Created in COMSOL Multiphysics 6.3

Surface Cracked Cylinder

Introduction

This model reproduces a NAFEMS benchmark (Ref. 1) in which a pressurized cylinder has a horizontal semi-elliptical crack on its inner surface. The energy release rate is calculated along the crack front using the J-integral method. The stress intensity factors calculated from the J-integral are then compared to the reference data.

Model Definition

Since the model consists on a cylinder with a horizontal crack on its mid plane, only a quarter of the whole geometry is built. A tube around the elliptical crack front is built to generate a swept mesh along it. The swept mesh makes the J-integral curve along the crack front smoother.

A **Symmetry** condition is applied on the thickness boundaries of the cylinder. On the crack boundaries, it is overridden by a **Crack** node. The crack is symmetric to define explicitly the crack front. Due to the symmetry, the stress intensity factors for modes II and III will be zero.

The pressure load can be applied with a **Boundary Load** on the inner face of the cylinder, and with a **Face Load** subnode on the crack face.

The J-integral at a point of the crack front is composed of a closed contour integral and a surface integral. The closed contour integral is composed of a circular integral around the crack front and an integral on the crack face to take into account the force applied on it:

$$J = \int_{\Gamma} W_{\mathrm{s}}\mathbf{m}\cdot\mathbf{e}_1 - (\sigma\cdot\mathbf{m})\cdot(\nabla\mathbf{u}\cdot\mathbf{e}_1)\mathrm{d}l + \int_{\Gamma\mathrm{face}} \mathbf{F}_{\mathrm{A}}\cdot(\nabla\mathbf{u}\cdot\mathbf{e}_1)\mathrm{d}l$$

$$+ \int_{\mathrm{A}} \nabla[(\sigma\cdot\mathbf{t}_1)\cdot(\nabla\mathbf{u}\cdot\mathbf{e}_1)]\mathrm{d}A$$

The stress intensity factor in mode I is then computed from the J-integral

$$K_{\mathrm{I}} = \sqrt{\frac{E^*}{1+\beta_K^2}J}$$

where E^* is the equivalent Young's modulus. A 2D plane strain condition is assumed, so

$$E^* = \frac{E}{1-\nu^2}\,.$$

Results and Discussion

Figure 1 shows the default stress plot, where a stress concentration around the crack front is visible. Figure 2 shows a closer view of this stress concentration

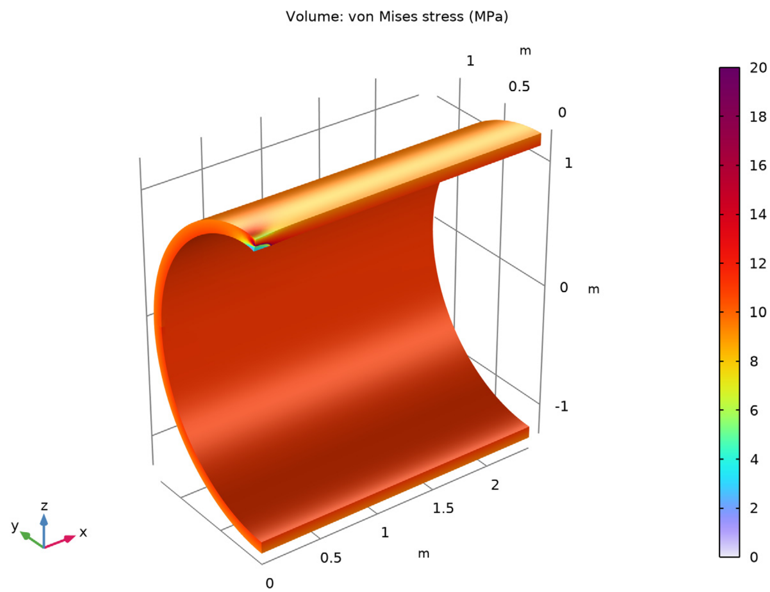


Figure 1: Von Mises stress in the cylinder and at the crack front.

