Created in COMSOL Multiphysics 6.2

Stress Analysis of an Elliptic Membrane

General Description

In this benchmark, the static stress analysis described in the NAFEMS Test LE1, "Elliptic Membrane", found on page 5 in Ref. 1 is performed. It is an analysis of a linear elastic plane stress model.

The computed stress level is compared with the values given in the benchmark report.

In addition to the original benchmark, a mesh convergence study is performed.

GEOMETRY

The geometry is an ellipse with an elliptical hole in it. The outer and inner edges are defined by the equations

$$\left(\frac{X}{3.25}\right)^2 + \left(\frac{Y}{2.75}\right)^2 = 1$$

$$\left(\frac{X}{2}\right)^2 + \left(\frac{Y}{1}\right)^2 = 1$$

The thickness (which actually does not influence the analysis) is 0.1 m.

Due to symmetry in load and in geometry, the analysis only includes a quarter of the geometry as shown in Figure 1.

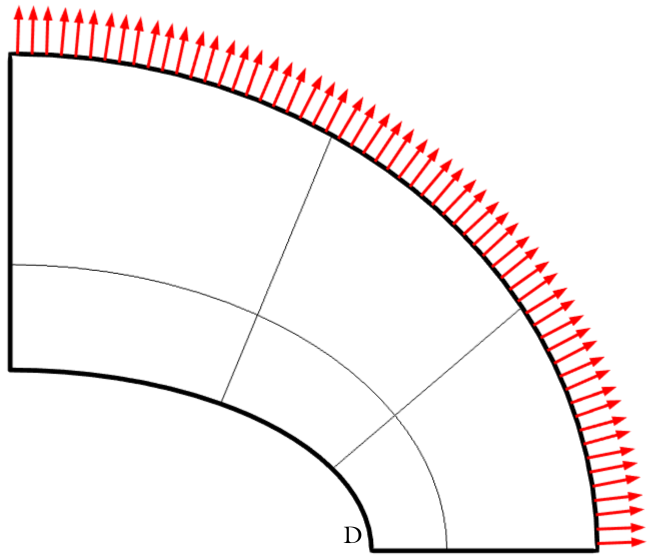

Figure 1: The geometry and load. Only the quarter that is analyzed is shown.

MATERIAL

Isotropic with $E = 2.1 \cdot 10^{11}$ Pa and $\nu = 0.3$.

LOAD

An evenly distributed load of 10 MPa acts along the outward normal of the outer boundary.

CONSTRAINTS

Symmetry conditions are used along the cuts at $X = 0$ and $Y = 0$.

Model Setup

The Solid Mechanics interface with the plane stress assumption is used.

Four meshes are exactly specified in Ref. 1. The “coarse” mesh has 6 quadrilateral or 12 triangular elements. The “fine” mesh has 24 quadrilateral or 48 triangular elements. The triangular elements are created by splitting the quadrilateral elements along a diagonal.

The specified meshes are shown in Figure 2 and Figure 3. For the quadrilateral meshes, a reduced integration scheme with hourglass stabilization is also used.

Note that the algorithm used to subdivide the quadrilaterals into triangles tries to place the diagonal so that the two new triangles have the best possible shape. As a result, the triangular mesh used is not identical to the one in the original benchmark, even though the number of elements and even node locations are identical. For very coarse meshes, this can change the stress results appreciably.

For the mesh convergence study, these meshes are uniformly refined using a parameter `div`. The number of elements along the elliptical boundaries is `3*div` and the number of elements along the symmetry cuts is `2*div`.

The number of displacement degrees of freedom varies from 48 (div = 1 and quadrilaterals with linear shape order) to 935810 (div = 64 and triangles with cubic shape order).

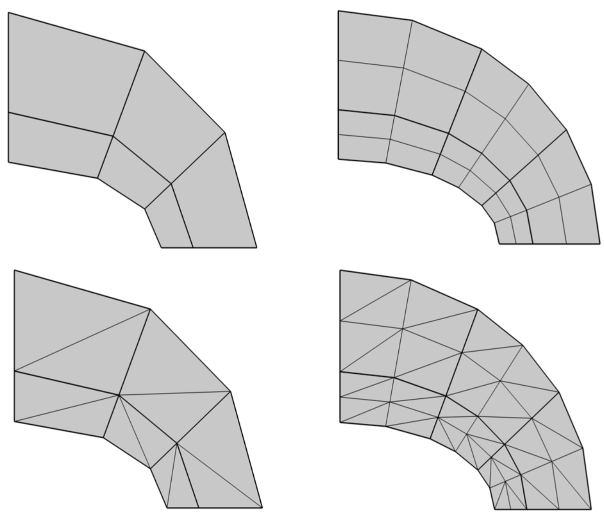

Figure 2: The meshes as specified in Ref. 1. Left column: "coarse" (div=1). Right column: "fine" (div=2).

