Created in COMSOL Multiphysics 6.0

Plastic Strain Mapping

This model is licensed under the COMSOL Software License Agreement 6.0.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Mapping of results, such as plastic strains, between dissimilar meshes can be useful in different situations. This model demonstrates a simple and effective way to accomplish this. In the example, you will model a perforated plate subjected to in plane loading. The analysis is performed in two separate studies. In the first study (Study 1), the plate is modeled using triangular mesh elements, and it is loaded well into its plastic regime. The elastoplastic solution at an intermediate stage of deformation is then used as a starting point for a second study (Study 2), where the plastic strains are mapped onto a plate discretized with quadrilateral mesh elements. The results from the two studies are compared after unloading.

Model Definition

Figure 1 shows the plate's geometry. Due to the double symmetry of the geometry you only need to analyze a quarter of the plate.

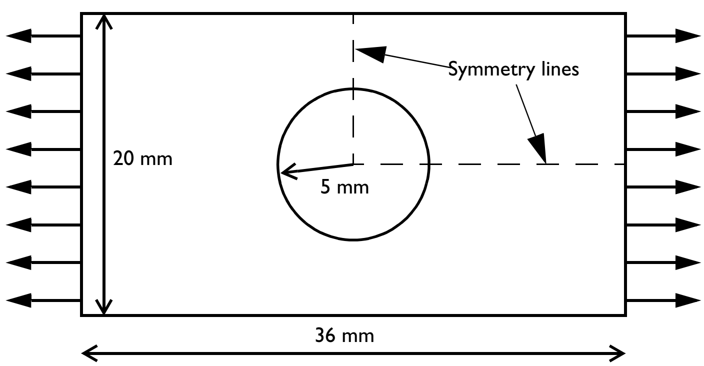

Figure 1: The plate geometry.

Because the plate is thin and the loads are in plane, you can assume a plane stress condition.

MATERIAL

- Elastic properties: $E = 70000$ MPa and $\nu = 0.2$.
- Plastic properties: Yield stress 243 MPa and a linear isotropic hardening with tangent modulus 2171 MPa.

CONSTRAINTS AND LOADS

- Symmetry plane constraints are applied on the vertical boundary to the left, and the lower horizontal boundary.
- The right vertical edge is subjected to a stress, which increases from zero to a maximum value of 133.65 MPa and then is released. The peak value is selected so that the mean stress over the section through the hole is 10% above the yield stress (=1.1·243·(20–10)/20). The complete release of the stress corresponds to a parameter value of 2.2, see Figure 2.

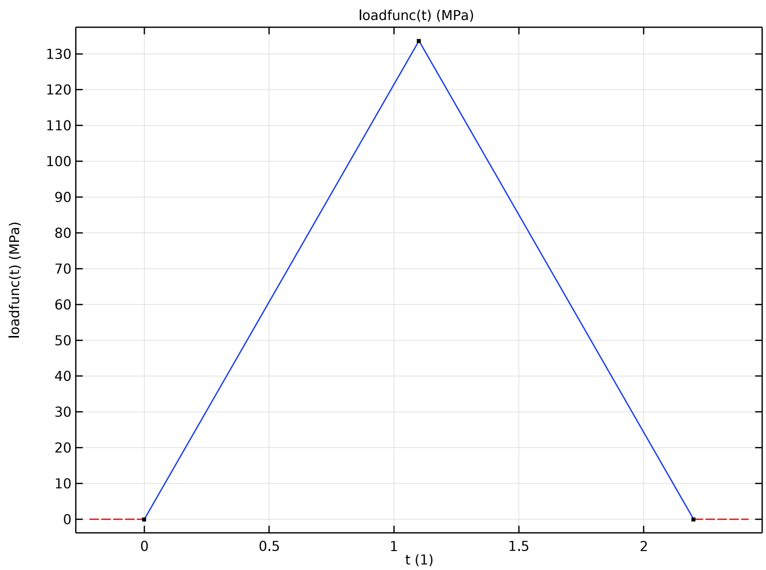

Figure 2: The stress applied to the right vertical edge.

