Kirsch Infinite Plate Problem

Introduction

In this model you perform a static stress analysis to obtain the stress distribution in the vicinity of a small hole in an infinite plate. The problem is a classic benchmark, and the theoretical solution was derived by G. Kirsch in 1898. This implementation is based on the Kirsch plate model described on page 184 in *Mechanics of Materials* D. Roylance (Ref. 1). The stress level is compared with the theoretical values.

Model Definition

Model the infinite plate in a 2D plane stress approximation as a 2 m-by-2 m plate with a hole with a radius of 0.1 m in the middle. Due to symmetry in load and geometry you need to analyze only a quarter of the plate.

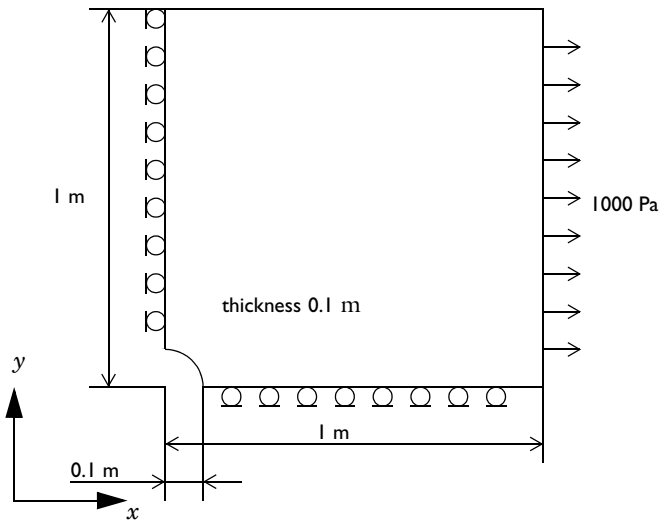

Figure 1: Geometry model of the Kirsch plate, making use of the symmetry.

MATERIAL

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

LOAD

A distributed stress of 10^3 Pa on the right edge pointing in the x direction.

CONSTRAINTS

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

Results

The distribution of the normal stress in the x-direction, σ_x, is shown in Figure 2.

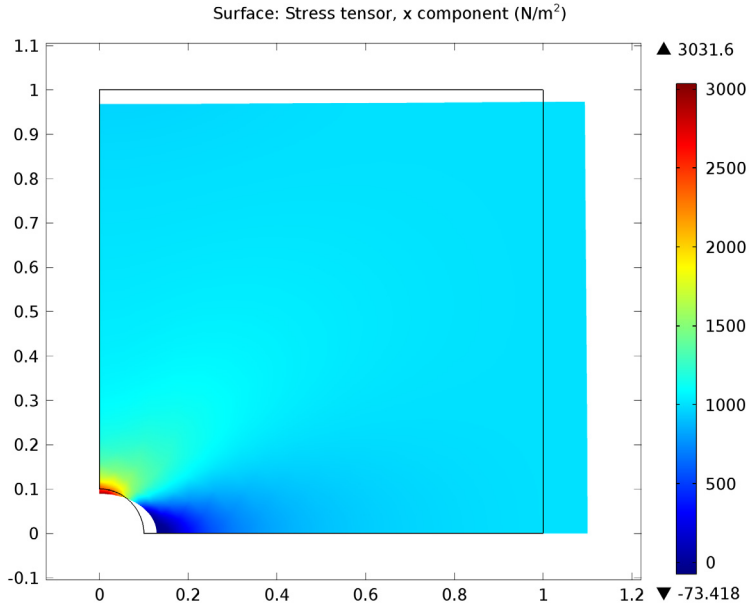

Figure 2: Distribution of the normal stress in the x direction.

According to Ref. 1 the stress σ_x along the vertical symmetry line can be calculated as

$$\sigma_x = \frac{1000}{2}\left(2 + \frac{0{,}1^2}{y^2} + 3\frac{0{,}1^4}{y^4}\right) \tag{1}$$

©2012 COMSOL

Figure 3 compares the stress σ_x obtained from the solved model, and plotted as a function of the y-coordinate along the left symmetry edge, to the theoretical value according to Equation 1.