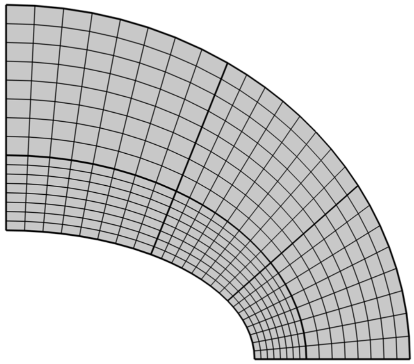

Figure 3: A quadrilateral mesh with div=8.

Due to the specification of the benchmark, the modeling differs somewhat from what you would use in practice:

- The interior boundaries in the model are created for matching the specification of the mesh in the NAFEMS benchmark as close as possible. If you were to solve the problem without these constraints, the modeling would be significantly simplified. Only two ellipses would be needed in the Geometry sequence.
- The knowledge about where a stress concentration is expected suggests that you should use a mesh such that more elements are present in the region around point D to get optimal accuracy, see Figure 1.
- Using the possibility to generate a free triangular mesh instead of one where quadrilateral elements are split along the diagonals would also give a mesh with better element quality.

Results and Discussion

The purpose of this test, in addition to a pure verification of the element formulation, is to check how well the software can represent a nontrivial geometrical shape such as an ellipse. It also evaluates the application of a distributed load.

The distribution of the direct stress in the Y direction is shown in Figure 4. As can be seen the result has steep gradients toward the point with maximum values.

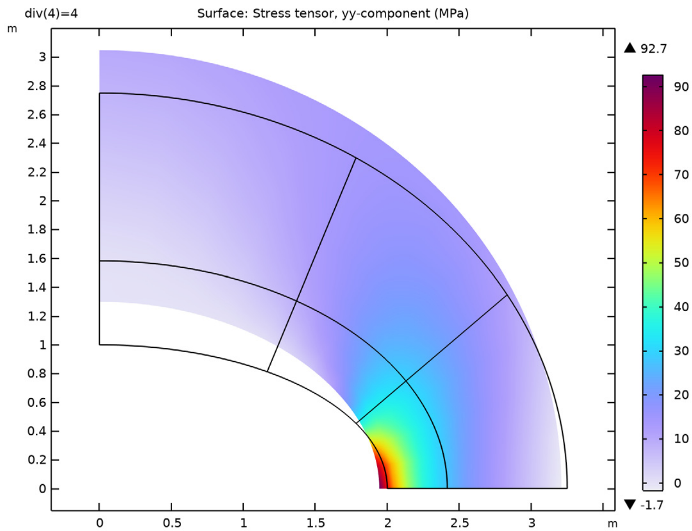

Figure 4: The distribution of the σ_y stress component using div=4 and second order quadrilateral elements.

The normal stress σ_y at the elliptic hole is evaluated at the point D located at $X = 2$, $Y = 0$ (see Figure 1). The target value according to Ref. 1 is 92.7 MPa. The value is based on an analytical result. The COMSOL Multiphysics results for the “coarse” and “fine” meshes are given in Table 1.

TABLE 1: COMPUTED RESULTS FOR THE MESHES SPECIFIED IN THE BENCHMARK.

STUDY NUMBER	ELEMENT TYPE	DISCRETIZATION	MESH	COMPUTED VALUE	RELATIVE ERROR
1	Quadrilateral	Linear	Coarse	71.0	-23.3%
1	Quadrilateral	Linear	Fine	85.5	-7.7%
2	Quadrilateral	Quadratic	Coarse	89.0	-4.0%
2	Quadrilateral	Quadratic	Fine	92.7	0.0%
3	Quadrilateral	Cubic	Coarse	93.7	+1.1%
3	Quadrilateral	Cubic	Fine	92.5	-0.2%
4	Triangle	Linear	Coarse	33.9	-63.4%
4	Triangle	Linear	Fine	54.1	-41.6%
5	Triangle	Quadratic	Coarse	74.2	-20.0%

TABLE 1: COMPUTED RESULTS FOR THE MESHES SPECIFIED IN THE BENCHMARK.

