Created in COMSOL Multiphysics 6.0

Kirsch Infinite Plate Problem

Introduction

In this example, you perform a static stress analysis to obtain the stress distribution in the vicinity of a small hole in an infinite plate. Two approximations of the infinite plate are evaluated. The first one uses a plate that is large compared to the hole while the second one employs an infinite element domain.

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, see Figure 1. Choose the size of the plate sufficiently large so that the stress concentration close to the hole is not affected.

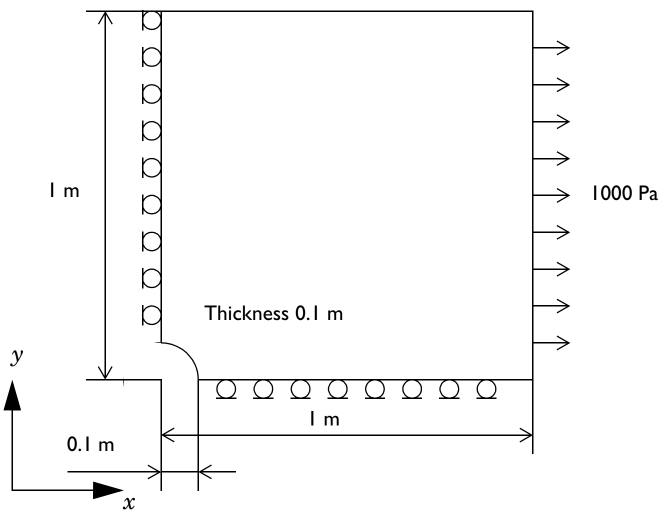

Figure 1: Geometry model of the Kirsch plate with rollers defining the symmetry plane.

When modeling a plate using the infinite element domain you need to create an additional layers around the plate. Those layers simulate the part that stretches to infinity and can have an arbitrarily length along the direction that stretches to infinity, for example 0.1 m.

In the model the infinite element domain is created along the y direction only since the numerical results along $x = 0$ symmetry plane are compared to an analytical reference and infinite element domain in x direction only have a minor influence.

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 and Discussion

The distribution of the normal stress in the x direction, σ_x, is shown in Figure 2and Figure 3. The stress contours of the finite model and the infinite model are very similar.

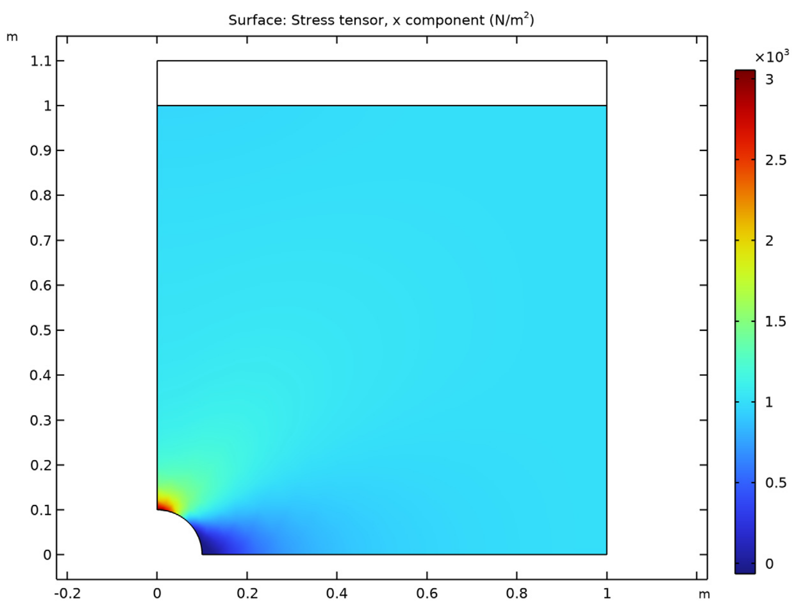

Figure 2: Distribution of the normal stress in the x direction for the finite model.

