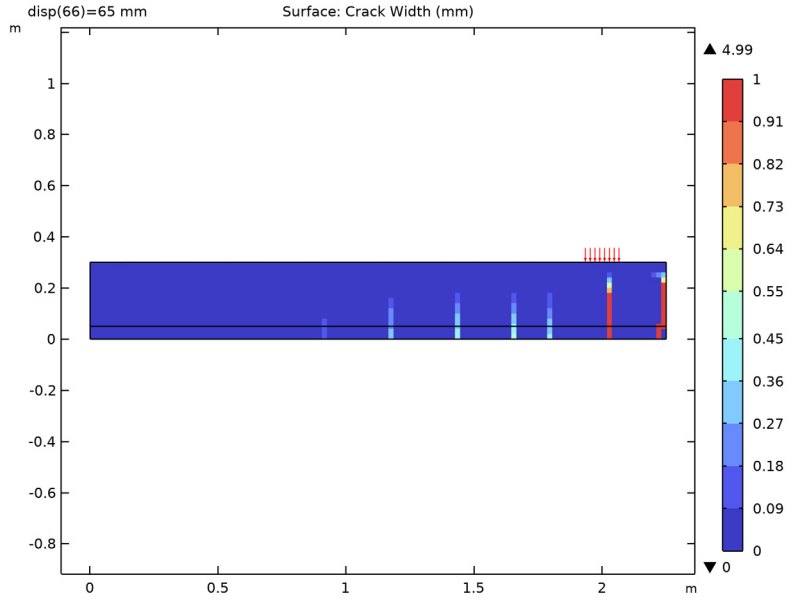
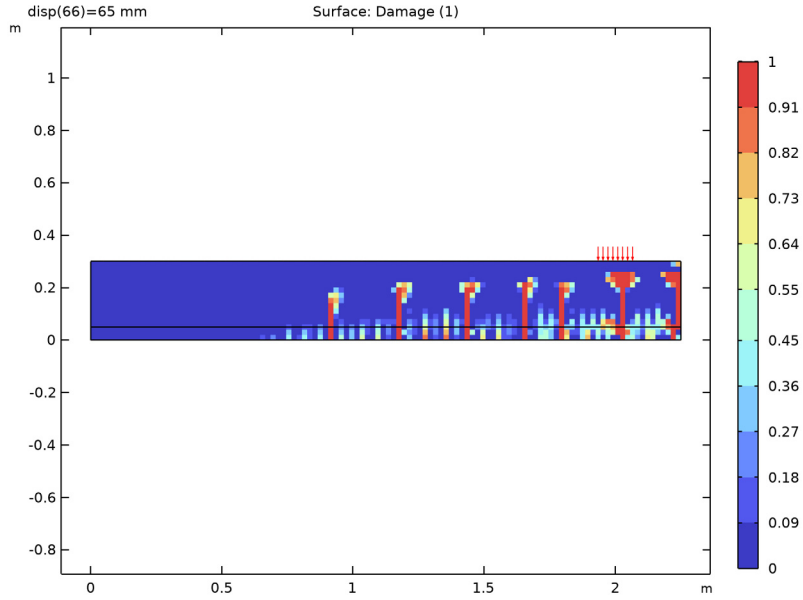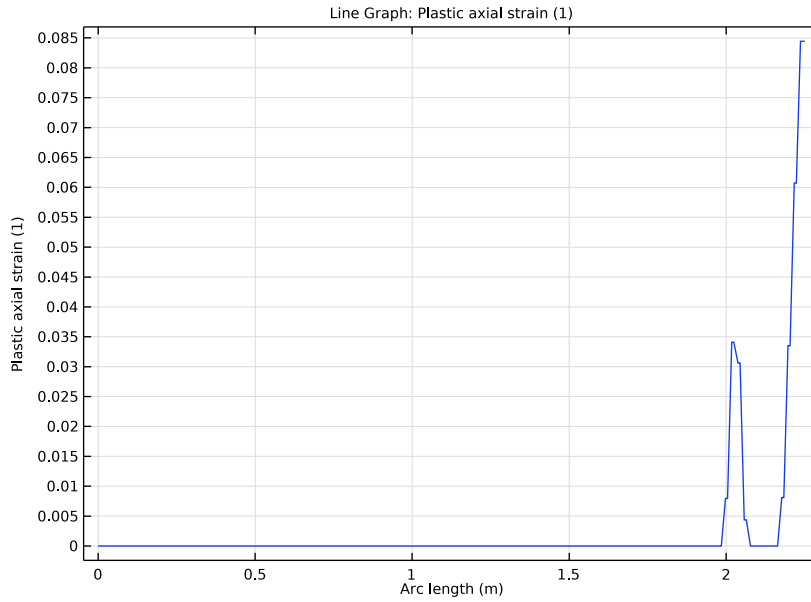*Figure 4: Estimated crack width at the last step of the analysis.*