STUDY NUMBER	ELEMENT TYPE	DISCRETIZATION	MESH	COMPUTED VALUE	RELATIVE ERROR
5	Triangle	Quadratic	Fine	87.7	-5.4%
6	Triangle	Cubic	Coarse	86.0	-7.2%
6	Triangle	Cubic	Fine	91.8	-0.9%
9	Quadrilateral	Linear, reduced	Coarse	47.1	-49.2%
10	Quadrilateral	Linear, reduced	Fine	62.1	-32.9%
11	Quadrilateral	Quadratic, reduced	Coarse	85.0	-8.3%
12	Quadrilateral	Quadratic, reduced	Fine	92.2	-0.5%
13	Quadrilateral	Cubic, reduced	Coarse	93.5	+1.0%
14	Quadrilateral	Cubic, reduced	Fine	92.4	-0.3%

As can be expected, the coarse mesh is not able to capture the stress concentration unless elements with high order are used. Generally the quadrilaterals perform better than the corresponding triangles.

The mesh that is denoted as “fine” is more similar to what you would use in an analysis of a larger structure in a case where you are not specifically interested in a high resolution of the stress concentration. Still, with quadratic shape order elements, the accuracy is good enough for most engineering purposes. With the current mesh, the triangular elements will have a small angle at the stress evaluation point, hence the less accurate result.

Using elements with linear shape functions for structural analysis is commonly avoided in the finite element community.

The results of the mesh convergence study are shown in Figure 5. The element size h is defined as `0.417[m]/div`, which is the length of an edge in the element where the stress is measured.

The target value in Ref. 1, 92.7 MPa, is given with only three digits. This is not accurate enough for the convergence study here. Instead, the error is measured relative to the value 92.65817 MPa, toward which σ_y converges.

The convergence behavior is as expected since it is faster for elements with a higher shape function order. It can also be seen that quadrilaterals are somewhat more accurate than triangles for quadratic and cubic elements. The quadrilateral elements with a reduced ingratiation rule perform similar to the triangles for first order shape functions. For higher order shape functions, they are almost on par with their fully integrated counterparts.

The other two in-plane stress components σ_x and τ_{xy} should both be zero at point D since the boundary is free. In Figure 6 and Figure 7 similar convergence graphs are shown for these stress components.

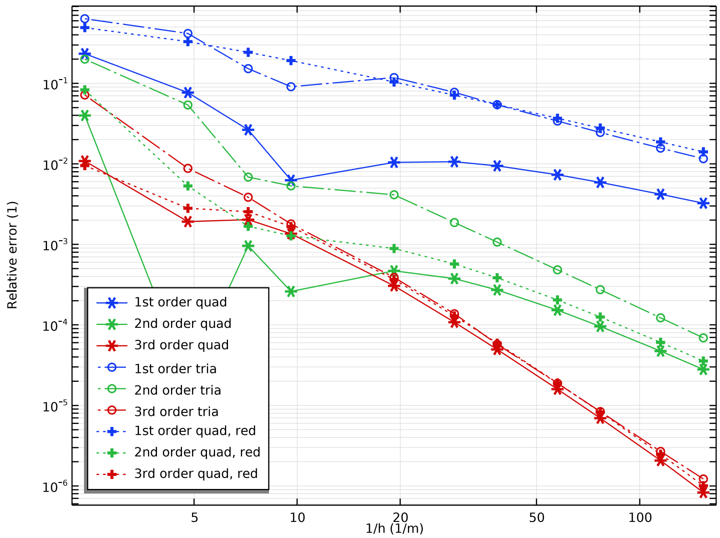

Figure 5: Error with respect to the stress target value as a function of the element size h.

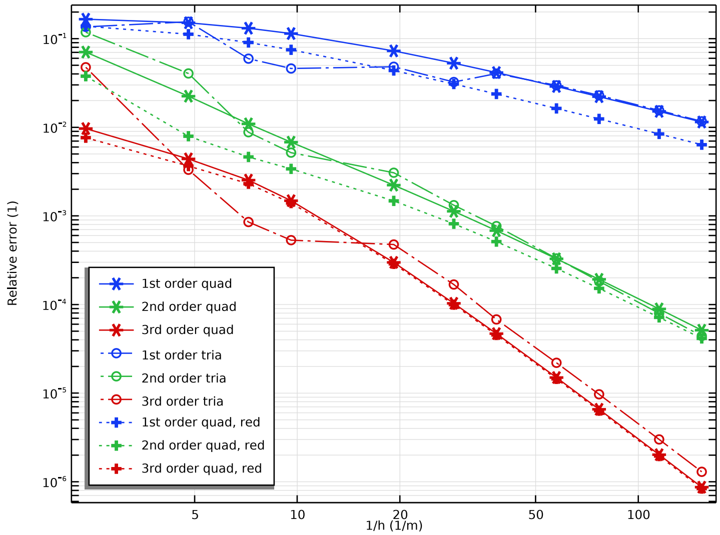

Figure 6: Error in the stress σ_x. The values are normalized with the target for σ_y.