STUDIES

- In Study 1, a plate that uses triangular mesh elements is used, see Figure 3. The plate is loaded, and then released using the prescribed stress on the right vertical edge, see Figure 2.
- In Study 2, a plate that uses quadrilateral mesh elements is used, see Figure 4. This study is started from an intermediate stage, corresponding to a parameter value of 0.8, see

Figure 2. The displacements and plastic strains from the plate in Study 1 are mapped onto the plate in Study 2..

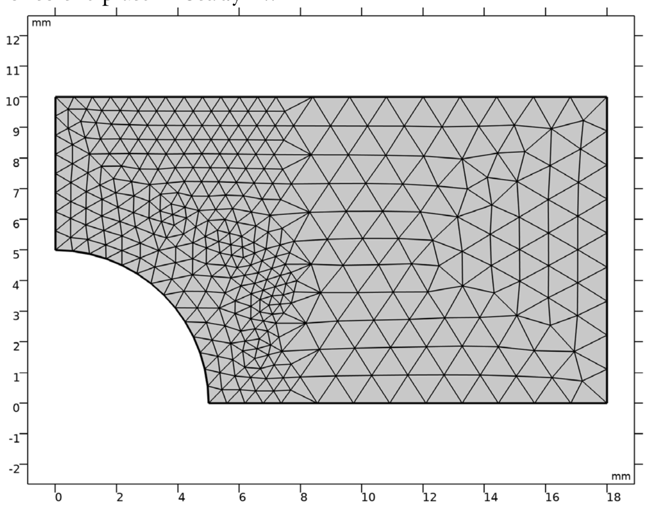

Figure 3: The plate, discretized with triangular mesh elements.

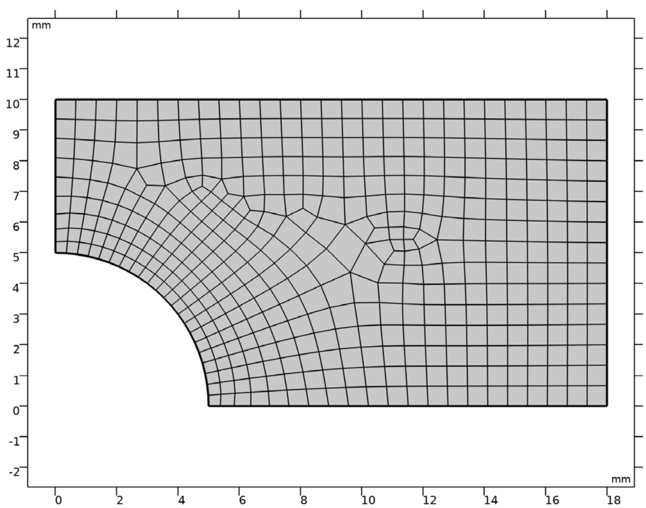

Figure 4: The plate, discretized with quadrilateral mesh elements.

Results and Discussion

Figure 5 and Figure 6 show the von Mises stress distribution after unloading, for the two cases of triangular and quadrilateral meshes. To within the small differences that can be attributed to different mesh element types and differences in mesh topology, the results are the same.

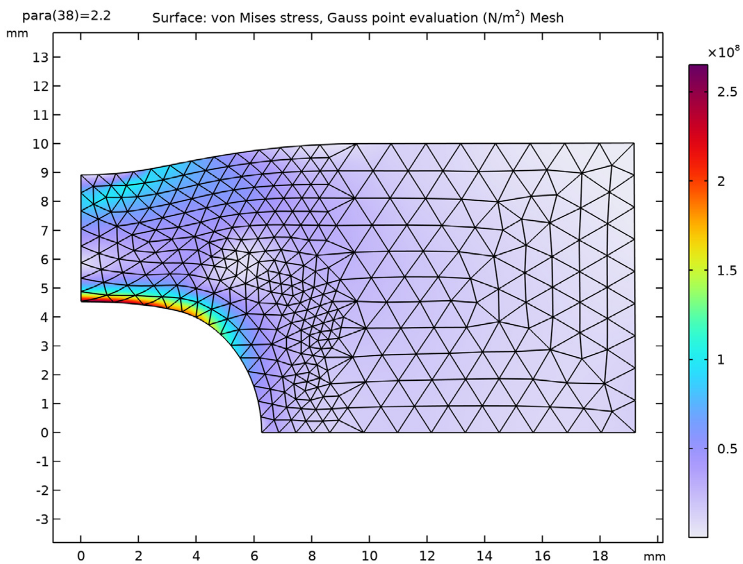

Figure 5: Deformation and von Mises stress after unloading, using the plate with triangular mesh elements.

