Concrete Beam With Reinforcement Bars

Introduction

Concrete structures almost always contain reinforcements in the shape of steel bars (*rebars*). In COMSOL Multiphysics, individual rebars can be modeled by adding a Truss interface to the Solid Mechanics interface used for the concrete beam. The solid mesh for the concrete and the rebar mesh can be independent of each other, since the displacements are mapped from within the solids onto the rebars.

Model Definition

This model shows how to include steel reinforcement that is much smaller than the geometrical dimensions of the concrete structure. The truss interface is used to model the steel reinforcements instead of a 3D solid. This removes the necessity to model the bars with small mesh which saves computational time. The geometry of the concrete beam is given in Figure 1.

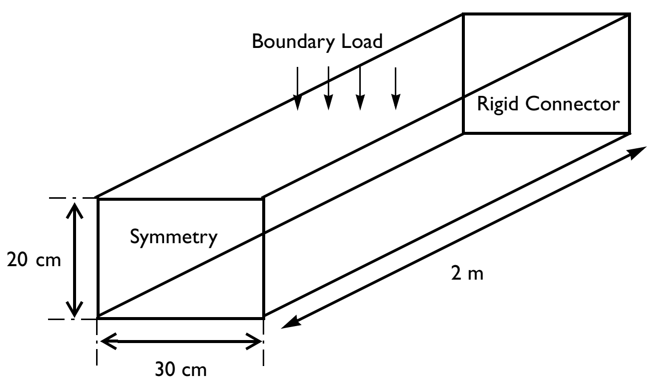

Figure 1: The concrete beam is 30 cm in width and 20 cm in height. Its length is 4 meters, but due to symmetry, half of its length is modeled.

In the model most dimensions such as height, width, and length of the concrete structure are parametrized. The number of rebars is also given by parameters, which makes it possible to change the number of layers and rebars per layer. In this example, six steel bars 10 mm in diameter are placed in four parallel layers along the concrete beam. See Figure 2.

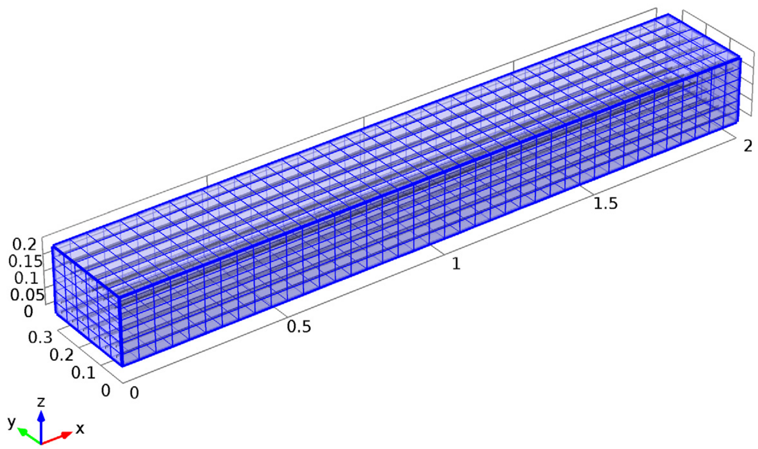

Figure 2: A mapped mesh of 6 by 6 elements is swept through the length of the concrete beam. One hundred elements are used for each reinforcement bar.

In this model the effect of the gravity load is simulated. Moreover a deflection due to a vertical boundary load is ramped up to 20 kN/m^2 by means of a parametric sweep.

Results and Discussion

Three different studies are done. The first study models the concrete beam as an isotropic elastic material, the second study adds the reinforcements bars, and the third study includes the effect of plastic deformation in the concrete, modeled using the Ottosen criterion. Figure 3 shows the comparison for the vertical displacement of the three studies.