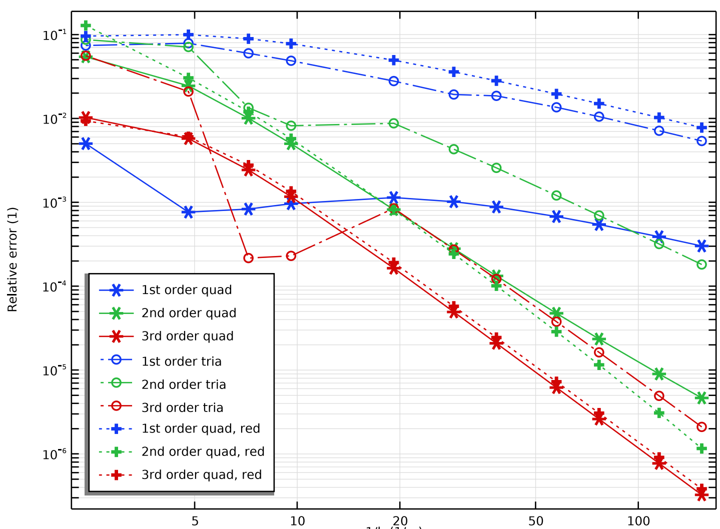

Figure 7: Error in the stress τ_{xy}. The values are normalized with the target for σ_y.

Since elements with different shape function orders are used, a comparison based only on element size may not be fair when efficiency is considered. The number of degrees of freedom in the model varies a lot for the same element size, and so does the solution time. In Figure 8, the error is shown as a function of the number of degrees of freedom. Also when compared this way, the elements with cubic shape functions have the best performance. This is usually true as long as the solutions are smooth, but it may not be true, for example, when solving nonlinear problems.

However, for coarser meshes, which are more representative or real life mesh resolution, the second order quadrilateral elements are quite efficient when compared versus the number of degrees of freedom.

Note that somewhat fewer degrees-of-freedom are used for when a reduced integration rule is used. This follows from the lower order of the auxiliary shape functions used for the out-of-plane displacement derivative.

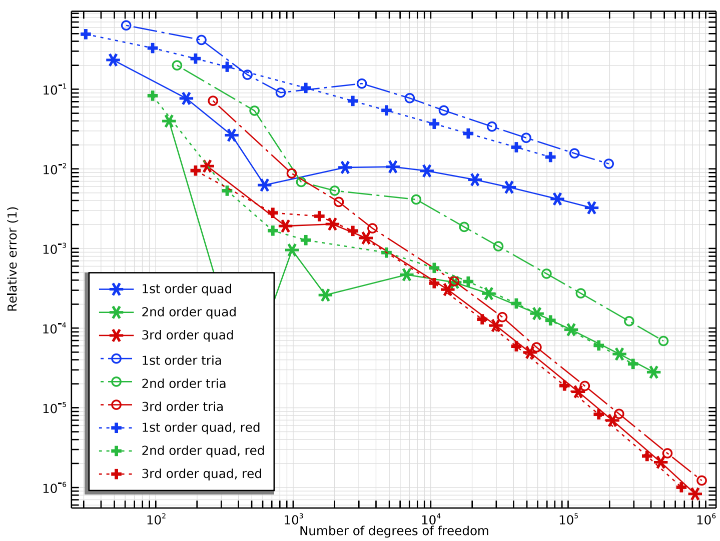

Figure 8: Error with respect to the stress target value as a function of the number of degrees of freedom.

Reference

1. G.A.O. Davies, R.T. Fenner, and R.W. Lewis, *Background to Benchmarks*, NAFEMS, Glasgow, 1993.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/elliptic_membrane`

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
div	1	1	Mesh refinement factor
sy_ref	92.65817[MPa]	9.2658E7 Pa	Target stress

GEOMETRY 1

Ellipse 1 (e1)

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

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

3 In the **Sector angle** text field, type 90.

4 In the **a-semiaxis** text field, type `3.25`.

5 In the **b-semiaxis** text field, type `2.75`.

Create an extra mesh control ellipse.

Ellipse 2 (e2)

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

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

3 In the **a-semiaxis** text field, type `2.417`.

4 In the **b-semiaxis** text field, type `1.583`.

5 In the **Sector angle** text field, type `90`.

Ellipse 3 (e3)

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

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

3 In the **a-semiaxis** text field, type `2`.

Difference 1 (dif1)

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

2 Select the objects **e1** and **e2** only.

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

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

5 Select the object **e3** only.

6 Click **Build All Objects**.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 Locate the **Starting Point** section. In the **x** text field, type `1.783` and **y** to `2.3`.

