Model created in COMSOL Multiphysics 6.4

Thick Plate Stress Analysis

Introduction

This example implements the static stress analysis described in the NAFEMS Test No LE10, "Thick Plate Pressure," found on page 77 in the NAFEMS report *Background to Benchmarks* (Ref. 1). The computed stress level is compared with the values given in the benchmark report.

Model Definition

The geometry is an ellipse with an ellipse-shaped hole in it. Due to symmetry in load and in geometry, the analysis only includes a quarter of the ellipse.

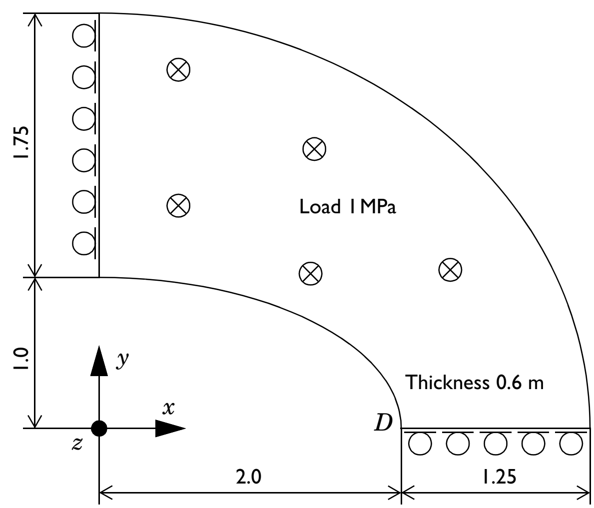


Figure 1: The thick plate geometry, reduced to a quarter of the ellipse due to symmetry.

MATERIAL

Isotropic with $E = 210$ GPa, $\nu = 0.3$.

LOAD

A distributed load of 1 MPa on the upper surface pointing in the negative z direction.

CONSTRAINTS

- Symmetry planes, $x = 0$, $y = 0$.

- Outer ellipse surface constrained in the x and y directions.
- Midplane on outer ellipse surface constrained in the z direction.

Results

The normal stress σ_y is evaluated on the top surface at the inside of the elliptical hole, point D in Figure 1 with coordinate (2, 0, 0.6). It is in good agreement with the NAFEMS benchmark (Ref. 1), considering the coarse mesh. The difference is about 6%.

RESULT	COMSOL MULTIPHYSICS	NAFEMS (Ref. 1)
σ_y (at D)	-5.72 MPa	-5.38 MPa

The y-component of the stress is shown in Figure 2.

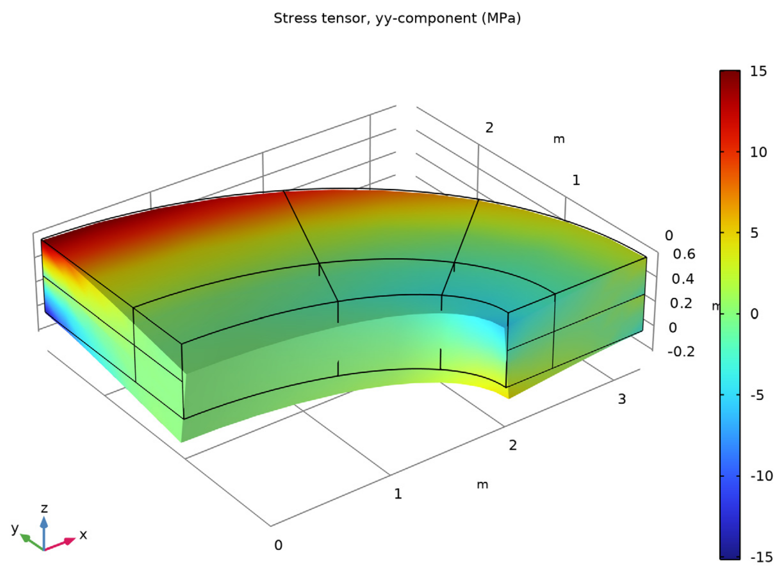


Figure 2: The stress in the y direction.