Figure 5: Damage at the last step of the analysis.

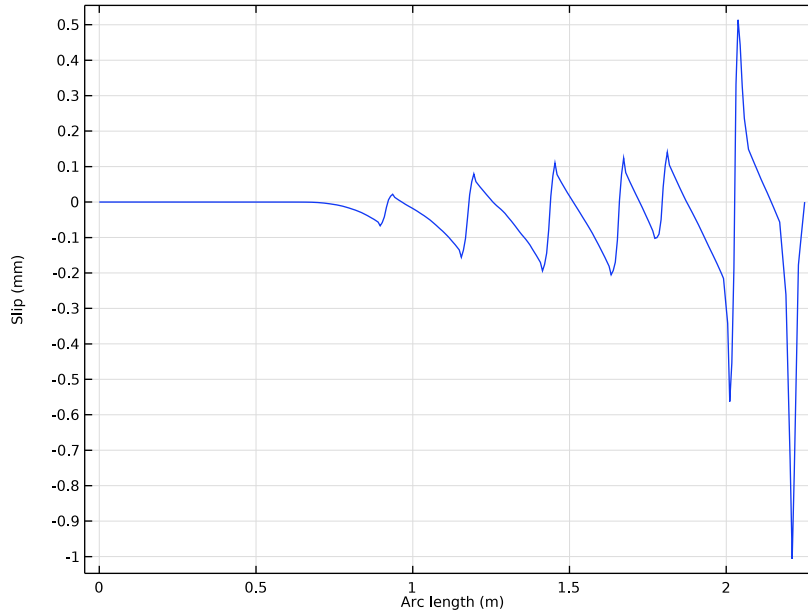*Figure 6: Plastic deformation in the reinforcement at the last step of the analysis.*

Figure 7: Slip between concrete and the rebar at the last step of the analysis.

Notes About the COMSOL Implementation

Failure of concrete and reinforced concrete structures is a very nonlinear process, and we cannot expect any boundary loads to be monotonically increasing. Hence, any distributed load needs to be indirectly controlled by some other measure. Here, the vertical displacement at the midsection of the beam is used as a control variable in a **Global equation**. This algebraic equation is used to compute a load proportionality factor.

When the crack-band method is used as a spatial regularization method for model softening, symmetry conditions need special attention. In principle, the crack-band width is doubled for mesh elements next to a symmetry plane. In order not to overestimate the dissipated energy during softening, it is necessary to compensate for this fact. Here, this is done by reducing the fracture energy G_{ft} by a factor 0.5 for the relevant mesh elements.

The coupled damage–plasticity model requires local computations at each Gauss point during the assembly process, which is expensive. By using **Reduced Integration**, the number of Gauss points are reduced by approximately a factor three for the given displacement shape-function order and mesh-element type. This setting speeds up the computation time significantly.

Application Library path: `Geomechanics_Module/Concrete/concrete_beam_cdpm`

Modeling Instructions

From the **File** menu, choose **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 In the **Select Physics** tree, select **Structural Mechanics>Truss (truss)**.