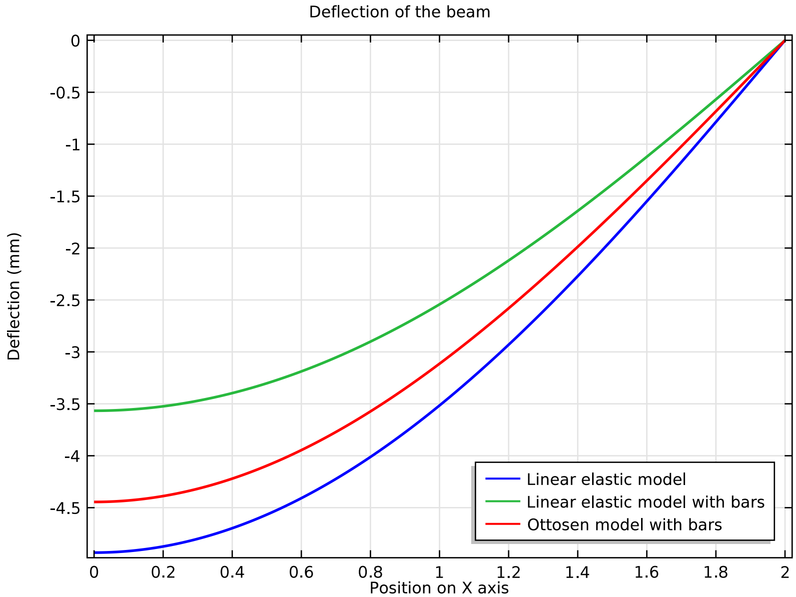

Figure 3: Deflection along the top surface of the concrete beam due to gravity and external load.

The simulation shows how force is transferred from the concrete beam to its steel reinforcement bars. Figure 4 shows effective stresses in the linear elastic model, Figure 5 shows the stress distribution in the reinforced linear elastic concrete. Compare both figures with each other and notice the change in stress level of the concrete once the bars are added. Figure 7 shows axial stresses in the bars. It is clear that the compressive stresses are experienced in the upper bars and tensile in the lower bars. Figure 8 shows the von Mises stresses in the concrete beam with Ottosen criterion.

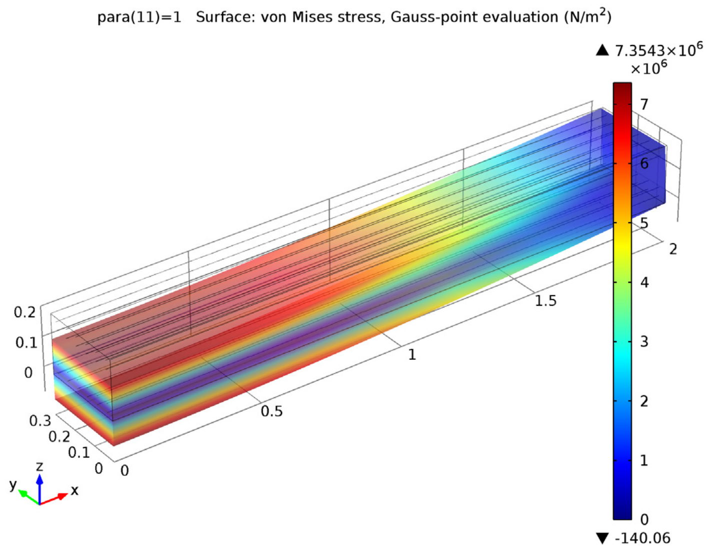

Figure 4: von Mises stress in a linear elastic beam.

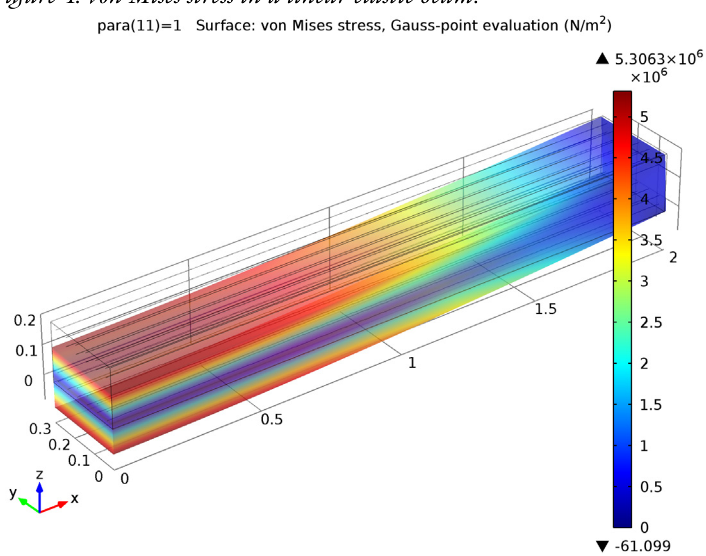

Figure 5: von Mises stress in a linear elastic beam after adding the reinforcement bars.