A note about this example is that the z direction constraint is applied to an edge only. This is a singular constraint, which causes local stresses at the constrained edge. These stresses are unlimited from a theoretical point of view, and in practice the stresses and vertical displacements are strongly mesh dependent. This does not invalidate the possibility to determine stresses at a distance far away from the singular constraint.

Notes About the COMSOL Implementation

In order to get the same mesh as in the original benchmark, some extra lines are drawn in the 2D geometry. As an effect, there will be several domains. This approach is efficient in this simple example, whereas for more complex geometries, the use of **Mesh Control Domains** should be considered.

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/thick_plate`

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 **3D**.
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

If you do not want to build all the geometry, you can load the geometry sequence from the stored model. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Geometry 1** and choose **Insert Sequence**. Browse to the model's Application Libraries folder and double-click the file **thick_plate.mph**. You can then continue to the **Add Material** section below.

To build the geometry from scratch, continue here.

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, click **Go to Plane Geometry**.

Work Plane 1 (wp1) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1) > Ellipse 1 (e1)

1 In the **Work Plane** toolbar, click **Ellipse**.

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

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

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

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

6 Click **Build Selected**.

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

Work Plane 1 (wp1) > Ellipse 2 (e2)

1 In the **Work Plane** 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`.

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

5 Click **Build Selected**.

Work Plane 1 (wp1) > Ellipse 3 (e3)

1 In the **Work Plane** toolbar, click **Ellipse**.

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

3 In the **a-semiaxis** text field, type `2.416`.

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

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

6 Click **Build Selected**.

Work Plane 1 (wp1) > Difference 1 (dif1)

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

2 Select the objects **e1** and **e3** 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 **e2** only.

6 Click **Build Selected**.

Work Plane 1 (wp1) > Polygon 1 (pol1)

1 In the **Work Plane** toolbar, click **Polygon**.

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

3 From the **Type** list, choose **Open curve**.

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

xw (m)	yw (m)
1.783	2.3
1.165	0.812

Work Plane 1 (wp1) > Polygon 2 (pol2)

1 In the **Work Plane** toolbar, click **Polygon**.

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

3 From the **Type** list, choose **Open curve**.

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

xw (m)	yw (m)
2.833	1.348
1.783	0.453

5 In the **Work Plane** toolbar, click **Build All**.

Work Plane 1 (wp1) > Plane Geometry

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

Work Plane 1 (wp1) > Partition Objects 1 (par1)

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

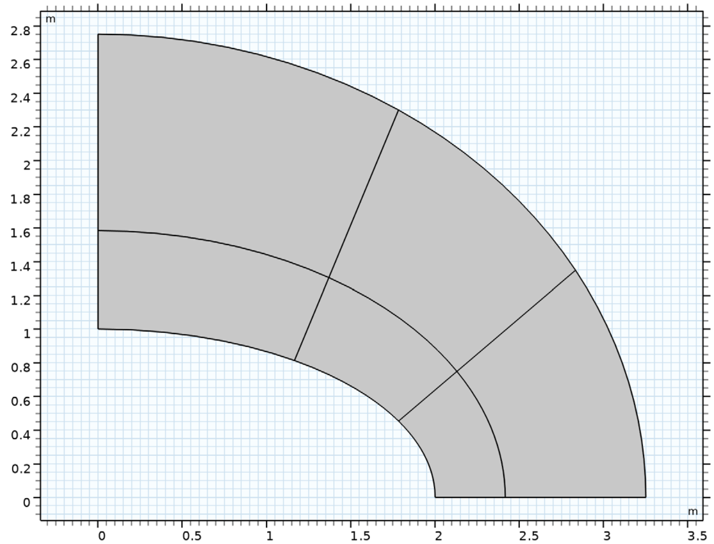
2 Select the object **dif1** only.