5 Click **Add**.

6 In the **Displacement field (m)** text field, type u.

7 Click ➔ **Study**.

8 In the **Select Study** tree, select **General Studies>Stationary**.

9 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
disp	0[mm]	0 m	
meshSize	20[mm]	0.02 m	

Geometry Parameters

1 In the **Home** toolbar, click ⊓ᵢ **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type Geometry Parameters in the **Label** text field.

3 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
length1	250[mm]	0.25 m	
length2	1750[mm]	1.75 m	
length3	500[mm]	0.5 m	
heigth	300[mm]	0.3 m	
width	200[mm]	0.2 m	
bcWidth	150[mm]	0.15 m	
rebarPosition	50[mm]	0.05 m	

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 length1+length2+length3/2.

4 In the **Height** text field, type heigth.

5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	length1-bcWidth/2
Layer 2	bcWidth
Layer 3	length2-bcWidth
Layer 4	bcWidth

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

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

Polygon 1 (pol1)

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

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

3 In the table, enter the following settings:

x (m)	y (m)
0	rebarPosition
length1+length2+length3/2	rebarPosition

Form Union (fin)

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

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

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

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, 7, 10, and 13 only.

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

4 Clear the **Include adjacent vertices** check box.

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 4, 6, 7, and 9 only.

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

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 width.

It is recommended to use a linear displacement field when the model includes a material model based on damage.

5 Click to expand the **Discretization** section. From the **Displacement field** list, choose **Linear**.

Linear Elastic Material 1

The local computations at Gauss points during assembly are expensive for the coupled damage-plasticity concrete model. Using a reduced integration scheme will reduce the overall simulation time.

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

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

3 Select the **Reduced integration** check box.

Concrete 1

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

2 In the **Settings** window for **Concrete**, locate the **Concrete Model** section.

3 From the **Material model** list, choose **Coupled damage-plasticity**.

Symmetry 1

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

2 Select Boundary 8 only.

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 y direction** check box.

The distributed loads will be added by indirect displacement control, forcing the displacement of a control point to be monotonically increasing.

DEFINITIONS

Integration 1 (intop1)

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

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

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

4 Select Point 8 only.

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

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

7 Click **OK**.

SOLID MECHANICS (SOLID)

Global Equations 1

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

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

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
load	intop1(v)+ disp	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 Disp in the text field.

12 Click ▼ **Filter**.

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

14 Click **OK**.

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

-load/2	y

Boundary Load 2

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

2 Select Boundaries 3, 6, and 7 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 $\mathbf{F}_{\text{tot}}$ vector as

-load/2	y

TRUSS (TRUSS)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Truss (truss)**.

2 Select Boundary 9 only.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Truss (truss)** click
Linear Elastic Material 1.

Plasticity 1

In the **Physics** toolbar, click ⬚ **Attributes** and choose **Plasticity**.

Cross-Section Data 1

The reinforced beam is assumed to have a geometrical reinforcement content equal to
0.4%.

1 In the **Model Builder** window, under **Component 1 (comp1)>Truss (truss)** click **Cross-Section Data 1**.

2 In the **Settings** window for **Cross-Section Data**, locate the **Cross-Section Definition** section.

3 In the *A* text field, type heigth*width*0.4[%].

Symmetry 1

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

2 Select Point 10 only.

MULTIPHYSICS

Embedded Reinforcement 1 (ere1)

1 In the **Physics** toolbar, click 🏗 **Multiphysics Couplings** and choose **Global>
Embedded Reinforcement**.

2 In the **Settings** window for **Embedded Reinforcement**, locate the **Edge Selection,
Embedded Structure** section.

3 Click to select the ▭ **Activate Selection** toggle button.

4 Select Boundary 9 only.

The default spring stiffness leads to an ill-condition system of equations. Use a lower value to avoid this.

5 Locate the **Connection Settings** section. From the **Connection type** list, choose **Spring constant per unit surface area**.