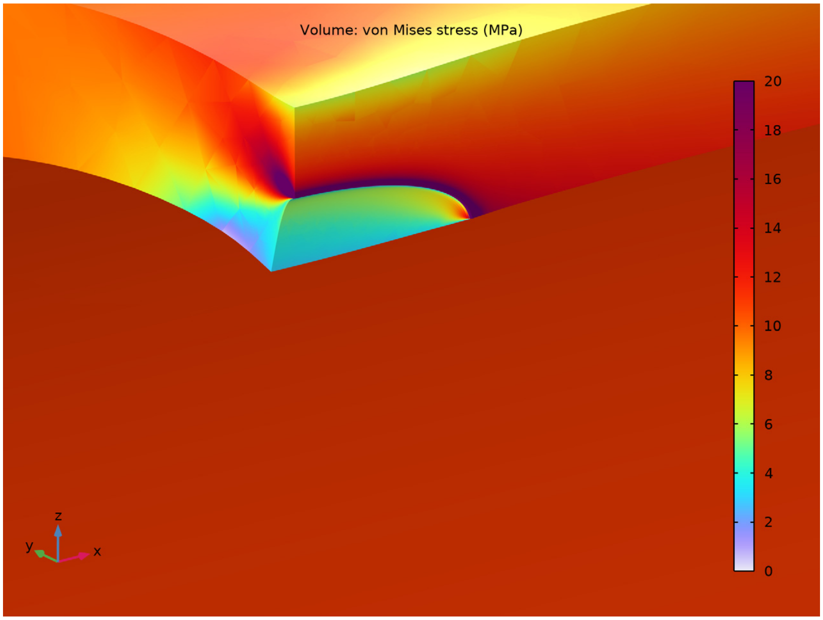


Figure 2: Close view of the stress concentration at the crack front.

The J-integral calculated along the crack front represents the energy release rate per surface area when the crack is locally extended in the crack plane direction. The crack extension direction, the J-integral intensity and the surface contour used for integral calculation are plotted in Figure 3

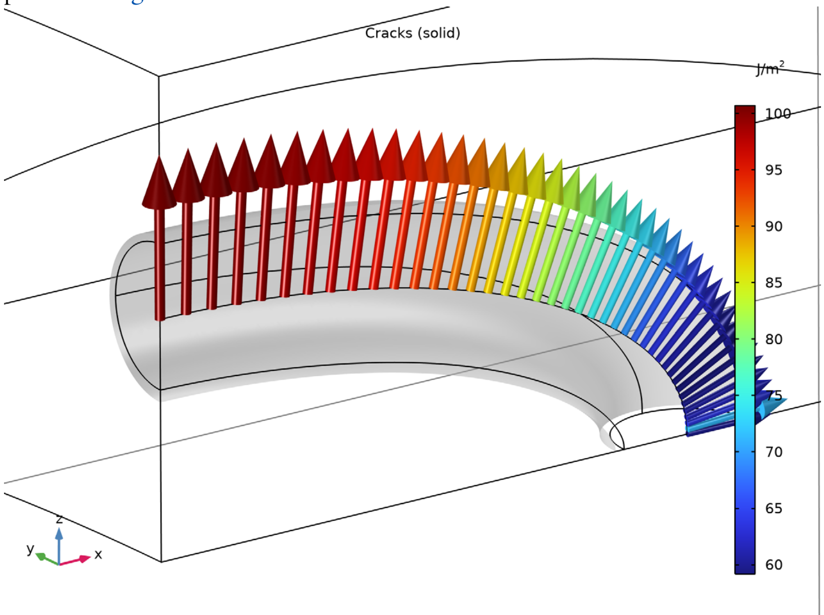


Figure 3: 3D plot of J-integral: crack growth direction, intensity, and contour used for the calculation.

Figure 4 plots the J-integral as function of the parametric angle along the elliptical crack. A jump in the curve at zero angle can be noticed. This glitch is often seen near free boundaries due to the singularities that contribute to the surface integral.