6 Locate the **Endpoint** section. In the **x** text field, type `1.165` and **y** to `0.812`.

Line Segment 2 (ls2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 Locate the **Starting Point** section. In the **x** text field, type `2.833` and **y** to `1.348`.

6 Locate the **Endpoint** section. In the **x** text field, type `1.783` and **y** to `0.453`.

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	210E3[MPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's ratio
Density	rho	0	kg/m³	Basic

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 `0.1`.

Add a second material to be able to switch between full and reduced integration.

Linear Elastic Material 1, Reduced Integration

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

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

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

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

5 In the **Label** text field, type `Linear Elastic Material 1, Reduced Integration`.

Symmetry 1

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

2 Select Boundaries 1, 2, 9, and 11 only.

Boundary Load 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
2 Select Boundaries 15, 18, and 21 only.
3 In the **Settings** window for **Boundary Load**, locate the **Force** section.
4 From the **Load type** list, choose **Pressure**.
5 In the p text field, type `-10[MPa]`.

MESH 1

Mapped 1

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

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.
2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
3 In the **Number of elements** text field, type `div`.
4 Select Boundaries 1, 2, 13, 16, and 19 only.
5 Click **Build All**.

The default discretization of the displacement field consists of quadratic serendipity shape functions. Change to Lagrange shape functions.

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**, click to expand the **Discretization** section.
3 From the **Displacement field** list, choose **Quadratic Lagrange**.

Add linear and cubic displacement fields as well. The actual selection of discretization type will be done in each study.

4 Click the **Show More Options** button in the **Model Builder** toolbar.
5 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.
6 Click **OK**.

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`.

5 Right-click **Discretization Linear** and choose **Duplicate**.

Discretization Cubic

1 In the **Model Builder** window, click **Discretization Linear 1**.

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

3 From the **Displacement field** list, choose **Cubic Lagrange**.

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

STUDY QUAD LINEAR

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

2 In the **Settings** window for **Study**, type `Study Quad Linear` in the **Label** text field.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
div (Mesh refinement factor)	1 2 3 4 8 12 16 24 32 48 64	

Step 1: Stationary

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

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

3 Select the **Modify model configuration for study step** check box.

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

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

ROOT

Add eight more studies for the other discretizations, integration orders, and element shapes. The parameter values are copied from the first study.

ADD STUDY

1 In the **Study** 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.

STUDY QUAD LINEAR

Parametric Sweep

In the **Model Builder** window, under **Study Quad Linear** right-click **Parametric Sweep** and choose **Copy**.

STUDY QUAD QUADRATIC

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

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

3 Right-click **Study Quad Quadratic** and choose **Paste Parametric Sweep**.

ADD STUDY

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

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

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

4 In the **Study** toolbar, click **Add Study** to close the **Add Study** window.

STUDY QUAD CUBIC

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

2 In the **Settings** window for **Study**, type `Study Quad Cubic` in the **Label** text field.

3 Right-click **Study Quad Cubic** and choose **Paste Parametric Sweep**.

Step 1: Stationary

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

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

3 Select the **Modify model configuration for study step** check box.

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

5 From the **Discretization** list, choose **Discretization Cubic**.

MESH 1

Create a triangular mesh. This mesh case will be the default for the new studies created from now on.

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Duplicate**.

MESH TRIA

1 In the **Model Builder** window, under **Component 1 (comp1)>Meshes** click **Mesh 2**.

2 In the **Settings** window for **Mesh**, type `Mesh Tria` in the **Label** text field.

Convert 1

In the **Mesh** toolbar, click **Modify** and choose **Convert**.

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.

STUDY TRIA LINEAR

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

2 In the **Settings** window for **Study**, type `Study Tria Linear` in the **Label** text field.

3 Right-click **Study Tria Linear** and choose **Paste Parametric Sweep**.

Step 1: Stationary

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

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

3 Select the **Modify model configuration for study step** check box.

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

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

6 Clear the **Modify model configuration for study step** check box.

ADD STUDY

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

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

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

STUDY TRIA QUADRATIC

1 In the **Model Builder** window, click **Study 5**.

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

3 Right-click **Study Tria Quadratic** and choose **Paste Parametric Sweep**.

ADD STUDY

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

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

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

STUDY TRIA CUBIC

1 In the **Model Builder** window, click **Study 6**.

2 In the **Settings** window for **Study**, type `Study Tria Cubic` in the **Label** text field.

3 Right-click **Study Tria Cubic** and choose **Paste Parametric Sweep**.

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

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

3 Select the **Modify model configuration for study step** check box.

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

5 From the **Discretization** list, choose **Discretization Cubic**.

ADD STUDY

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

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

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

STUDY QUAD LINEAR

Parametric Sweep

In the **Model Builder** window, under **Study Quad Linear** right-click **Parametric Sweep** and choose **Copy**.