6 In the k_a text field, type `((1e-1*ere1.Eequ)*ere1.perimeter)/(h^2)`.

7 In the k_t text field, type `((1e1*ere1.Eequ)*ere1.perimeter)/(h^2)`.

Add a bond-slip law. A typical bond-slip curve for concrete is most easily implemented as a user-defined hardening model using a **Piecewise** function.

8 Find the **Bond slip model** subsection. From the list, choose **Friction**.

9 In the c_0 text field, type `1[MPa]`.

DEFINITIONS

Piecewise 1 (pw1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Local>Piecewise**.

2 In the **Settings** window for **Piecewise**, locate the **Units** section.

3 In the **Arguments** text field, type `mm`.

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

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

Start	End	Function
0	0.3	`max((x/0.3),eps)^0.4`
0.3	2	`1`
2	6	`1-(1-0.15)*(x-2)/(6-2)`

6 Click ▶ **Plot**.

MULTIPHYSICS

Embedded Reinforcement 1 (ere1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Multiphysics** click **Embedded Reinforcement 1 (ere1)**.

2 In the **Settings** window for **Embedded Reinforcement**, locate the **Connection Settings** section.

3 Find the **Bond slip model** subsection. From the **Hardening model** list, choose **User defined**.

4 In the ch text field, type `5[MPa]*pw1(ere1.upe)`.

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	25[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	0	kg/m³	Basic
Tensile strength	sigmaut	2[MPa]	Pa	Yield stress parameters
Compressive strength	sigmauc	30[MPa]	Pa	Yield stress parameters
Tensile fracture energy	Gft	220[J/m^2]	J/m²	Fracture parameters

The crack-band width is doubled for mesh elements next to a symmetry plane. Reduce the fracture energy in such mesh elements to compensate.

DEFINITIONS

Analytic 1 (an1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Local>Analytic**.

2 In the **Settings** window for **Analytic**, locate the **Definition** section.

3 In the **Expression** text field, type `if(x>=length1+length2+length3/2-1.05* meshSize/2,1/2,1)`.

4 Locate the **Units** section. In the **Function** text field, type `1`.

5 In the table, enter the following settings:

Argument	Unit
x	m

MATERIALS

Material 1 (mat1)

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

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
Tensile fracture energy	Gft	220[J/m^2]* an1(centroid(X))	J/m²	Fracture parameters

Material 2 (mat2)

1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

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

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

4 Select Boundary 9 only.

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	208[GPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	I	Young's modulus and Poisson's ratio
Density	rho	0	kg/m³	Basic
Initial yield stress	sigmags	500[MPa]	Pa	Elastoplastic material model
Isotropic tangent modulus	Et	1[GPa]	Pa	Elastoplastic material model

MESH 1

Mapped 1

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

Edge 1

1 In the **Mesh** toolbar, click △ **Edge**.

2 Select Boundary 9 only.

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 meshSize.

5 Click ▦ **Build All**.

STUDY 1

Step 1: Stationary

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

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

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

4 Click ＋ **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
disp	range(0,1,65)	mm

Show the default solver in order to make some modifications to the automatic suggestion.

Solution 1 (sol1)

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

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

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

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

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

6 In the **Initial step size** text field, type 0.5/1000.

7 In the **Minimum step size** text field, type 1e-8.

8 In the **Maximum step size** text field, type 2/1000.

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

10 Locate the **General** section. From the **On error** list, choose **Skip parameter step**.

11 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)> Stationary Solver 1** click **Fully Coupled 1**.

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

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

14 In the **Maximum number of iterations** text field, type 12.

15 In the **Tolerance factor** text field, type 1.0e-1.

Add a **Global Variable Probe** to monitor the solution while solving.

DEFINITIONS

Global Variable Probe 1 (var1)

1 In the **Definitions** toolbar, click **Probes** and choose **Global Variable Probe**.

2 In the **Settings** window for **Global Variable Probe**, locate the **Expression** section.

3 In the **Expression** text field, type load.

4 From the **Table and plot unit** list, choose **kN**.

Also show the default plots and add a **Damage** predefined plot to further monitor the solution while solving.