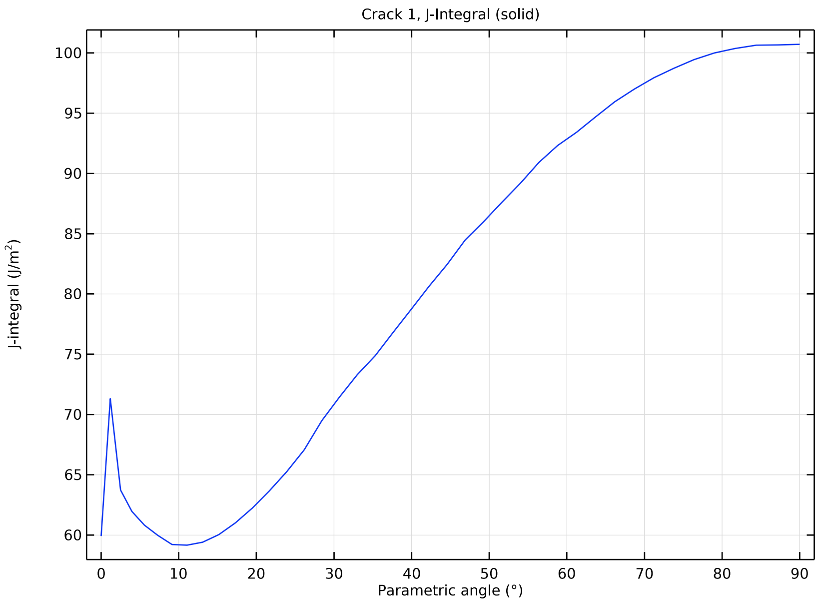


Figure 4: J-integral along the crack front.

From the J-integral the stress intensity factor in mode I can be calculated and compared to the benchmark in Ref. 1, see Figure 5.

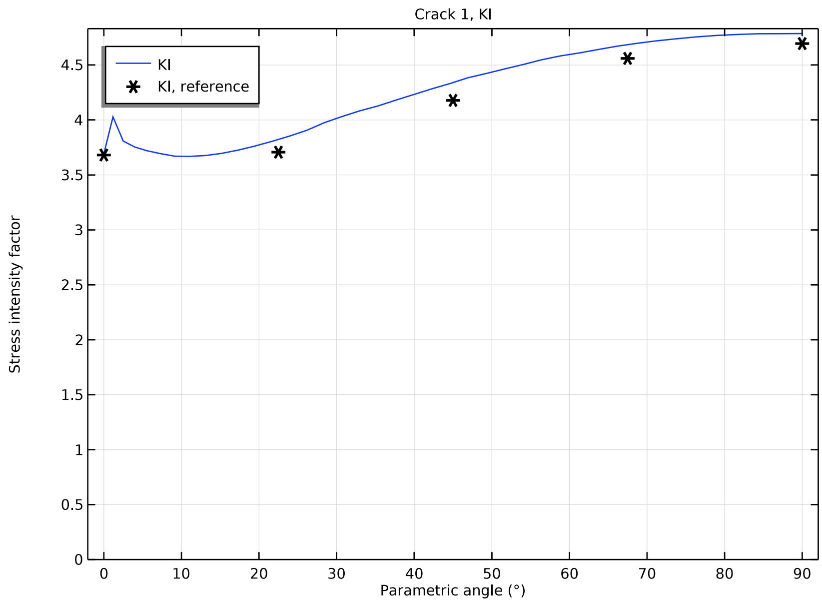


Figure 5: Stress intensity factor along the crack front compared to the benchmark.

Reference

1. R. Judge and B. Mardsen, *Three Dimensional Test Cases in Linear Elastic Fracture Mechanics, part 4: Surface Cracked Cylinder*, NAFEMS, 1993.

Application Library path: `Structural_Mechanics_Module/Fracture_Mechanics/surface_cracked_cylinder`

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**.