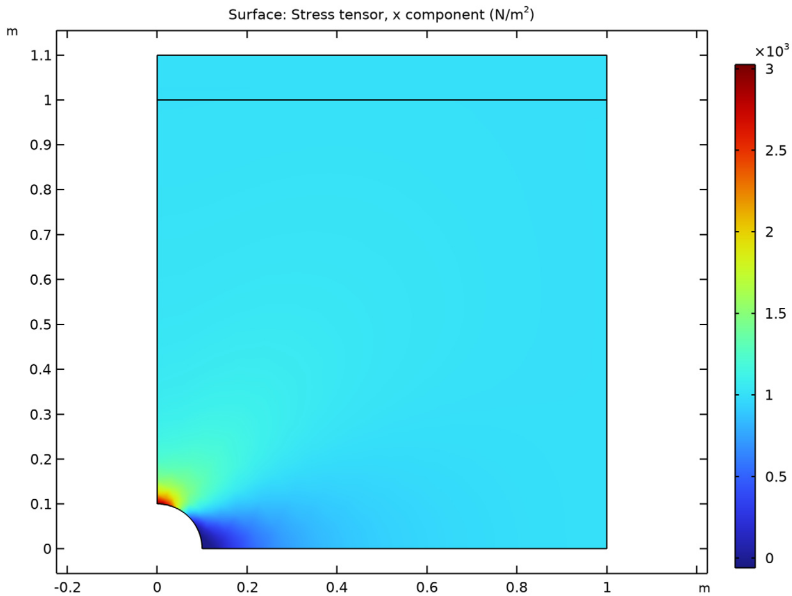

Figure 3: Distribution of the normal stress in the x direction for the infinite model.

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

$$\sigma_x = \frac{1000}{2}\left(2 + \frac{0.1^2}{y^2} + 3\frac{0.1^4}{y^4}\right) \tag{1}$$

Figure 4 shows the stress σ_x obtained from the solved models, and plotted as a function of the true y-coordinate along the left symmetry edge, which are in close agreement with the theoretical value according to Equation 1.

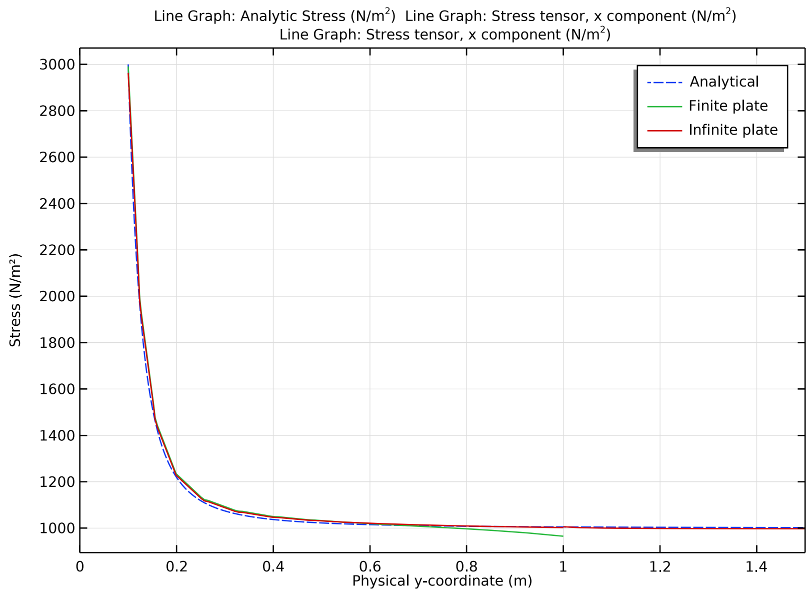

Figure 4: Normal stress, simulated results (solid line) versus the theoretical values (dashed line).

Away from the hole, stresses from the finite model starts drifting from the theoretical values, while stresses from the infinite model matches closely with the theoretical value.

The stress error is summarized in the following table:

TABLE 1: STRESS ERROR RELATIVE TO ANALYTICAL SOLUTION.

	FINITE PLATE	INFINITE PLATE
Near hole	1.1 %	0.2 %
Away from hole	-4.0 %	-0.1 %

Notes About the COMSOL Implementation

The default scaling function in **Infinite Element Domain** is rational. This type of function is well adapted to cases where the degrees of freedom vanish to zero at infinity. The present model is submitted to infinite loads at infinity, that means that constant strain and linear displacement are expected. For this type of infinite solution, polynomial functions are preferred. The relation between the stretched and geometric coordinates is

$$X_{\mathrm{m}} - X_0 = f\left(\frac{X - X_0}{\Delta X}\right)$$

where the function f is defined with an analytic function. Here we want f as a second-order polynomial: $f(\xi) = a\xi^2 + b\xi + c$. The continuity condition at X0, $f(0) = 0$, and at the end of the domain $f(1) = p_{\mathrm{w}}$ imply that the polynomial is:

$$f(\xi) = (p_{\mathrm{w}} - b)\xi^2 + b\xi$$

The infinite element domain gives best results when meshed with rectangular elements; see Figure 5.