STUDY 1

In the **Study** toolbar, click **Show Default Plots**.

ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Windows** and choose **Add Predefined Plot**.

2 Go to the **Add Predefined Plot** window.

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

4 Click **Add Plot** in the window toolbar.

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, click to expand the **Results While Solving** section.

3 Select the **Plot** check box.

4 From the **Plot group** list, choose **Damage (solid)**.

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

6 In the **Home** toolbar, click ▬ **Compute**.

RESULTS

Probe Plot Group 1

1 In the **Model Builder** window, under **Results** click **Probe Plot Group 1**.

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

3 Select the **x-axis label** check box. In the associated text field, type Vertical displacement (mm).

4 Locate the **Legend** section. Clear the **Show legends** check box.

5 In the **Probe Plot Group 1** toolbar, click ⊡ **Plot**.

The default stress plot shows the undamaged von Mises stress, it is of greater interest to show a component of the damaged stress tensor.

Surface 1

1 In the **Model Builder** window, expand the **Results>Stress (solid)** 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>Stress tensor, damaged (spatial frame) - N/m²>solid.sdGpxx - Stress tensor, damaged, xx-component**.

3 Locate the **Expression** section. From the **Unit** list, choose **MPa**.

4 Locate the **Coloring and Style** section. Click ┃┃┃ **Change Color Table**.

5 In the **Color Table** dialog box, select **Rainbow>Rainbow** in the tree.

6 Click **OK**.

Arrow Line 1

1 In the **Model Builder** window, right-click **Damage (solid)** and choose **Arrow Line**.

2 In the **Settings** window for **Arrow Line**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Solid Mechanics>Load>solid.F_Ax,solid.F_Ay - Load (spatial frame)**.

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

4 Locate the **Arrow Positioning** section. From the **Placement** list, choose **Gauss points**.

5 Locate the **Coloring and Style** section. From the **Arrow base** list, choose **Head**.

6 In the **Damage (solid)** toolbar, click ⬚ **Plot**.

Create a plot that shows an estimate of the crack width. Note that this variable should be evaluated at Gauss points.

Damage (solid)
Right-click **Damage (solid)** and choose **Duplicate**.

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

2 In the **Settings** window for **2D Plot Group**, type Crack Width (solid) in the **Label** text field.

3 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

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

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

3 In the **Expression** text field, type solid.gpeval(solid.lemm1.cm1.wdmgt).

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

5 In the **Description** text field, type Crack Width.

6 Click to expand the **Range** section. Select the **Manual color range** check box.

7 In the **Maximum** text field, type 1.

8 In the **Crack Width (solid)** toolbar, click ⬚ **Plot**.

Create a plot that shows the plastic strain in the reinforcement.

Plastic Strain (truss)
1 In the **Home** toolbar, click ⬚ **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type Plastic Strain (truss) in the **Label** text field.

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

Line Graph 1

1 Right-click **Plastic Strain (truss)** and choose **Line Graph**.

2 Select Boundary 9 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)>Truss>Strain> truss.epnGp - Plastic axial strain - 1**.

4 In the **Plastic Strain (truss)** toolbar, click ▣ **Plot**.

Create a plot that shows the slip between the reinforcement and the concrete. Note that this variable should be evaluated at Gauss points.

Plastic Strain (truss)

In the **Model Builder** window, right-click **Plastic Strain (truss)** and choose **Duplicate**.

Slip

1 In the **Model Builder** window, click **Plastic Strain (truss) 1**.

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

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

4 In the **Label** text field, type Slip.

5 Locate the **Plot Settings** section.

6 Select the **y-axis label** check box. In the associated text field, type Slip (mm).

Line Graph 1

1 In the **Model Builder** window, expand the **Slip** 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 gpeval(2,ere1.un).

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

5 In the **Slip** toolbar, click ▣ **Plot**.