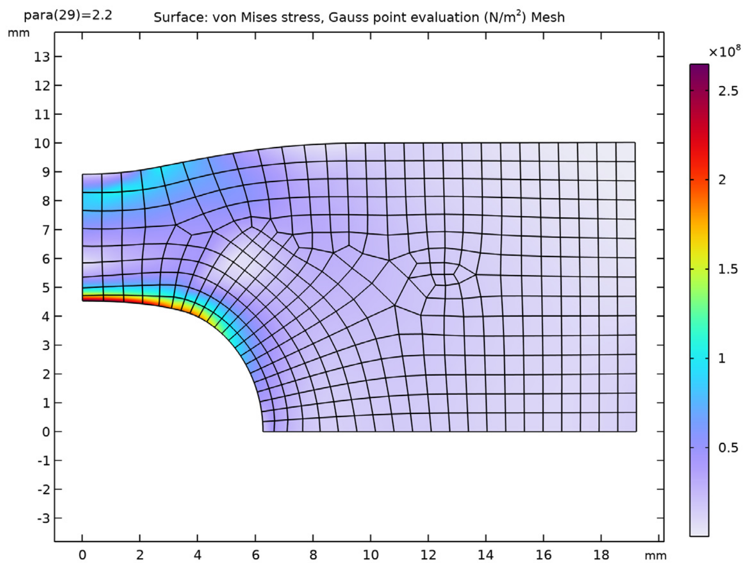

Figure 6: Deformation and von Mises stress after unloading, using the plate with quadrilateral mesh elements.

Figure 7 and Figure 8 show the regions where plastic straining has occurred, after unloading, for the two cases of triangular and quadrilateral meshes. In these figures, a value of one means that the equivalent plastic strain is greater than zero, and a value of zero means that the material at that location has deformed purely elastically throughout the process. In this comparison, the differences appear larger, however the differences stem from the discrete (zero to one) character of the output, in combination with the differences in mesh element types and differences in topology.

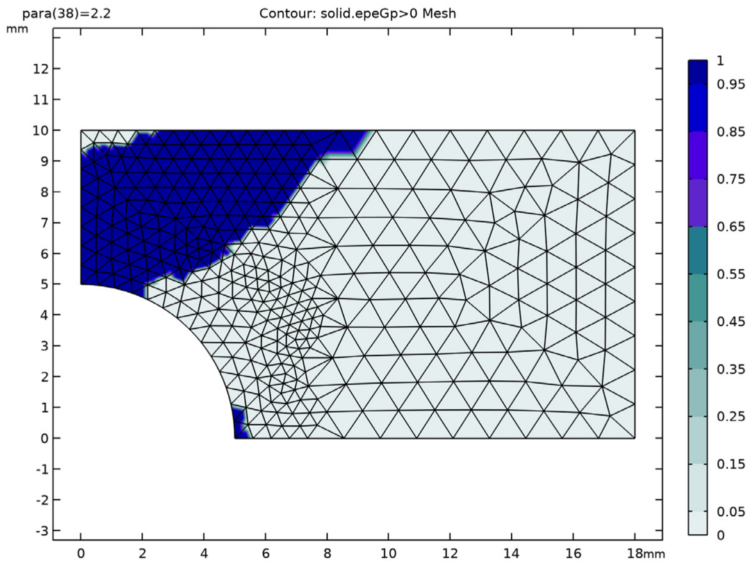

Figure 7: Plastic region after unloading, using the plate with triangular mesh elements.