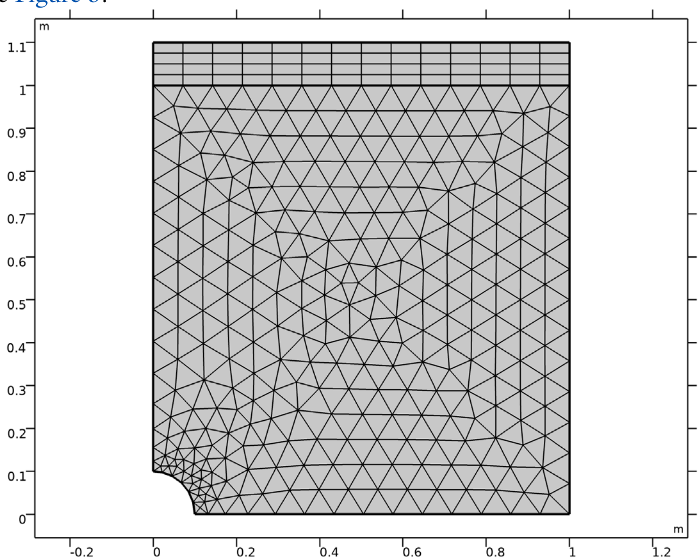

Figure 5: Infinite element domain modeled with rectangular elements.

Reference

1. D. Roylance, *Mechanics of Materials*, John Wiley & Sons, 1996.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/kirsch_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 **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
`pw`	`10[m]`	10 m	`Physical width of infinite element domain`
`deltaY`	`0.1[m]`	0.1 m	`Geometric thickness of infinite element layer`

Draw a rectangle with a top layer that represents the infinite element domain.

GEOMETRY 1

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 **Height** text field, type `1+deltaY`.

4 Click to expand the **Layers** section. Clear the **Layers on bottom** check box.

5 Select the **Layers on top** check box.

6 In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	deltaY

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 `0.1`.

4 Click **Build Selected**.

Difference 1 (dif1)

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

2 Add the rectangle and remove the circle in the **Difference** section.

3 In the **Settings** window for **Difference**, click **Build All Objects**.

GLOBAL DEFINITIONS

First add an analytical function for stress, based on Kirsch's theoretical solution of an infinite plate.

Analytic Stress

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

2 In the **Settings** window for **Analytic**, type `Analytic Stress` in the **Label** text field.

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

4 Locate the **Definition** section. In the **Expression** text field, type `1000/2*(2+(0.1/y)^2+3*(0.1/y)^4)`.

5 In the **Arguments** text field, type `y`.

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

Argument	Unit
y	m

7 In the **Function** text field, type `N/m^2`.

Create an analytic polynomial function to define the scaling in the infinite element domain.

DEFINITIONS

Analytic 2 (an2)

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

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

3 In the **Expression** text field, type `(pw-10*deltaY)*x^2+10*deltaY*x`.

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

Argument	Unit
x	m

5 In the **Function** text field, type `m`.

Infinite Element Domain 1 (ie1)

1 In the **Definitions** toolbar, click **Infinite Element Domain**.

2 Select Domain 2 only.

3 In the **Settings** window for **Infinite Element Domain**, locate the **Scaling** section.

4 From the **Coordinate stretching type** list, choose **User defined**.

5 From the **Stretching function** list, choose **Analytic 2 (an2)**.

Add a variable representing the physical y-coordinate to be used in postprocessing.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
ym	if(dom==2,ie1.Ym,y)	m	Physical y-coordinate

SOLID MECHANICS (SOLID)

First set up a model without the **Infinite Element Domain**.

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

2 In the **Settings** window for **Solid Mechanics**, locate the **Domain Selection** section.

3 In the list, select **2 (infinite elements)**.

4 Click **Remove from Selection**.

5 Select Domain 1 only.

6 Locate the **2D Approximation** section. From the list, choose **Plane stress**.

7 Locate the **Thickness** section. In the d text field, type `0.1`.

Symmetry 1

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

2 Select Boundaries 1, 2, and 5 only.

Boundary Load 1

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

2 Select Boundaries 6 and 7 only.

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

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

1e3	x
0	y

Now set up a model with the infinite element domain.