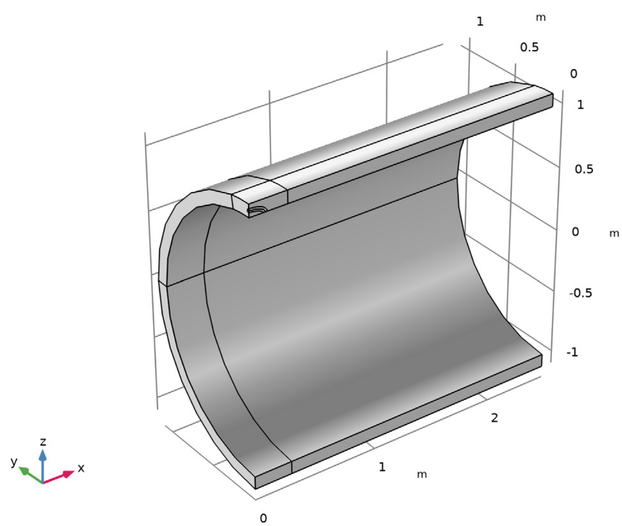
GEOMETRY 1

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `surface_cracked_cylinder_geom_sequence.mph`.

3 In the **Geometry** toolbar, click **Build All**.

4 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.



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
p	1[MPa]	1E6 Pa	Pressure load

MATERIALS

Material 1 (mat1)

1 In the **Materials** toolbar, click **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	207[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	8000	kg/m³	Basic

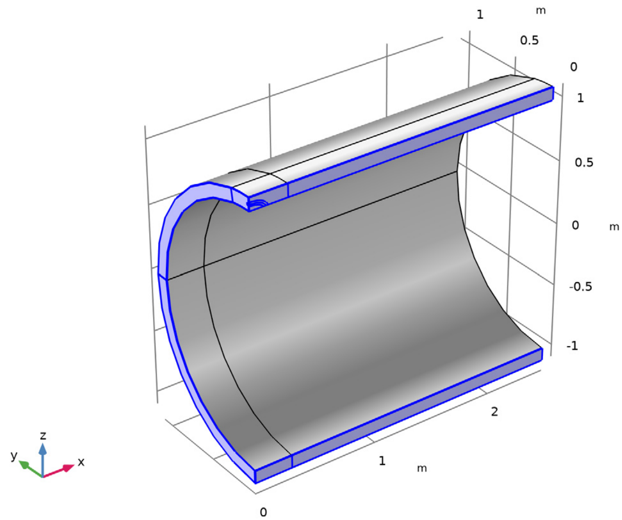
SOLID MECHANICS (SOLID)

Symmetry 1

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

2 Select Boundaries 1, 2, 5, 6, 8, 9, 11, 12, 15, 22, and 26 only.

The **Symmetry** condition is applied on the whole thickness of the cylinder. It will be overridden by the **Crack** feature on the crack surface.



Prescribed Displacement 1

1 In the **Physics** toolbar, click **Points** and choose **Prescribed Displacement**.

2 Select Point 11 only.

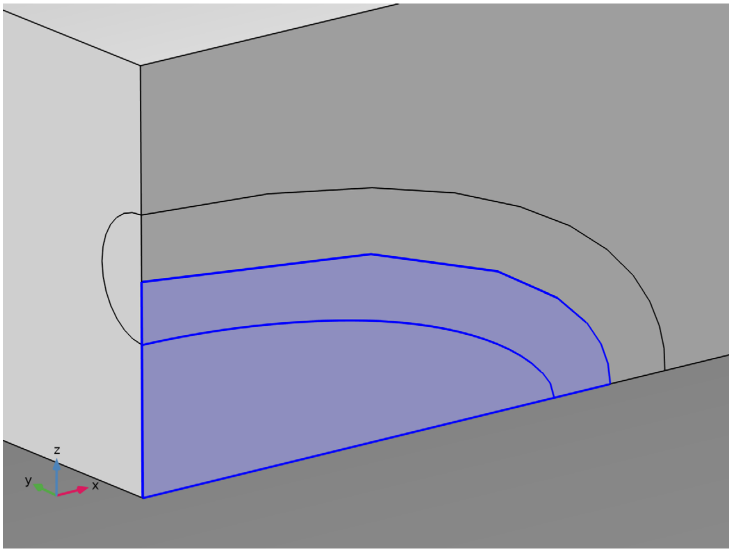
3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 From the **Displacement in z direction** list, choose **Prescribed**.

