Examples of Structural Mechanics Models

Two structural mechanics benchmark models show how to perform a linear static stress analysis using:

- A uniformly distributed horizontal load along an outer edge
- A gravity loading

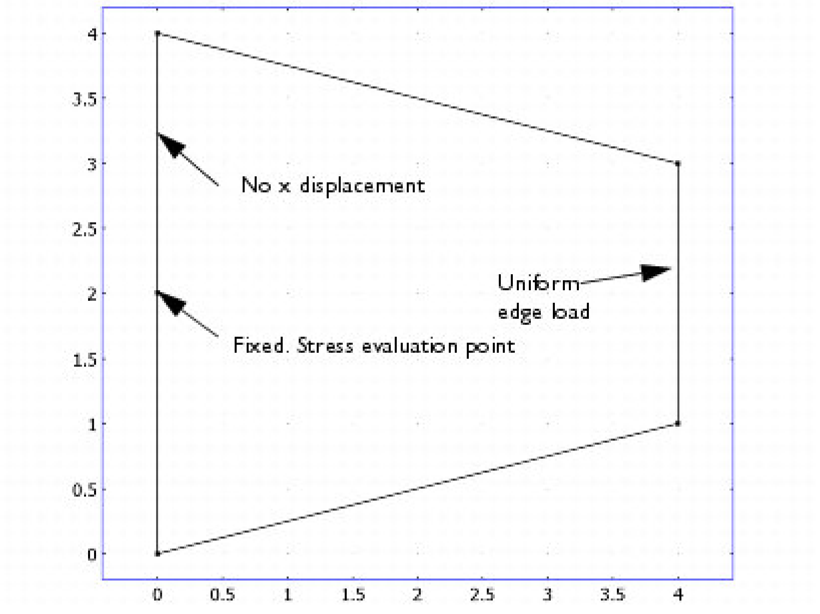
The examples are taken from a NAFEMS benchmark collection (Ref. 1).

Tapered Membrane End Load

The first example shows a 2D plane stress model of a membrane with a thickness of 0.1 m. The load is a uniformly distributed horizontal load of 10 MN/m (that is, a pressure of 100 MPa) along the right end. At the left end, there is no displacement in the x direction. Also, at a midpoint location, the left end is fixed also in the y direction.

The model uses the following material properties:

- The material is isotropic.
- The Young's modulus (elasticity modulus) is $210 \cdot 10^3$ MPa.
- The Poisson ratio is 0.3.



Modeling in COMSOL Multiphysics

Using a Plane Stress application modes and a static analysis, it is straightforward to perform this stress analysis. The finite element model uses second-order triangular Lagrange elements. To show convergence toward the benchmark value, refine the mesh and recompute the solution twice.

Results

The solution shows an x-direction stress at the point (0, 2) that is in good agreement with the benchmark target value of 61.3 MPa. Using the initial mesh, the COMSOL Multiphysics solution gives a value of 61.41 MPa. Two successive mesh refinements provide stress values of 61.36 MPa and 61.35 MPa.

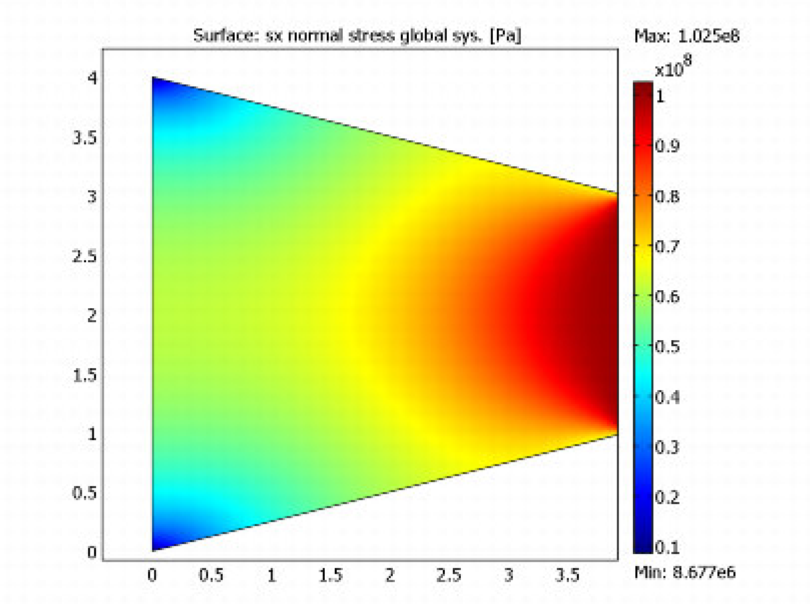


Figure 8-1: The x-direction stress distribution with a uniform edge load

---

Model Library Path: `COMSOL_Multiphysics/Structural_Mechanics/edge_load_2d`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **2D** in the **Space dimension** list.