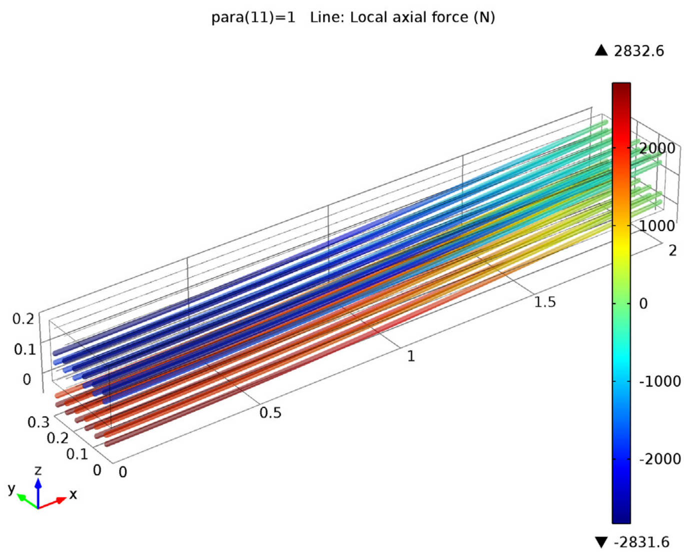

Figure 6: Axial force in the reinforcements bars.

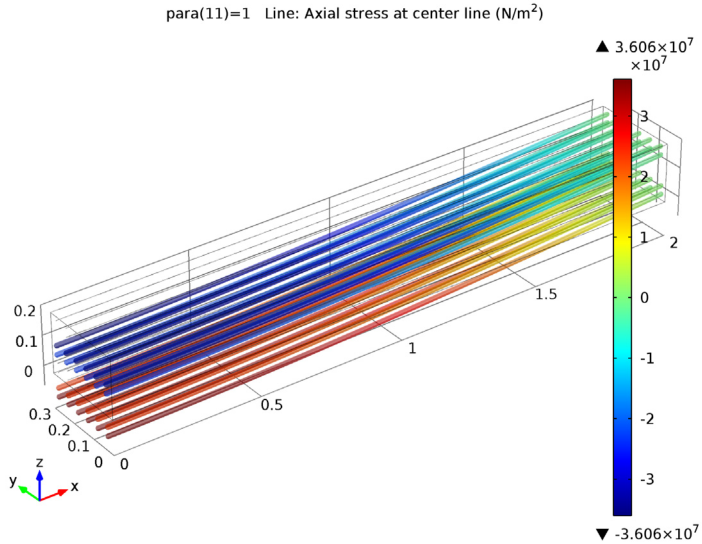

Figure 7: Axial stress in the steel rebars.

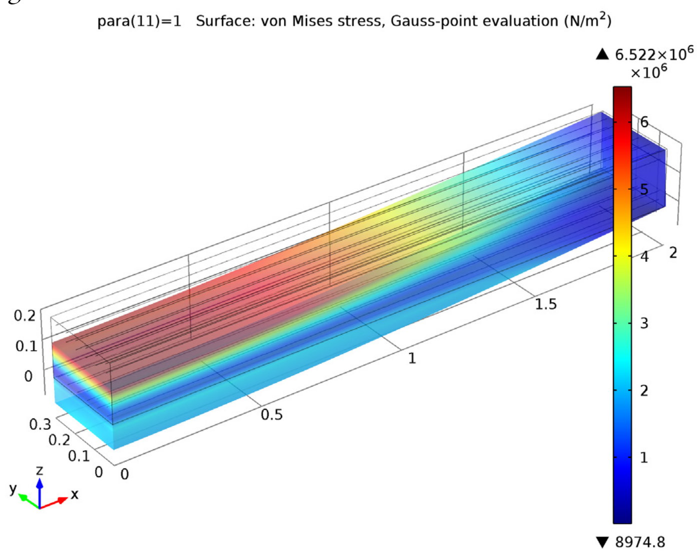

Figure 8: von Mises stress in the reinforced beam after adding the Ottosen criterion for the concrete.

Notes About the COMSOL Implementation

Since steel reinforcements are relatively thin compared to the concrete structures, it is assumed that they are only capable of transmitting axial forces. The bending stiffness of each bar does not contribute much to the overall total bending stiffness of the section, therefore the reinforcement bars are modeled with *truss* elements instead of *beam* elements.