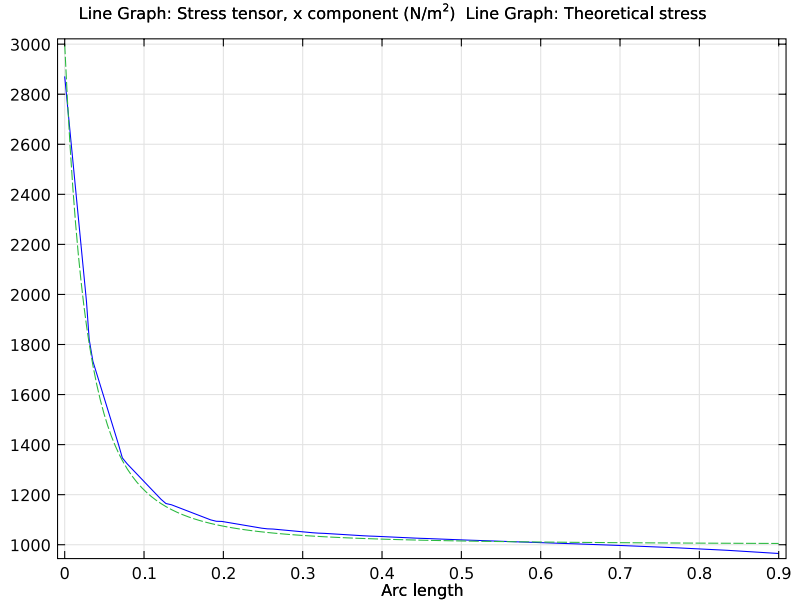

Figure 3: Normal stress, simulated results (solid line) versus the theoretical values (dashed line).

The theoretical values from Ref. 1 are in close agreement with the result from COMSOL Multiphysics.

Reference

1. D. Roylance, *Mechanics of Materials*, Wiley, 1996.

Model Library path: `Structural_Mechanics_Module/Verification_Models/kirsch_plate`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click the **2D** button.

3 Click **Next**.

4 In the **Add physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

5 Click **Next**.

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

7 Click **Finish**.

GEOMETRY 1

Square 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Square**.

2 Right-click **Square 1** and choose **Build Selected**.

Circle 1

1 Right-click **Geometry 1** and choose **Circle**.

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

3 In the **Radius** edit field, type `0.1`.

4 Click the **Build Selected** button.

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

6 Highlight the **Circle** and the **Square** in the **Graphics** window.

7 Click the **Difference** button on the main toolbar.

Form Union

In the **Model Builder** window, under **Model 1>Geometry 1** right-click **Form Union** and choose **Build Selected**.

SOLID MECHANICS

1 In the **Model Builder** window, under **Model 1** click **Solid Mechanics**.

2 In the **Solid Mechanics** settings window, locate the **2D Approximation** section.

3 From the **2D approximation** list, choose **Plane stress**.

4 Locate the **Thickness** section. In the d edit field, type `0.1`.

 ©2012 COMSOL

Linear Elastic Material 1

1 In the **Model Builder** window, expand the **Solid Mechanics** node, then click **Linear Elastic Material 1**.

2 In the **Linear Elastic Material** settings window, locate the **Linear Elastic Material** section.

3 From the E list, choose **User defined**. In the associated edit field, type `2.1e11`.

4 From the ν list, choose **User defined**. In the associated edit field, type `0.3`.

5 From the ρ list, choose **User defined**. In the associated edit field, type `7850`.

Symmetry 1

1 In the **Model Builder** window, right-click **Solid Mechanics** and choose the boundary condition **Symmetry**.

2 Select Boundaries 1 and 3 only.

Boundary Load 1

1 In the **Model Builder** window, right-click **Solid Mechanics** and choose the boundary condition **Boundary Load**.

2 Select Boundary 4 only.

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

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

1e3	X
0	Y

MESH 1

In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **Build All**.

STUDY 1

In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

Stress (solid)

The default plot shows the Von Mises stress combined with a scaled deformation of the plate. Use the distribution of the normal stress in the X-direction instead as the external load is only oriented in the x-direction.

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

2 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Solid Mechanics>Stress>Stress tensor (Spatial)>Stress tensor, x component (solid.sx)**.

3 Click the **Plot** button.

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

1D Plot Group 2

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

2 Right-click **1D Plot Group 2** and choose **Line Graph**.

3 Select Boundary 1 only.

4 In the **Line Graph** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Solid Mechanics>Stress>Stress tensor (Spatial)>Stress tensor, x component (solid.sx)**.

5 In the **Model Builder** window, right-click **1D Plot Group 2** and choose **Line Graph**.

6 Select Boundary 1 only.

7 In the **Line Graph** settings window, locate the **y-Axis Data** section.

8 In the **Expression** edit field, type `1000/2*(2+0.1^2/y^2+3*0.1^4/y^4)`.

9 Select the **Description** check box.

10 In the associated edit field, type `Theoretical stress`.

11 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

12 Click the **Plot** button.

 ©2012 COMSOL