Crack 1

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

2 Select Boundaries 6 and 9 only.



3 In the **Settings** window for **Crack**, locate the **Crack Definition** section.

4 From the **Crack surface** list, choose **Symmetric**.

5 Click to expand the **Crack Front** section. Click **Clear Selection**.

6 Select Edge 13 only.

J-Integral 1

In the **Physics** toolbar, click **Attributes** and choose **J-Integral**.

Boundary Load 1

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

Apply a pressure load on the inner face of the cylinder. The pressure on the crack will be applied under the **Crack** feature.

2 Select Boundaries 4, 7, 17, 24, 27, and 31 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 `p`.

Crack 1

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

Face Load 1

1 In the **Physics** toolbar, click **Attributes** and choose **Face Load**.

2 In the **Settings** window for **Face Load**, locate the **Load** section.

3 In the *p* text field, type `p`.

MESH 1

Free Triangular 1

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

Size

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

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `th/2`.

5 In the **Minimum element size** text field, type `a/200`.

Free Triangular 1

1 In the **Model Builder** window, click **Free Triangular 1**.

2 Select Boundary 8 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 5 only.

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

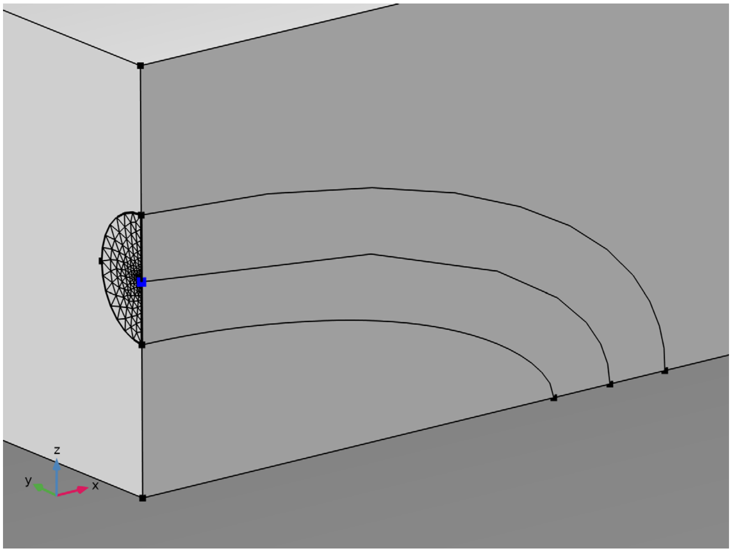
6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** checkbox. In the associated text field, type `a/100`.

8 Select the **Minimum element size** checkbox. In the associated text field, type `a/200`.

9 Select the **Maximum element growth rate** checkbox. In the associated text field, type `1.2`.

10 Click **Build Selected**.



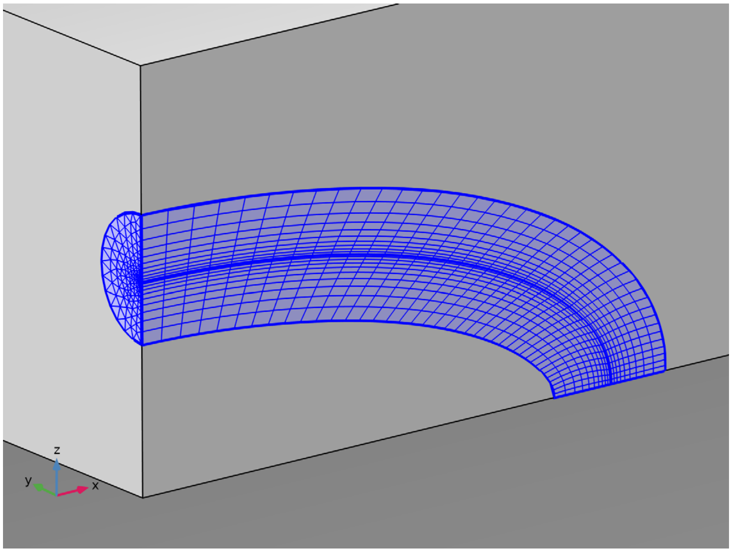
Swept 1

1 In the **Mesh** toolbar, click **Swept**.
2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Select Domain 3 only.
5 Click to expand the **Sweep Method** section. From the **Destination mesh generation** list, choose **Morph source onto destination**.