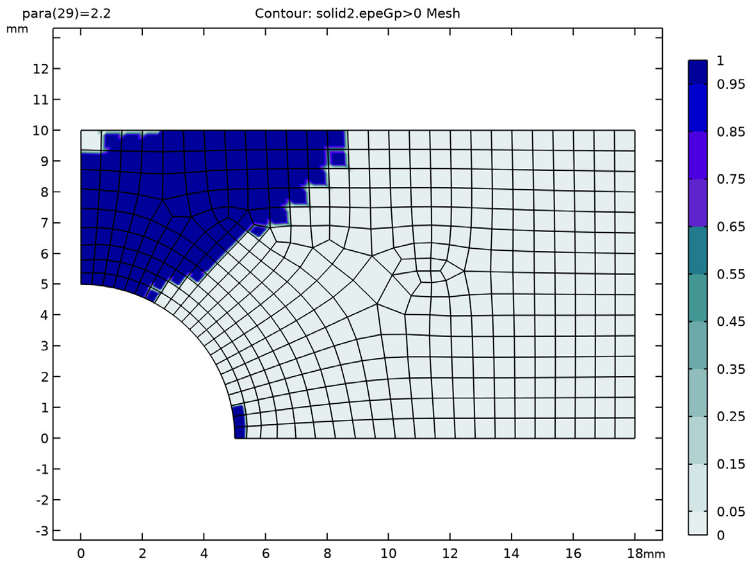

Figure 8: Plastic region after unloading, using the plate with quadrilateral mesh elements.

Notes About the COMSOL Implementation

To properly initialize the study of the plate using quadrilateral mesh elements, there are two modeling aspects to consider:

- The plastic strains have to be mapped between two different components in the model.
- The mapped (non zero) plastic strains have to be assigned as initial values of the plastic strains in the plate in the second study.

Because the mapping is to be performed between two different components, a general extrusion operator is used. The assignment of plastic strains is performed using the **Set Variables** subnode to the **Plasticity** node. With this subnode, you can set plastic variables to certain values, on satisfying a logical condition that you define. The study using quadrilateral elements is performed from an intermediate deformation stage, corresponding to a parameter value para = 0.8, and the logical condition is expressed accordingly.

To properly initialize the second study, the equivalent plastic strain (epe) and the plastic strain tensor (ep) are prescribed.

Application Library path: `Nonlinear_Structural_Materials_Module/Plasticity/plastic_strain_mapping`

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**.

GEOMETRY 1

Begin by changing the length unit to millimeters.

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 `18`.

4 In the **Height** text field, type `10`.

5 Click **Build Selected**.

Circle 1 (c1)

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

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type 5.

4 Click **Build Selected**.

Difference 1 (dif1)

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

2 Select the object **r1** only.

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

4 Find the **Objects to subtract** subsection. Click to select the **Activate Selection** toggle button.

5 Select the object **c1** only.

6 Click **Build Selected**.

Define the elastoplastic material properties.

GLOBAL DEFINITIONS

Material 1 (mat1)

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

2 Expand the **Material 1 (mat1)** node.

3 In the **Model Builder** window, under **Global Definitions>Materials>Material 1 (mat1)** click **Basic (def)**.

4 In the **Settings** window for **Basic**, locate the **Output Properties** section.

5 Click **Select Quantity**.

6 In the **Physical Quantity** dialog box, click **Filter**.

7 In the tree, select **Solid Mechanics>Young's modulus (Pa)**.

8 Click **OK**.

9 In the **Settings** window for **Basic**, locate the **Output Properties** section.

10 In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Young's modulus	E	70e9	Pa	1x1

11 Click **Select Quantity**.

12 In the **Physical Quantity** dialog box, select **Solid Mechanics>Poisson's ratio (1)** in the tree.

13 Click **OK**.

14 In the **Settings** window for **Basic**, locate the **Output Properties** section.

15 In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Poisson's ratio	nu	0.2	1	1x1

16 Click **Select Quantity**.

17 In the **Physical Quantity** dialog box, select **General>Density (kg/m^3)** in the tree.

18 Click **OK**.

19 In the **Settings** window for **Basic**, locate the **Output Properties** section.

20 In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Density	rho	7850	kg/m³	1x1

21 In the **Model Builder** window, click **Material 1 (mat1)**.

22 In the **Settings** window for **Material**, click to expand the **Material Properties** section.

23 In the **Material properties** tree, select **Solid Mechanics>Elastoplastic Material**.

24 Click **Add to Material**.

25 In the **Material properties** tree, select **Solid Mechanics>Elastoplastic Material>Elastoplastic Material Model**.

26 Click **Add to Material**.

27 In the **Model Builder** window, under **Global Definitions>Materials>Material 1 (mat1)** click **Elastoplastic material model (ElastoplasticModel)**.