STUDY QUAD LINEAR REDUCED

1 In the **Model Builder** window, click **Study 7**.

2 In the **Settings** window for **Study**, type `Study Quad Linear Reduced` in the **Label** text field.

3 Right-click **Study Quad Linear Reduced** and choose **Paste Parametric Sweep**.

Step 1: Stationary

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

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

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Linear Elastic Material 2**.

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

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

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

8 Click to expand the **Mesh Selection** section. In the table, enter the following settings:

Component	Mesh
Component 1	Mesh 1

ADD STUDY

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

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

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

STUDY QUAD LINEAR

Parametric Sweep

In the **Model Builder** window, under **Study Quad Linear** right-click **Parametric Sweep** and choose **Copy**.

STUDY QUAD QUADRATIC REDUCED

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

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

3 Right-click **Study Quad Quadratic Reduced** and choose **Paste Parametric Sweep**.

Step 1: Stationary

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

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

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Linear Elastic Material 2**.

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

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

Component	Mesh
Component 1	Mesh 1

ADD STUDY

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

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

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

4 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY QUAD LINEAR

Parametric Sweep

In the **Model Builder** window, under **Study Quad Linear** right-click **Parametric Sweep** and choose **Copy**.

STUDY QUAD CUBIC REDUCED

1 In the **Model Builder** window, click **Study 9**.

2 In the **Settings** window for **Study**, type `Study Quad Cubic Reduced` in the **Label** text field.

3 Right-click **Study Quad Cubic Reduced** and choose **Paste Parametric Sweep**.

Step 1: Stationary

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

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

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Linear Elastic Material 2**.

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

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

7 From the **Discretization** list, choose **Discretization Cubic**.

8 Locate the **Mesh Selection** section. In the table, enter the following settings:

Component	Mesh
Component 1	Mesh 1

STUDY QUAD LINEAR

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

STUDY QUAD QUADRATIC

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

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

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

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

STUDY QUAD CUBIC

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

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

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

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

STUDY TRIA LINEAR

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

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

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

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

STUDY TRIA QUADRATIC

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

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

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

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

STUDY TRIA CUBIC

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

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

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

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

STUDY QUAD LINEAR REDUCED

1 In the **Model Builder** window, click **Study Quad Linear Reduced**.

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

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

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

STUDY QUAD QUADRATIC REDUCED

1 In the **Model Builder** window, click **Study Quad Quadratic Reduced**.

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

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

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

STUDY QUAD CUBIC REDUCED

1 In the **Model Builder** window, click **Study Quad Cubic Reduced**.

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

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

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

RESULTS

Mesh Convergence sy at D

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

2 In the **Settings** window for **1D Plot Group**, type `Mesh Convergence sy at D` in the **Label** text field.

Point Graph 1

1 Right-click **Mesh Convergence sy at D** and choose **Point Graph**.

2 Select Point 11 only.

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

4 From the **Dataset** list, choose **Study Quad Linear/Parametric Solutions 1 (sol2)**.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `abs(solid.sGpy/sy_ref-1)`.

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

7 In the **Expression** text field, type `div/0.417`.

8 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.

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

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

11 In the table, enter the following settings:

Legends
1st order quad

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

Point Graph 2

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

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

3 From the **Dataset** list, choose **Study Quad Quadratic/Parametric Solutions 2 (sol15)**.

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

Legends
2nd order quad

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

Point Graph 3

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

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

3 From the **Dataset** list, choose **Study Quad Cubic/Parametric Solutions 3 (sol28)**.

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

Legends
3rd order quad

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

Point Graph 4

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

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

3 From the **Dataset** list, choose **Study Tria Linear/Parametric Solutions 4 (sol41)**.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dash-dot**.

5 From the **Color** list, choose **Cycle (reset)**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

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

Legends
1st order tria

8 Right-click **Point Graph 4** and choose **Duplicate**.

Point Graph 5

1 In the **Model Builder** window, click **Point Graph 5**.

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

3 From the **Dataset** list, choose **Study Tria Quadratic/Parametric Solutions 5 (sol54)**.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **Cycle**.

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

Legends
`2nd order tria`

6 Right-click **Point Graph 5** and choose **Duplicate**.

Point Graph 6

1 In the **Model Builder** window, click **Point Graph 6**.

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

3 From the **Dataset** list, choose **Study Tria Cubic/Parametric Solutions 6 (sol67)**.

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

Legends
`3rd order tria`

5 Right-click **Point Graph 6** and choose **Duplicate**.

Point Graph 7

1 In the **Model Builder** window, click **Point Graph 7**.

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

3 From the **Dataset** list, choose **Study Quad Linear Reduced/Parametric Solutions 7 (sol80)**.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.