2 In the list of application modes, open the **COMSOL Multiphysics>Structural Mechanics** folder and then the **Plane Stress** node. Select **Static analysis**.

3 Click **OK**.

GEOMETRY MODELING

1 On the **Draw** menu, point to **Specify Objects** and then click **Line**.

2 In the **Line** dialog box, type `0 4 4 0 0` in the **x** edit field and `0 1 3 4 0` in the **y** edit field.

3 Click **OK**.

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

5 Click the **Coerce to Solid** button in the Draw toolbar.

6 On the **Draw** menu, point to **Specify Objects** and then click **Point**.

7 In the **Point** dialog box, type `0` in the **x** edit field and `2` in the **y** edit field.

8 Click **OK**.

The point is the location for the point constraint and also the benchmark value for the stress.

PHYSICS SETTINGS

Boundary and Point Conditions—Loads and Constraints

1 From the **Physics** menu, choose **Boundary Settings**.

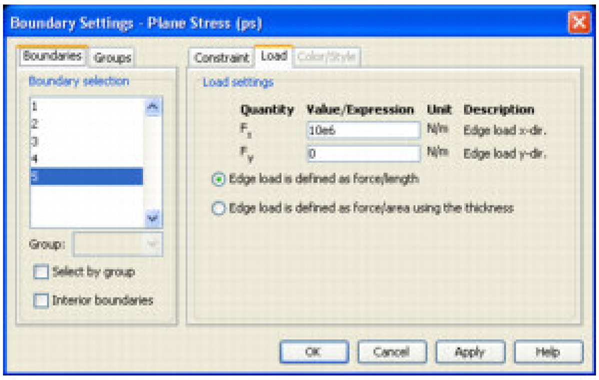
2 Select boundaries 1 and 3 in the **Boundary selection** list.

3 Select the **Rx** check box.

4 Click the **Load** tab.

5 Select boundary 5 in the **Boundary selection** list.

6 Type `10e6` in the **Fx** edit field.



7 Click **OK**.

8 From the **Physics** menu, choose **Point Settings**.

9 Select point 2 in the **Point selection** list.

10 Select the **Rx** and **Ry** check boxes.

11 Click **OK**.

Subdomain Settings—Material Properties

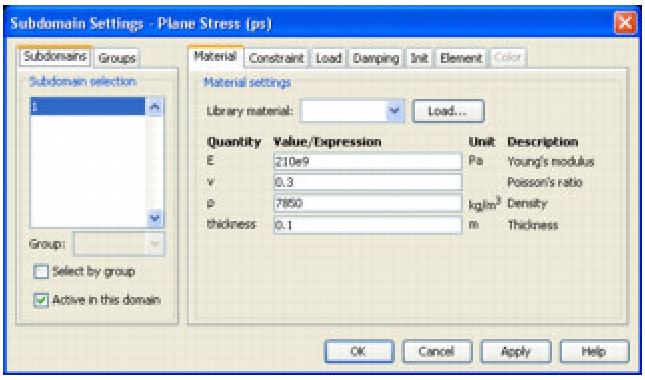
1 From the **Physics** menu, choose **Subdomain Settings**.

2 Select subdomain 1 in the **Subdomain selection** list.

3 Type `210e9` in the **E** edit field for the Young's modulus.

4 Type `0.3` in the v edit field for the Poisson's modulus.

5 Type `0.1` in the **thickness** edit field.



6 Click **OK**.

The other material properties are not used in this model.

MESH GENERATION

Click the **Initialize Mesh** button.

COMPUTING THE SOLUTION

Click the **Solve** button.

POSTPROCESSING AND VISUALIZATION

The default plot shows the von Mises stress. To plot the x-direction stress, use the following steps:

1 From the **Postprocessing** menu, select **Plot Parameters**.

2 In the **Plot Parameters** dialog box, click the **Surface** tab.

3 Under **Surface data**, select **sx normal stress global sys.** in the **Predefined quantities** list.

4 Click **OK**.

To get a better look at the value of the x-direction stress at the point (0,2), use the **Cross-Section Plot Parameters** dialog box:

1 From the **Postprocessing** menu, select **Cross-Section Plot Parameters**.

2 Click the **Point** tab.

3 Select **sx normal stress global sys.** in the **Predefined quantities** list.

4 Type 0 in the **x** edit field and 2 in the **y** edit field.

5 Click **OK**.

This plot shows a straight line representing the value of the x-direction stress at the point (0,2). You can also click at that point in the results plot. That prints the numeric value in the Message log.