ADD PHYSICS

1 In the **Physics** 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 1** in the window toolbar.

5 In the **Physics** 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 `0.1`.

Symmetry 1

1 Right-click **Component 1 (comp1)>Solid Mechanics 2 (solid2)** and choose **More Constraints>Symmetry**.

2 Select Boundaries 1, 2, and 5 only.

Boundary Load 1

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

2 Select Boundaries 6 and 7 only.

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

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

1e3	x
0	y

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	2.1e11	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's ratio
Density	rho	7800	kg/m³	Basic

MESH 1

For the finite plate selection, a customized free triangular mesh must be used for getting a better solution in the stress concentration region. A smaller element size is set at the expected location of stress concentration.

Free Triangular 1

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

2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.

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

4 Select Domain 1 only.

Size 1

1 Right-click **Free Triangular 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 **Point**.

4 Select Point 1 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 In the associated text field, type `0.02`.

Infinite elements give better results when meshed with rectangular elements.

Mapped 1

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

Distribution 1

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

2 Select Boundary 2 only.

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

4 In the **Number of elements** text field, type `4`.

5 Click **Build All**.

STUDY 1

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

RESULTS

To check the error in the computed results, make a point evaluation of stresses near the hole ($y = 0.1$) and away from the hole ($y = 1$) for the solution computed with and without the infinite element domain. The error can be determined by finding the difference between computed stresses and analytical stresses.

Error Evaluation

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

2 In the **Settings** window for **Point Evaluation**, type `Error Evaluation` in the **Label** text field.

3 Select Points 1 and 2 only.

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

Expression	Unit	Description
`(solid.sx-AnaStress(y))/AnaStress(y)`		`Error in finite plate`
`(solid2.sx-AnaStress(y))/AnaStress(y)`		`Error in infinite plate`

5 Click **Evaluate**.

Stress (solid)

The default plots show the von Mises stress combined with a scaled deformation of the plate. Remove deformation and display the stress field in the x direction instead since the external load is oriented in that direction.

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/m²>solid.sx - Stress tensor, x component**.

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

Deformation

1 In the **Model Builder** window, expand the **Surface 1** node.

2 Right-click **Results>Stress (solid)>Surface 1>Deformation** and choose **Delete**.

Surface 1

1 In the **Model Builder** window, expand the **Results>Stress (solid2)** 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 2>Stress>Stress tensor (spatial frame) - N/m²>solid2.sx - Stress tensor, x component**.

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

Deformation

1 In the **Model Builder** window, expand the **Surface 1** node.

2 Right-click **Results>Stress (solid2)>Surface 1>Deformation** and choose **Delete**.

Stress Profile

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Stress Profile` in the **Label** text field.

Line Graph 1

1 Right-click **Stress Profile** and choose **Line Graph**.

2 Select Boundaries 1 and 2 only.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `AnaStress(ym)`.

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

6 In the **Expression** text field, type `ym`.

7 In the **Stress Profile** toolbar, click **Plot**.

8 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

9 Select the **Show legends** check box.

10 In the table, enter the following settings:

Legends
Analytical

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

12 In the **Stress Profile** toolbar, click **Plot**.

Line Graph 2

1 In the **Model Builder** window, right-click **Stress Profile** and choose **Line Graph**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)> Solid Mechanics>Stress>Stress tensor (spatial frame) - N/m²>solid.sx - Stress tensor, x component**.

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

5 In the **Expression** text field, type `y`.

6 Locate the **Legends** section. Select the **Show legends** check box.

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

8 In the table, enter the following settings:

Legends
Finite plate

Line Graph 3

1 Right-click **Stress Profile** and choose **Line Graph**.

2 Select Boundaries 1 and 2 only.

3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)> Solid Mechanics 2>Stress>Stress tensor (spatial frame) - N/m²>solid2.sx - Stress tensor, x component**.

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

5 In the **Expression** text field, type `ym`.

6 Locate the **Legends** section. Select the **Show legends** check box.

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

8 In the table, enter the following settings:

Legends
`Infinite plate`

Stress Profile

1 In the **Model Builder** window, click **Stress Profile**.

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

3 Select the **Manual axis limits** check box.

4 In the **x minimum** text field, type `0`.

5 In the **x maximum** text field, type `1.5`.

6 Locate the **Plot Settings** section. Select the **x-axis label** check box.

7 In the associated text field, type `Physical y-coordinate (m)`.

8 Select the **y-axis label** check box.

9 In the associated text field, type `Stress (N/m^2)`.

10 In the **Stress Profile** toolbar, click **Plot**.