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

4 Click to select the **Activate Selection** toggle button for **Tool objects**.

5 Select the objects **pol1** and **pol2** only.

6 Click **Build Selected**.



Extrude 1 (ext1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Work Plane 1 (wp1)** and choose **Extrude**.

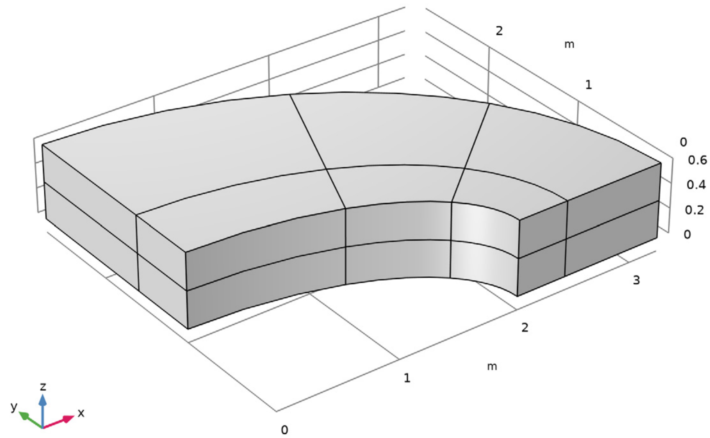
2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (m)
0.3
0.6

4 Click **Build Selected**.

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



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	210[GPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's ratio
Density	rho	7850	kg/m³	Basic

SOLID MECHANICS (SOLID)

Symmetry 1

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

2 Select Boundaries 1, 4, 8, 11, 40, 41, 49, and 50 only.

Prescribed Displacement 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.
2. Select Boundaries 15, 16, 31, 32, 51, and 52 only.
3. In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.
4. From the **Displacement in x direction** list, choose **Prescribed**.
5. From the **Displacement in y direction** list, choose **Prescribed**.

Prescribed Displacement 2

1. In the **Physics** toolbar, click **Edges** and choose **Prescribed Displacement**.
2. Select Edges 20, 41, and 72 only.
3. In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.
4. From the **Displacement in z direction** list, choose **Prescribed**.

Boundary Load 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
2. Select Boundaries 7, 14, 23, 30, 39, and 48 only.
3. In the **Settings** window for **Boundary Load**, locate the **Force** section.
4. Specify the $\mathbf{f}_A$ vector as

0	x
0	y
-1[MPa]	z

MESH 1

Mapped 1

In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.

Distribution 1

Right-click **Mapped 1** and choose **Distribution**.

Mapped 1

Select Boundaries 7, 14, 23, 30, 39, and 48 only.

Distribution 1

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

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

3 In the **Number of elements** text field, type 2.

4 Locate the **Edge Selection** section. From the **Selection** list, choose **All edges**.

5 Click **Build Selected**.

Swept 1

1 In the **Mesh** toolbar, click **Swept**.

2 In the **Settings** window for **Swept**, click **Build All**.

STUDY 1

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

Set default units for result presentation.

RESULTS

Preferred Units 1

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

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

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, select **Solid Mechanics** > **Stress tensor (N/m^2)** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Preferred Units**, locate the **Units** section.

7 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Stress tensor	N/m^2	MPa

8 Select the **Apply conversions to expressions with the same dimensions** checkbox.

9 Click **Apply**.

Point Evaluation 1

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

2 Select Point 24 only.

This corresponds to point D in Figure 1.

3 In the **Settings** window for **Point Evaluation**, 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**.

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

Expression	Unit	Description
solid.sGpyy	MPa	Stress tensor, y-component (COMSOL)
-5.38[MPa]	MPa	Stress tensor, y-component (NAFEMS)

5 Click **Evaluate**.

Stress (solid)

Modify the default surface plot to show the y-component of the stress tensor.

Volume 1

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

2 In the **Settings** window for **Volume**, 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/m²** > **solid.sGpyy - Stress tensor, yy-component**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **Rainbow**.

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