5 From the **Color** list, choose **Cycle (reset)**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Plus sign**.

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

Legends
`1st order quad, red`

8 Right-click **Point Graph 7** and choose **Duplicate**.

Point Graph 8

1 In the **Model Builder** window, click **Point Graph 8**.

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

3 From the **Dataset** list, choose **Study Quad Quadratic Reduced/ Parametric Solutions 8 (sol93)**.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **Cycle**.

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

Legends
`2nd order quad, red`

6 Right-click **Point Graph 8** and choose **Duplicate**.

Point Graph 9

1 In the **Model Builder** window, click **Point Graph 9**.

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

3 From the **Dataset** list, choose **Study Quad Cubic Reduced/Parametric Solutions 9 (sol106)**.

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

Legends
`3rd order quad, red`

Mesh Convergence sy at D

1 In the **Model Builder** window, click **Mesh Convergence sy at D**.

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 `1/h (1/m)`.

4 Select the **y-axis label** check box. In the associated text field, type `Relative error (1)`.

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

6 Locate the **Axis** section. Select the **x-axis log scale** check box.

7 Select the **y-axis log scale** check box.

8 Locate the **Legend** section. From the **Position** list, choose **Lower left**.

9 In the **Mesh Convergence sy at D** toolbar, click  **Plot**.

10 Right-click **Mesh Convergence sy at D** and choose **Duplicate**.

Mesh Convergence sx at D

1 In the **Model Builder** window, under **Results** click **Mesh Convergence sy at D 1**.

2 In the **Settings** window for **1D Plot Group**, type `Mesh Convergence sx at D` in the **Label** text field.

FIND AND REPLACE

1 Press Ctrl+F.

2 Go to the **Find and Replace** window.

3 In the **Contains** text field, type `abs(solid.sGpy/sy_ref-1)`.

4 In the **Replace with** text field, type `abs(solid.sGpx/sy_ref)`.

5 Click next to **Add Filter**, then choose **Node Filter**.

6 In the **Node Filter** dialog box, type `sx` in the **Find** text field.

7 Click **OK**.

8 Go to the **Find and Replace** window.

9 Click **Search**.

10 Click next to **Replace**, then choose **Replace All**.

RESULTS

Point Graph 1

1 In the **Model Builder** window, under **Results>Mesh Convergence sx at D** click **Point Graph 1**.

2 In the **Mesh Convergence sx at D** toolbar, click **Plot**.

Mesh Convergence sx at D

In the **Model Builder** window, right-click **Mesh Convergence sx at D** and choose **Duplicate**.

Mesh Convergence sxy at D

1 In the **Model Builder** window, under **Results** click **Mesh Convergence sx at D 1**.

2 In the **Settings** window for **1D Plot Group**, type `Mesh Convergence sxy at D` in the **Label** text field.

FIND AND REPLACE

1 Press Ctrl+F.

2 Go to the **Find and Replace** window.

3 In the **Contains** text field, type `abs(solid.sGpx/sy_ref)`.

4 In the **Replace with** text field, type `abs(solid.sGpxy/sy_ref)`.

5 Click next to **Add Filter**, then choose **Node Filter**.

6 In the **Node Filter** dialog box, type `sxy` in the **Find** text field.

7 Click **OK**.

8 Go to the **Find and Replace** window.

9 Click **Search**.

10 Click next to **Replace**, then choose **Replace All**.

RESULTS

Point Graph 1

1 In the **Model Builder** window, under **Results>Mesh Convergence sxy at D** click **Point Graph 1**.

2 In the **Mesh Convergence sxy at D** toolbar, click **Plot**.

Mesh Convergence sy at D

In the **Model Builder** window, under **Results** right-click **Mesh Convergence sy at D** and choose **Duplicate**.

Mesh Convergence sy at D (by DOFs)

1 In the **Model Builder** window, under **Results** click **Mesh Convergence sy at D 1**.

2 In the **Settings** window for **1D Plot Group**, type `Mesh Convergence sy at D (by DOFs)` in the **Label** text field.

FIND AND REPLACE

1 Press Ctrl+F.

2 Go to the **Find and Replace** window.

3 In the **Contains** text field, type `div/0.417`.

4 In the **Replace with** text field, type `numberofdofs`.

5 Click next to **Add Filter**, then choose **Node Filter**.

6 In the **Node Filter** dialog box, type `by DOFs` in the **Find** text field.

7 Click **OK**.

8 Go to the **Find and Replace** window.

9 Click **Search**.

10 Click next to **Replace**, then choose **Replace All**.

RESULTS

Mesh Convergence sy at D (by DOFs)

1 In the **Model Builder** window, expand the **Mesh Convergence sy at D (by DOFs)** node, then click **Results>Mesh Convergence sy at D (by DOFs)**.