Tapered Cantilever Gravity Load

The second example shows another 2D plane stress model of a membrane with a thickness of 0.1 m. The load is a gravity load that acts in the negative y direction with an acceleration of 9.81 m/s^2. The left end boundary is fully fixed (no displacements).

The model uses the following material properties:

- The material is isotropic.
- The Young's modulus (elasticity modulus) is $210 \cdot 10^3$ MPa.
- The Poisson ratio is 0.3.
- The density is 7000 kg/m^3.

Modeling in COMSOL Multiphysics

Using a Plane Stress application modes and a static analysis, it is straightforward to perform this stress analysis. You enter the gravity load as force/volume. COMSOL Multiphysics then computes the load using the thickness of the material. The finite element model uses second-order triangular Lagrange elements. For postprocessing, use a cross-section plot to show the value of the shear stress at the location (0,2).

Results

The solution shows a shear stress (s_{xy}) at the point (0, 2) that is in good agreement with the benchmark target value of −0.200 MPa. Using the initial mesh, the COMSOL Multiphysics solution gives a value of −0.199 MPa.

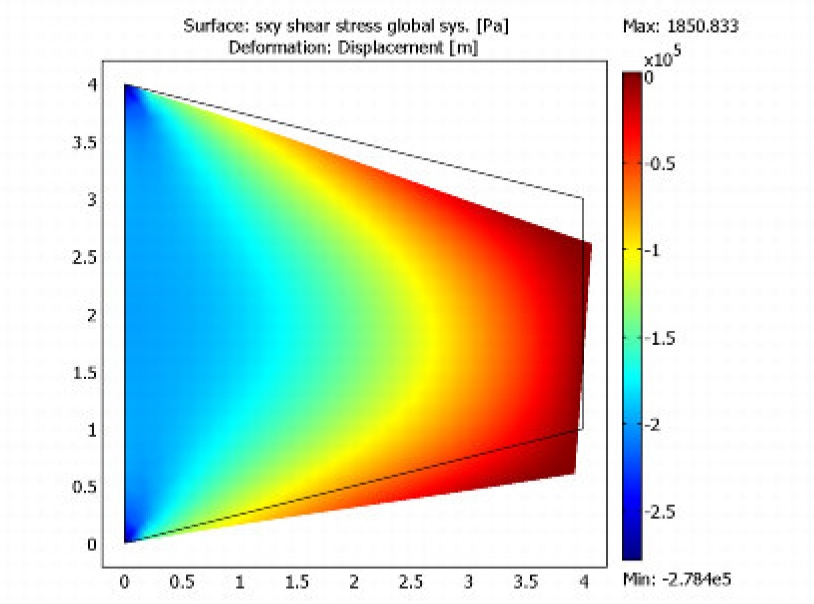


Figure 8-2: The shear stress and the displacement (exaggerated) from a gravity load.

Model Library Path: `COMSOL_Multiphysics/Structural_Mechanics/gravity_load_2d`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Select **2D** in the **Space dimension** list.

2 In the list of application modes, open the **COMSOL Multiphysics>Structural Mechanics** folder and then the **Plane Stress** node. Select **Static analysis**.