In civil engineering it is also common practice that the rebars are pretensioned, but this effect is not included in the model. However it can easily be incorporated by adding **Initial strain** in the trusses.

In this example, the concrete is “glued” to the steel rebars, so bonding effects are not included.

Reference

1. W.F. Chen, *Plasticity in Reinforced Concrete*, McGraw-Hill, 1982.

Model Library path: `Geomechanics_Module/Tutorial_Models/concrete_beam`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

1 In the **New** window, click the **Model Wizard** button.

MODEL WIZARD

1 In the **Model Wizard** window, click the **3D** button.
2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
3 Click the **Add** button.
4 In the **Select Physics** tree, select **Structural Mechanics>Truss (truss)**.
5 Click the **Add** button.
6 Click the **Study** button.
7 In the **Select Study** tree, select **Preset Studies for Selected Physics>Stationary**.

8 Click the **Done** button.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
height	0.2[m]	0.2 m	Height of the beam
width	0.3[m]	0.3 m	Width of the beam
length	2[m]	2 m	Length of the beam
diam_bar	10[mm]	0.01 m	Diameter of the bar
bar_layers	2	2	Number of bar layers
layer_spacing	30[mm]	0.03 m	Layer spacing
bars_across_width	6	6	Number of bars across the width
width_spacing	width/ bars_across_width	0.05 m	Width spacing
para	0	0	Parameter

GEOMETRY 1

Block 1

1 On the **Geometry** toolbar, click **Block**.

2 In the **Block** settings window, locate the **Size** section.

3 In the **Width** edit field, type `length`.

4 In the **Depth** edit field, type `width`.

5 In the **Height** edit field, type `height`.

6 Click the **Build Selected** button.

Bézier Polygon 1

1 On the **Geometry** toolbar, click **More Primitives** and choose **Bézier Polygon**.

2 In the **Bézier Polygon** settings window, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click the **Add Linear** button.

4 Find the **Control points** subsection. In row **1**, set **y** to `width_spacing/2`.

5 In row **1**, set **z** to `layer_spacing/2`.

6 In row **2**, set **x** to `length`.

7 In row **2**, set **y** to `width_spacing/2`.

8 In row **2**, set **z** to `layer_spacing/2`.

9 Click the **Build Selected** button.

Array 1

1 On the **Geometry** toolbar, click **Array**.

2 In the **Array** settings window, locate the **Input** section.

3 Select the object **b1** only.

4 In the **Array** settings window, locate the **Size** section.

5 In the **y size** edit field, type `bars_across_width`.

6 In the **z size** edit field, type `bar_layers`.

7 Locate the **Displacement** section. In the **y** edit field, type `width_spacing`.

8 In the **z** edit field, type `layer_spacing`.

9 Click the **Build Selected** button.

Mirror 1

1 On the **Geometry** toolbar, click **Mirror**.

2 In the **Mirror** settings window, locate the **Input** section.

3 Select all bars.

4 In the **Mirror** settings window, locate the **Input** section.

5 Select the **Keep input objects** check box.

6 Locate the **Point on Plane of Reflection** section. In the **z** edit field, type `height/2`.

7 Click the **Build Selected** button.

Form Union

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

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

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

4 Clear the **Create pairs** check box.

5 Click the **Build All** button.

To make the displacements in the beam available for the bars, use a general extrusion operator.

DEFINITIONS

General Extrusion 1

1 On the **Definitions** toolbar, click **Component Couplings** and choose **General Extrusion**.

2 In the **General Extrusion** settings window, locate the **Source Selection** section.

3 Select Domain 1 only.

4 In the **General Extrusion** settings window, locate the **Destination Map** section.

5 In the **x-expression** edit field, type `X`.

6 In the **y-expression** edit field, type `Y`.

7 In the **z-expression** edit field, type `Z`.

8 Locate the **Source** section. From the **Source frame** list, choose **Material (X, Y, Z)**.

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Explicit** settings window, locate the **Input Entities** section.

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

4 Select Edges 13–36 only.

5 Right-click **Component 1>Definitions>Explicit 1** and choose **Rename**.

6 Go to the **Rename Explicit** dialog box and type `Rebars` in the **New name** edit field.

7 Click **OK**.

To model the failure of the material, add a material model to the Solid Mechanics interface.

SOLID MECHANICS

Concrete 1

1 In the **Model Builder** window, under **Component 1>Solid Mechanics** right-click **Linear Elastic Material 1** and choose **Concrete**.