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.
2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
3 From the **Distribution type** list, choose **Predefined**.
4 In the **Number of elements** text field, type 40.
5 In the **Element ratio** text field, type 6.
6 Select the **Reverse direction** checkbox.

7 Click **Build Selected**.



Free Tetrahedral 1

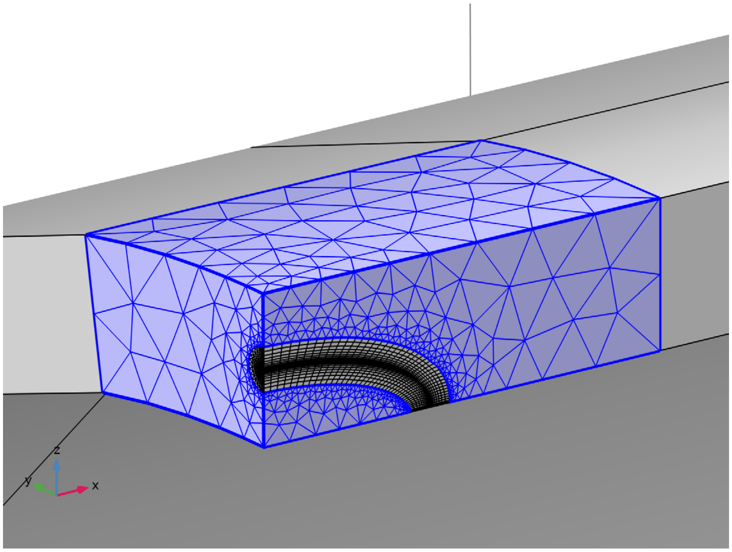
1 In the **Mesh** toolbar, click **Free Tetrahedral**.

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

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

4 Select Domain 2 only.

5 Click **Build Selected**.



Swept 2

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

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

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

4 Select Domains 1 and 4 only.

Swept 3

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

Distribution 1

1 Right-click **Swept 3** and choose **Distribution**.

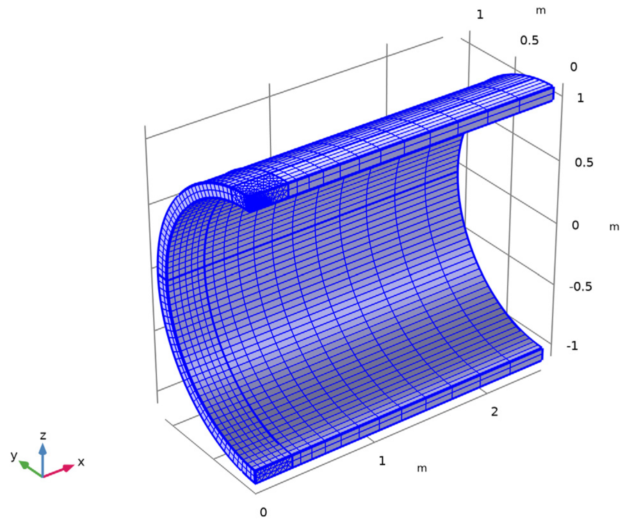
2 In the **Settings** window for **Distribution**, locate the **Distribution** section.