28 In the **Settings** window for **Elastoplastic Material Model**, locate the **Output Properties** section.

29 In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Initial yield stress	sigmags	243e6	Pa	1x1
Isotropic tangent modulus	Et	2.171e9	Pa	1x1

Add a solver parameter for controlling the load expression.

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
para	0	0	Horizontal load parameter

Interpolation 1 (int1)

1 In the **Home** toolbar, click f(x) **Functions** and choose **Global>Interpolation**.

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

3 In the **Function name** text field, type `loadfunc`.

4 In the table, enter the following settings:

t	f(t)
0	0
1.1	133.65
2.2	0

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

Argument	Unit
t	1

6 In the **Function** table, enter the following settings:

Function	Unit
loadfunc	MPa

7 Click **Plot**.

The interpolation function defines the load function.

MATERIALS

Material Link 1 (matlnk1)

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials>Material Link**.

SOLID MECHANICS (SOLID)

1 In the **Settings** window for **Solid Mechanics**, locate the **2D Approximation** section.

2 From the list, choose **Plane stress**.

3 Locate the **Thickness** section. In the d text field, type `10[mm]`.

Linear Elastic Material 1

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

Plasticity 1

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

Symmetry 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
2. Select Boundaries 1 and 3 only.

Boundary Load 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
2. Select Boundary 4 only.
3. In the **Settings** window for **Boundary Load**, locate the **Force** section.
4. Specify the $\mathbf{F}_A$ vector as

`loadfunc(para)`	x
`0`	y

MESH 1

Free Triangular 1

1. In the **Mesh** toolbar, click **Free Triangular**.

   The mesh should be refined in areas of anticipated high stress and strain gradients.

Refine 1

1. In the **Mesh** toolbar, click **Modify** and choose **Refine**.
2. In the **Settings** window for **Refine**, click to expand the **Refine Elements in Box** section.
3. Select the **Specify bounding box** check box.
4. In row **x**, set **Upper bound** to `8`.
5. In row **y**, set **Upper bound** to `10`.
6. In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

STUDY 1

Step 1: Stationary

Set up an auxiliary sweep for the `para` parameter.

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
para (Horizontal load parameter)	`0 range(0.40,0.05,2.2)`

With these settings, the edge load you defined earlier increases from zero to a maximum value of 133.65 MPa and is then released.

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

RESULTS

The default plot shows the von Mises stress for the final parameter value.

Stress, Triangular Mesh

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

2 In the **Settings** window for **2D Plot Group**, type `Stress, Triangular Mesh` in the **Label** text field.

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Mesh 1

Right-click **Stress, Triangular Mesh** and choose **Mesh**.

Mesh 1

1 In the **Model Builder** window, expand the **Results>Stress, Triangular Mesh** node, then click **Mesh 1**.

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

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

Deformation 1

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

Visualize the plastic zone using a Boolean expression `solid.epeGp>0`; which is 1 in the plastic region and 0 elsewhere.

Plastic Region, Triangular Mesh

1 In the **Model Builder** window, under **Results** click **Equivalent Plastic Strain (solid)**.

2 In the **Settings** window for **2D Plot Group**, type `Plastic Region, Triangular Mesh` in the **Label** text field.

Contour 1

1 In the **Model Builder** window, expand the **Plastic Region, Triangular Mesh** node, then click **Contour 1**.

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

3 In the **Expression** text field, type `solid.epeGp>0`.

4 In the **Description** text field, type `solid.epeGp>0`.

Mesh 1

1 In the **Model Builder** window, right-click **Plastic Region, Triangular Mesh** and choose **Mesh**.

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

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

Now, make a second component that will define the same plate, but using quadrilateral mesh elements.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component>2D**.

GEOMETRY 2

Import the geometry from the first component into the second.

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.

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

3 From the **Source** list, choose **Geometry sequence**.

4 From the **Geometry** list, choose **Geometry 1**.

5 Click **Import**.

MATERIALS

Material Link 2 (matlnk2)

In the **Model Builder** window, under **Component 2 (comp2)** right-click **Materials** and choose **More Materials>Material Link**.