2 In the **Concrete** settings window, locate the **Concrete Model** section.

3 From the **Concrete criterion** list, choose **Ottosen**.

Body Load 1

On the **Physics** toolbar, click **Domains** and from **Solid Mechanics** choose **Body Load**.

1 In the **Body Load** settings window, locate the **Domain Selection** section.

2 Select Domain 1 only.

3 In the **Body Load** settings window, locate the **Force** section.

4 Specify the $\mathbf{F}_V$ vector as

0	x
0	y
-solid.rho*g_const	z

Symmetry 1

1 On the **Physics** toolbar, click **Boundaries** and from **Solid Mechanics** choose **Symmetry**.

2 In the **Symmetry** settings window, locate the **Boundary Selection** section.

3 Select Boundary 1 only.

Rigid Connector 1

1 On the **Physics** toolbar, click **Boundaries** and from **Solid Mechanics** choose **Rigid Connector**.

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

3 Select Boundary 6 only.

4 In the **Rigid Connector** settings window, locate the **Prescribed Displacement at Center of Rotation** section.

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

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

7 Select the **Prescribed in z direction** check box.

8 Locate the **Prescribed Rotation at Center of Rotation** section. From the **By** list, choose **Constrained rotation**.

9 Select the **Constrain rotation around x-axis** check box.

10 Select the **Constrain rotation around z-axis** check box.

Boundary Load 1

1 On the **Physics** toolbar, click **Boundaries** and from **Solid Mechanics** choose **Boundary Load**.

2 In the **Boundary Load** settings window, locate the **Boundary Selection** section.

3 Select Boundary 4 only.

4 In the **Boundary Load** settings window, locate the **Force** section.

5 Specify the $\mathbf{F}_A$ vector as

0	x
0	y
-2e4*para	z

TRUSS

1 In the **Model Builder** window, under **Component 1** click **Truss**.

2 In the **Truss** settings window, locate the **Edge Selection** section.

3 From the **Selection** list, choose **Rebars**.

Cross Section Data 1

1 In the **Model Builder** window, expand the **Truss** node, then click **Cross Section Data 1**.

2 In the **Cross Section Data** settings window, locate the **Cross Section Data** section.

3 In the A edit field, type `pi/4*(diam_bar)^2`.

Prescribed Displacement 1

1 On the **Physics** toolbar, click **Edges** and choose **More>Prescribed Displacement**.

Use the general extrusion operator to prescribe the displacements of the bars.

2 In the **Prescribed Displacement** settings window, locate the **Edge Selection** section.

3 From the **Selection** list, choose **Rebars**.

4 Locate the **Prescribed Displacement** section. Select the **Prescribed in x direction** check box.

5 In the u_{0x} edit field, type `genext1(u)`.

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

7 In the u_{0y} edit field, type `genext1(v)`.

8 Select the **Prescribed in z direction** check box.

9 In the u_{0z} edit field, type `genext1(w)`.

Because the bar displacements are prescribed, the feature **Straight Edge Constraint** should be removed.

Straight Edge Constraint 1

1 In the **Model Builder** window, under **Component 1>Truss** right-click **Straight Edge Constraint 1** and choose **Delete**. In the **Confirm Delete** dialog box, click **Yes** to confirm.

MATERIALS

On the **Home** toolbar, click **Add Material**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Concrete**.

3 In the **Add Material** window, click **Add to Component**.

MATERIALS

Concrete

1 In the **Model Builder** window, under **Component 1>Materials** click **Concrete**.

2 In the **Material** settings window, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Uniaxial compressive strength	sigmauc	20e6	Pa	Yield stress parameters
Ottosen a parameter	aOttosen	1.3	1	Ottosen
Ottosen b parameter	bOttosen	3.2	1	Ottosen
Size factor	k1Ottosen	11.8	1	Ottosen
Shape factor	k2Ottosen	0.98	1	Ottosen

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Structural steel**.

3 In the **Add Material** window, click **Add to Component**.

MATERIALS

Structural steel

1 In the **Model Builder** window, under **Component 1>Materials** click **Structural steel**.

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

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

4 From the **Selection** list, choose **Rebars**.

MESH 1

Edge 1

1 In the **Model Builder** window, under **Component 1** right-click **Mesh 1** and choose **More Operations>Edge**.

2 In the **Edge** settings window, locate the **Edge Selection** section.

3 From the **Selection** list, choose **Rebars**.

Distribution 1

1 Right-click **Component 1>Mesh 1>Edge 1** and choose **Distribution**.

2 In the **Distribution** settings window, locate the **Distribution** section.

3 In the **Number of elements** edit field, type `100`.

Mapped 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **More Operations>Mapped**.