3 From the **Distribution type** list, choose **Predefined**.

4 In the **Number of elements** text field, type 12.

5 In the **Element ratio** text field, type 4.

6 Click **Build All**.



STUDY 1

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

RESULTS

Volume 1

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

2 In the **Settings** window for **Volume**, locate the **Expression** section.

3 From the **Unit** list, choose **MPa**.

4 Click to expand the **Range** section. Select the **Manual color range** checkbox.

5 In the **Maximum** text field, type 20.

Stress (solid)

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

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 Clear the **Plot dataset edges** checkbox.

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

The default plot shows von Mises stress. At a closer view the maximum stress is visible along the crack front, see Figure 2.

From **Result Templates**, you can also add a plot that shows the crack direction and the intensity of the J-integral, Figure 3. The contour used to calculate the J-integral is displayed as well.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Solid Mechanics** > **Cracks (solid)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Cracks (solid)

Add a new plot group from **Result Templates** to show the J-integral along crack front.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Solid Mechanics** > **Crack 1, J-Integral (solid)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

To reproduce Figure 4 you may introduce the parametric angle along the ellipse.

DEFINITIONS

Variables 1

1 In the **Definitions** toolbar, click a= **Local Variables**.

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

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

4 Select Domains 2 and 3 only.

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

Name	Expression	Unit	Description
angle	atan2((Z-R1)/a,X/c)	rad	Parametric angle

Study 1

In the **Study** toolbar, click **Update Solution**.

Results

J-Integral 1

1 In the **Model Builder** window, expand the **Crack 1, J-Integral (solid)** node, then click **J-Integral 1**.

2 In the **Settings** window for **Line Graph**, locate the **x-Axis Data** section.

3 From the **Parameter** list, choose **Expression**.

4 In the **Expression** text field, type `angle`.

5 From the **Unit** list, choose **°**.

6 In the **Crack 1, J-Integral (solid)** toolbar, click **Plot**.

Duplicate the last plot group to plot and compare the stress intensity factor to the reference value, Figure 5

Crack 1, KI

1 In the **Model Builder** window, right-click **Crack 1, J-Integral (solid)** and choose **Duplicate**.

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

KI

1 In the **Model Builder** window, expand the **Crack 1, KI** node, then click **J-Integral 1**.

2 In the **Settings** window for **Line Graph**, type `KI` in the **Label** text field.

3 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Cracks** > **solid.crack1.jint1.KI - Stress intensity factor, mode I**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `solid.crack1.jint1.KI/1e6`.

Crack 1, KI

1 In the **Model Builder** window, click **Crack 1, KI**.

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

3 Select the **x-axis label** checkbox.

4 Select the **y-axis label** checkbox. In the associated text field, type `Stress intensity factor`.

5 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

6 In the **y minimum** text field, type `0`.

7 In the **Crack 1, KI** toolbar, click **Plot**.

Import the stress intensity factors from the benchmark to compare with the computed results.

Table 1

1 In the **Results** toolbar, click **Table**.

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

3 Click **Import**.

4 Browse to the model's Application Libraries folder and double-click the file `surface_cracked_cylinder_results.txt`.

Table Graph 1

1 Right-click **Crack 1, KI** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, locate the **Coloring and Style** section.

3 From the **Color** list, choose **From theme**.

4 Find the **Line style** subsection. From the **Line** list, choose **None**.

5 Find the **Line markers** subsection. From the **Marker** list, choose **Asterisk**.

6 In the **Crack 1, KI** toolbar, click **Plot**.

Crack 1, KI

1 In the **Model Builder** window, click **Crack 1, KI**.

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

3 Select the **Show legends** checkbox.

4 From the **Position** list, choose **Upper left**.

KI

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

2 In the **Settings** window for **Line Graph**, click to expand the **Legends** section.

3 Select the **Show legends** checkbox.

4 Find the **Include** subsection. Clear the **Solution** checkbox.

5 Select the **Label** checkbox.

Table Graph 1

1 In the **Model Builder** window, click **Table Graph 1**.

2 In the **Settings** window for **Table Graph**, click to expand the **Legends** section.

3 Select the **Show legends** checkbox.

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

5 In the table, enter the following settings:

Legends
KI, reference

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Solid Mechanics** > **Fracture Mechanics Results (solid)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.