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

3 In the **x-axis label** text field, type `Number of degrees of freedom`.

4 In the **Mesh Convergence sy at D (by DOFs)** toolbar, click **Plot**.

Evaluation Group 1

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, locate the **Transformation** section.

3 Select the **Transpose** check box.

Point Evaluation 1

1 Right-click **Evaluation Group 1** and choose **Point Evaluation**.

2 Select Point 11 only.

3 In the **Settings** window for **Point Evaluation**, locate the **Data** section.

4 From the **Dataset** list, choose **Study Quad Linear/Parametric Solutions 1 (sol2)**.

5 From the **Parameter selection (div)** list, choose **From list**.

6 In the **Parameter values (div)** list, choose **1** and **2**.

7 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Stress>Stress tensor (spatial frame) - N/m²>solid.sGpyy - Stress tensor, yy-component**.

8 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`solid.sGpyy`	`MPa`	`Stress, quad linear`
`(solid.sGpyy-sy_ref)/sy_ref`	`%`	`Relative error`

9 Right-click **Point Evaluation 1** and choose **Duplicate**.

Point Evaluation 2

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

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

3 From the **Dataset** list, choose **Study Quad Quadratic/Parametric Solutions 2 (sol15)**.

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

Expression	Unit	Description
`solid.sGpyy`	`MPa`	`Stress, quad quadratic`

5 Right-click **Point Evaluation 2** and choose **Duplicate**.

Point Evaluation 3

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

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

3 From the **Dataset** list, choose **Study Quad Cubic/Parametric Solutions 3 (sol28)**.

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

Expression	Unit	Description
solid.sGpyy	MPa	Stress, quad cubic

5 Right-click **Point Evaluation 3** and choose **Duplicate**.

Point Evaluation 4

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

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

3 From the **Dataset** list, choose **Study Tria Linear/Parametric Solutions 4 (sol41)**.

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

Expression	Unit	Description
solid.sGpyy	MPa	Stress, tria linear

5 Right-click **Point Evaluation 4** and choose **Duplicate**.

Point Evaluation 5

1 In the **Model Builder** window, click **Point Evaluation 5**.

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

3 From the **Dataset** list, choose **Study Tria Quadratic/Parametric Solutions 5 (sol54)**.

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

Expression	Unit	Description
solid.sGpyy	MPa	Stress, tria quadratic

5 Right-click **Point Evaluation 5** and choose **Duplicate**.

Point Evaluation 6

1 In the **Model Builder** window, click **Point Evaluation 6**.

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

3 From the **Dataset** list, choose **Study Tria Cubic/Parametric Solutions 6 (sol67)**.

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

Expression	Unit	Description
solid.sGpyy	MPa	Stress, tria cubic

5 Right-click **Point Evaluation 6** and choose **Duplicate**.

Point Evaluation 7

1 In the **Model Builder** window, click **Point Evaluation 7**.

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

3 From the **Dataset** list, choose **Study Quad Linear Reduced/Parametric Solutions 7 (sol80)**.

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

Expression	Unit	Description
solid.sGpyy	MPa	Stress, quad linear, red

5 Right-click **Point Evaluation 7** and choose **Duplicate**.

Point Evaluation 8

1 In the **Model Builder** window, click **Point Evaluation 8**.

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

3 From the **Dataset** list, choose **Study Quad Quadratic Reduced/ Parametric Solutions 8 (sol93)**.

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

Expression	Unit	Description
solid.sGpyy	MPa	Stress, quad quadratic, red

5 Right-click **Point Evaluation 8** and choose **Duplicate**.

Point Evaluation 9

1 In the **Model Builder** window, click **Point Evaluation 9**.

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

3 From the **Dataset** list, choose **Study Quad Cubic Reduced/Parametric Solutions 9 (sol106)**.

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

Expression	Unit	Description
solid.sGpyy	MPa	Stress, quad cubic, red

5 In the **Evaluation Group 1** toolbar, click = **Evaluate**.

Stress (solid)

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

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

3 From the **Dataset** list, choose **Study Quad Quadratic/Parametric Solutions 2 (sol15)**.

4 From the **Parameter value (div)** list, choose **4**.

Surface 1

1 In the **Model Builder** window, expand the **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>Stress>Stress tensor (spatial frame) - N/m^2>solid.sGpyy - Stress tensor, yy-component**.

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

4 In the **Stress (solid)** toolbar, click **Plot**.

Stress (solid)

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

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

3 Select the **Show maximum and minimum values** check box.

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