2 Select Boundary 1 only.

Distribution 1

1 Right-click **Component 1>Mesh 1>Mapped 1** and choose **Distribution**.

2 Select Edges 1 and 4 only.

3 In the **Distribution** settings window, locate the **Distribution** section.

4 In the **Number of elements** edit field, type `6`.

Swept 1

In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

Distribution 1

1 In the **Model Builder** window, under **Component 1>Mesh 1** right-click **Swept 1** and choose **Distribution**.

2 In the **Distribution** settings window, locate the **Distribution** section.

3 In the **Number of elements** edit field, type `40`.

4 Click the **Build All** button.

5 Click the **Go to Default 3D View** button on the Graphics toolbar.

6 Click the **Transparency** button on the Graphics toolbar.

7 Click the **Zoom Extents** button on the Graphics toolbar.

   The mesh should look like the one in Figure 2.

The first study solves only the linear elastic problem in the concrete beam without the reinforcement bars.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Stationary**.
2 In the **Stationary** settings window, locate the **Physics and Variables Selection** section.
3 Select the **Modify physics tree and variables for study step** check box.
4 In the **Physics and Variables Selection** tree, select **Component 1>Solid Mechanics>Linear Elastic Material 1>Concrete 1**.
5 Click **Disable**.
6 In the **Physics and Variables Selection** tree, select **Component 1>Truss**.
7 Click **Disable**.
8 Click to expand the **Study Extensions** section. Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.
9 Click **Add**.
10 In the table, enter the following settings:

Auxiliary parameter	Parameter value list
para	`range(0,0.1,1)`

11 Locate the **Physics and Variables Selection** section. In the **Physics and Variables Selection** tree, select **Component 1**.
12 On the **Home** toolbar, click **Compute**.

RESULTS

Stress (solid)

1 On the **3D Plot Group** toolbar, click **Plot**.
2 Click the **Zoom Extents** button on the Graphics toolbar.

ROOT

Add a second study to solve the model with the reinforcement bars.

1 On the **Home** toolbar, click **Add Study**.

ADD STUDY

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

2 Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.

3 In the **Add Study** window, click **Add Study**.

STUDY 2

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2** click **Step 1: Stationary**.

2 In the **Stationary** settings window, locate the **Physics and Variables Selection** section.

3 Select the **Modify physics tree and variables for study step** check box.

4 In the **Physics and Variables Selection** tree, select **Component 1>Solid Mechanics>Linear Elastic Material 1>Concrete 1**.

5 Click **Disable**.

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

7 Click **Add**.

8 In the table, enter the following settings:

Auxiliary parameter	Parameter value list
para	`range(0,0.1,1)`

9 Locate the **Physics and Variables Selection** section. In the **Physics and variables selection** tree, select **Component 1**.

Solver 2

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

This problem is better solved fully coupled.

2 In the **Model Builder** window, expand the **Solver 2** node.

3 Right-click **Stationary Solver 1** and choose **Fully Coupled**.

4 On the **Home** toolbar, click **Compute**.

RESULTS

Stress (solid) 1

The first default plot generated by the **Study 2** shows the von Mises stress in Figure 5. This result can be compared to the result without reinforcement bars shown in Figure 4.

1 In the **Model Builder** window, under **Results** right-click **Stress (solid) 1** and choose **Rename**.
2 Go to the **Rename 3D Plot Group** dialog box and type `Stress with bars` in the **New name** edit field.
3 Click **OK**.
4 On the **3D Plot Group** toolbar, click **Plot**.
5 Click the **Zoom Extents** button on the Graphics toolbar.

Force (truss)

The second default plot generated by the **Study 2** shows the force in the bars in Figure 6.

1 On the **3D plot group** toolbar, click **Plot**.
2 Click the **Zoom Extents** button on the Graphics toolbar.

Stress (truss)

The third default plot generated by the **Study 2** shows the axial stress in the bars as shown in Figure 7.

1 On the **3D plot group** toolbar, click **Plot**.
2 Click the **Zoom Extents** button on the Graphics toolbar.

ADD STUDY

1 Go to the **Add Study** window.
2 Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.
3 In the **Add study** window, click **Add Study**.