3 Click **OK**.

GEOMETRY MODELING

1 On the **Draw** menu, point to **Specify Objects** and then click **Line**.

2 In the **Line** dialog box, type `0 4 4 0 0` in the **x** edit field and `0 1 3 4 0` in the **y** edit field.

3 Click **OK**.

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

5 Click the **Coerce to Solid** button in the Draw toolbar.

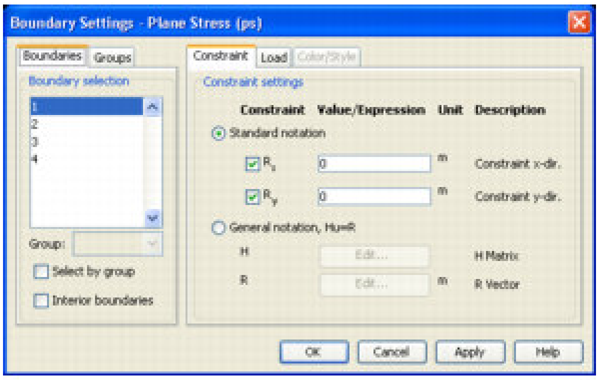
PHYSICS SETTINGS

Boundary Conditions—Constraints

1 From the **Physics** menu, choose **Boundary Settings**.

2 Select boundary 1 in the **Boundary selection** list.

3 Select the **Rx** and **Ry** check boxes.



4 Click **OK**.

Subdomain Settings—Material Properties and Loads

1 From the **Physics** menu, choose **Subdomain Settings**.

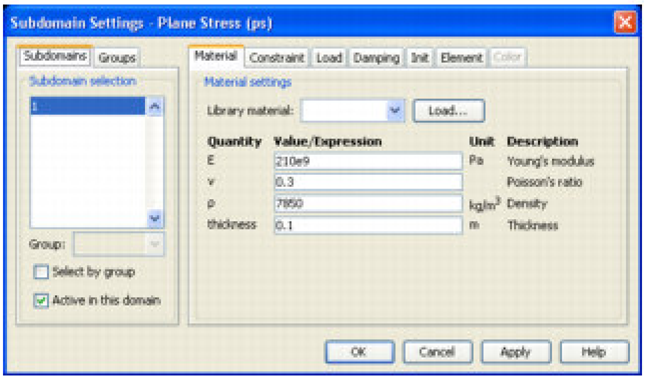
2 Select subdomain 1 in the **Subdomain selection** list.

3 Type `210e9` in the **E** edit field for the Young's modulus.

4 Type `0.3` in the ν edit field for the Poisson's ratio.

5 Type `0.1` in the **thickness** edit field.

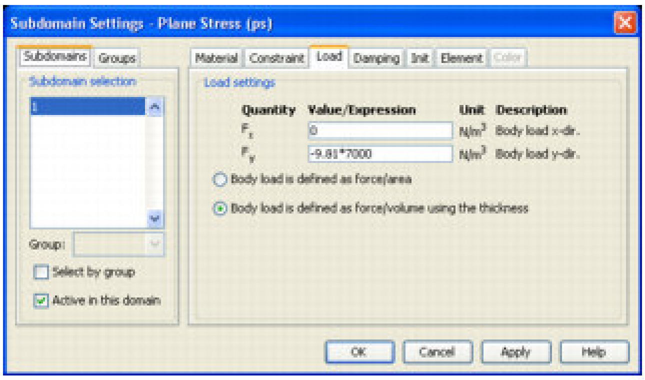
You can also type `7000` in the ρ edit field for the density, but the density value from this edit field is not used in the static analysis. The gravity appears in the specification of the gravity load below.



6 Click the **Load** tab.

7 Type `-9.81*7000` in the **Fy** edit field for the y-direction body load.

8 Click the **Body load is defined as force/volume using the thickness** button.



9 Click **OK**.

MESH GENERATION

Click the **Initialize Mesh** button.

COMPUTING THE SOLUTION

Click the **Solve** button.

POSTPROCESSING AND VISUALIZATION

The default plot shows the von Mises stress. To plot the shear stress and also the deformed shape (displacements), use the following steps:

1. From the **Postprocessing** menu, select **Plot Parameters**.
2. In the **Plot Parameters** dialog box, click the **Surface** tab.
3. Under **Surface data**, select **sxy shear stress global sys.** in the **Predefined quantities** list.
4. Click the **Deform** tab.
5. Select the **Deformed shape plot** check box. The deformation data is the x and y displacement by default.
6. Click **OK**.

The deformed shape plot uses a scaling factor to clearly show the deformation. The actual displacements are small compared to the model geometry. Click the **Deform** tab too see the scale factor or to enter a different scale factor.

To get a better look at the value of the shear stress at the point (0,2), use the **Cross-Section Plot Parameters** dialog box:

1. From the **Postprocessing** menu, select **Cross-Section Plot Parameters**.
2. Click the **Point** tab.
3. Select **sxy shear stress global sys.** in the **Predefined quantities** list.
4. Type `0` in the **x** edit field and `2` in the **y** edit field.
5. Click **OK**.

The plot shows a straight line at the value of the shear stress at (0,2). You can also click at that point in the results plot. That prints the numeric value in the Message log.

Reference

1. D. Hitchings, A. Kamoulakos, G. A. O. Davies: *Linear Statics Benchmarks Vol. 1*, NAFEMS Ltd., Glasgow, 1987.