ADD PHYSICS

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

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

3 In the tree, select **Structural Mechanics>Solid Mechanics (solid)**.

4 Click **Add to Component 2** in the window toolbar.

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

SOLID MECHANICS 2 (SOLID2)

1 In the **Settings** window for **Solid Mechanics**, locate the **2D Approximation** section.

2 From the list, choose **Plane stress**.

3 Locate the **Thickness** section. In the d text field, type `10[mm]`.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 2 (comp2)>Solid Mechanics 2 (solid2)** click **Linear Elastic Material 1**.

Plasticity 1

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

Set Variables 1

In the **Physics** toolbar, click **Attributes** and choose **Set Variables**.

To be able to map plastic strains from one component to another, a general extrusion operator is used.

DEFINITIONS (COMP1)

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

General Extrusion 1 (genext1)

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

2 In the **Settings** window for **General Extrusion**, locate the **Destination Map** section.

3 In the **x-expression** text field, type `X`.

4 In the **y-expression** text field, type `Y`.

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

6 Select Domain 1 only.

SOLID MECHANICS 2 (SOLID2)

Set Variables 1

The **Setting condition** is phrased so that it is fulfilled only at the start of the second study, when the parameter value is 0.8.

1 In the **Model Builder** window, under **Component 2 (comp2)>Solid Mechanics 2 (solid2)>Linear Elastic Material 1>Plasticity 1** click **Set Variables 1**.

2 In the **Settings** window for **Set Variables**, locate the **Set Variables** section.

3 In the **Setting condition** text field, type `para<0.8001`.

4 In the ε_{pe} text field, type `comp1.genext1(solid.epe)`.

5 In the ε_p table, enter the following settings:

comp1.genext1(solid.epXX)	comp1.genext1(solid.epXY)	0
comp1.genext1(solid.epXY)	comp1.genext1(solid.epYY)	0
0	0	comp1.genext1(solid.epZZ)

Symmetry 1

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

2 Select Boundaries 1 and 3 only.

Boundary Load 1

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

2 Select Boundary 4 only.

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

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

loadfunc(para)	x
0	y

MESH 2

Free Quad 1

In the **Mesh** toolbar, click **Free Quad**.

Size

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

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

3 From the **Predefined** list, choose **Finer**.

Size 1

1 In the **Model Builder** window, right-click **Free Quad 1** and choose **Size**.

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

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

4 Select Boundary 5 only.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Extra fine**.

6 Click **Build All**.

ADD STUDY

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

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

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

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

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 **Physics and Variables Selection** section.

2 In the table, clear the **Solve for** check box for **Solid Mechanics (solid)**.

3 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

4 From the **Study** list, choose **Study 1, Stationary**.

5 From the **Parameter value (para)** list, choose **0.8**.

6 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

7 From the **Method** list, choose **Solution**.

8 From the **Study** list, choose **Study 1, Stationary**.

9 From the **Parameter value (para)** list, choose **0.8**.

10 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

11 Click **Add**.

12 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Horizontal load parameter)	`range(0.80,0.05,2.2)`	

13 Click **Compute**.

RESULTS

Stress, Quadrilateral Mesh

1 In the **Model Builder** window, expand the **Results>Stress (solid2)** node, then click **Stress (solid2)**.

2 In the **Settings** window for **2D Plot Group**, type `Stress, Quadrilateral Mesh` in the **Label** text field.

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Mesh 1

1 Right-click **Stress, Quadrilateral Mesh** and choose **Mesh**.

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

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

Deformation 1

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

Plastic Region, Quadrilateral Mesh

1 In the **Model Builder** window, expand the **Results>Equivalent Plastic Strain (solid2)** node, then click **Equivalent Plastic Strain (solid2)**.

2 In the **Settings** window for **2D Plot Group**, type `Plastic Region, Quadrilateral Mesh` in the **Label** text field.

Contour 1

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

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

3 In the **Expression** text field, type `solid2.epeGp>0`.

4 In the **Description** text field, type `solid2.epeGp>0`.

Mesh 1

1 In the **Model Builder** window, right-click **Plastic Region, Quadrilateral Mesh** and choose **Mesh**.

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

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