STUDY 3

Step 1: Stationary

1 In the **Model Builder** window, under **Study 3** click **Step 1: Stationary**.
2 In the **Stationary** settings window, locate the **Study Extensions** section.
3 Select the **Auxiliary sweep** check box.
4 Click **Add**.
5 In the table, enter the following settings:

Auxiliary parameter	Parameter value list
para	`range(0,0.1,1)`

6 From the **Run continuation for** list, choose **Manual**.

7 From the **Continuation parameter** list, choose **para**.

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

9 In the **Study** settings window, locate the **Study Settings** section.

10 Clear the **Generate default plots** check box.

Solver 3

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

2 In the **Model Builder** window, expand the **Solver 3** node.

3 Right-click **Stationary Solver 1** and choose **Fully Coupled**.

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

5 From the **Predictor** list, choose **Constant** to improve the convergence for the elastoplastic case.

6 On the **Home** toolbar, click **Compute**.

Duplicate the first von Mises stress plot group to compare results with or without the failure behavior.

RESULTS

Stress (solid) 1

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

2 In the **3D Plot Group** settings window, locate the **Data** section.

3 From the **Data set** list, choose **Solution 3**.

4 On the **3D plot group** toolbar, click **Plot**.

5 Click the **Zoom Extents** button on the Graphics toolbar.

6 Right-click **Results>Stress (solid) 1** and choose **Rename**.

7 Go to the **Rename 3D Plot Group** dialog box and type `Stress with bars and Ottosen` in the **New name** edit field.

8 Click **OK**.

Compare the results to Figure 8.

To visualize the plastic zone, proceed as follows.

3D Plot Group 6

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **3D Plot Group** settings window, locate the **Data** section.
3 From the **Data set** list, choose **Solution 3**.
4 Right-click **Results>3D Plot Group 6** and choose **Surface**.
5 In the **Surface** settings window, locate the **Expression** section.
6 In the **Expression** edit field, type `solid.epe>0`.
7 On the **3D plot group** toolbar, click **Plot**.
8 Right-click **Results>3D Plot Group 6>Surface 1** and choose **Deformation**.
9 On the **3D plot group** toolbar, click **Plot**.
10 Click the **Zoom Extents** button on the Graphics toolbar.

To compare the deflection of the beam for the three models, proceed as follows.

1D Plot Group 7

1 On the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **1D Plot Group** settings window, click to expand the **Title** section.
3 From the **Title type** list, choose **Manual**.
4 In the **Title** text area, type `Deflection of the beam`.
5 Locate the **Plot Settings** section. Select the **x-axis label** check box.
6 In the associated edit field, type `Position on X axis`.
7 Select the **y-axis label** check box.
8 In the associated edit field, type `Deflection (mm)`.
9 Click to expand the **Legend** section. From the **Position** list, choose **Lower right**.
10 On the **1D plot group** toolbar, click **Line Graph**.
11 In the **Line Graph** settings window, locate the **Data** section.
12 From the **Data set** list, choose **Solution 1**.
13 From the **Parameter selection (para)** list, choose **Last**.
14 Select Edge 5 only.
15 Locate the **y-axis data** section. Type **w**.
16 Locate the **y-Axis Data** section. From the **Unit** list, choose **mm**.
17 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
18 In the **Expression** edit field, type `X`.
19 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. In the **Width** edit field, type `2`.

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

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

22 In the table, enter the following settings:

Legends
`Linear elastic model`

23 Right-click **Results>1D Plot Group 7>Line Graph 1** and choose **Duplicate**.

24 Click **Line Graph 2**. In the **Line Graph** settings window, locate the **Data** section.

25 From the **Data set** list, choose **Solution 2**.

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

Legends
`Linear elastic model with bars`

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

28 Click **Line Graph 3**. In the **Line Graph** settings window, locate the **Data** section.

29 From the **Data set** list, choose **Solution 3**.

30 On the **1D plot group** toolbar, click **Plot**.

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

Legends
`Ottosen model with bars`

32 Click the **Zoom Extents** button on the Graphics toolbar.

33 In the **Model Builder** window, right-click **1D Plot Group 7** and choose **Rename**.

34 Go to the **Rename 1D Plot Group** dialog box and type `Deflection` in the **New name** edit field.

35 Click **OK**.