Created in COMSOL Multiphysics 6.2

Single Edge Crack

Introduction

This example deals with the stability of a plate with an edge crack that is subjected to a tensile load. To analyze the stability of existing cracks, you can apply the principles of fracture mechanics.

A common parameter in fracture mechanics is the so-called stress intensity factor K_I, which provides a means to predict if a specific crack may lead to a fracture. When K_I becomes equal to the critical fracture toughness of the material, K_{Ic} (a material property), then usually catastrophic fracture occurs.

Determining the stress intensity factor directly from the local state at the crack tip is problematic, since the stresses are singular there. Therefore, more indirect energy-based methods are attractive. In this example, K_I is computed using the J-integral and from the energy release rate.

In addition, the crack growth rate and number of cycles needed to propagate the crack a certain distance are computed.

Model Definition

A plate with a width $w = 1.5$ m and height $h = 4.5$ m has a single horizontal edge-crack at the middle of the left vertical edge. The length of the crack is varied from $a = 0.5$ m to $a = 0.7$ m, see Figure 1. An external load is pulling the plate such that the top and bottom edges experience tensile stress, σ, of 20 MPa.

Because of the symmetry, only half of the plate is modeled. There are three paths for computing the J-integral:

1. A circle around the crack tip with the radius being half the crack length.
2. A circle around the crack tip with the radius being 0.7 times the crack length.
3. The external boundaries, excluding the crack surface.

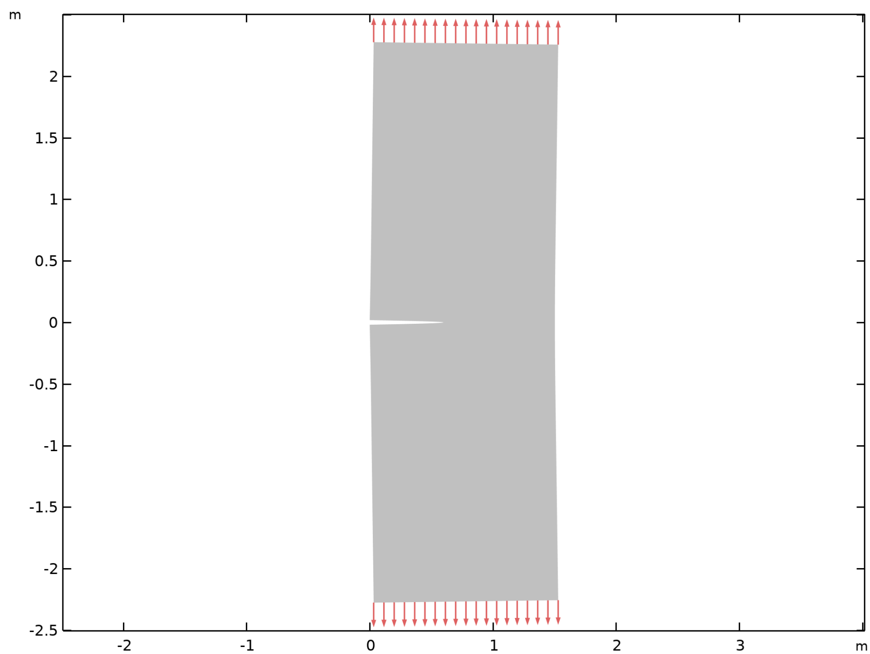

Figure 1: Plate geometry and loading (before applying symmetry conditions).

You apply a tensile load to the upper horizontal edge, while the lower horizontal edge is constrained in the y direction using a symmetry condition. One point is constrained in the horizontal direction in order to suppress rigid body motion.

MATERIAL MODEL

The material properties are representative for steel.

TABLE 1: MATERIAL DATA.

Quantity	Name	Expression
Young's modulus	E	206 GPa
Poisson's ratio	ν	0.3
Coefficient in Paris' law	A	$1.4 \cdot 10^{-11}$ (K_I unit system: $MN/m^{3/2}$)
Exponent in Paris' law	m	3.1

THE J-INTEGRAL

In this model, you determine the stress intensity factor K_I using the so-called J-integral.

The J-integral is a two-dimensional path independent line integral along a counterclockwise contour, Γ, surrounding the crack tip. For a crack extending along the positive *x*-axis, the J-integral is defined as

$$J = \int_{\Gamma} W dy - T_i \frac{\partial u_i}{\partial x} ds = \int_{\Gamma} \left(W n_x - T_i \frac{\partial u_i}{\partial x} \right) ds$$

where W is the strain energy density

$$W = \frac{1}{2}(\sigma_x \cdot \varepsilon_x + \sigma_y \cdot \varepsilon_y + \sigma_{xy} \cdot 2 \cdot \varepsilon_{xy})$$

and $\mathbf{T}$ is the traction vector defined as

$$\mathbf{T} = \begin{bmatrix} \sigma_x \cdot n_x + \sigma_{xy} \cdot n_y \\ \sigma_{xy} \cdot n_x + \sigma_y \cdot n_y \end{bmatrix}$$

σ_{ij} denotes the stress components, ε_{ij} the strain components, and n_i the normal vector components.

The J-integral has the following relation to the Mode I stress intensity factor for a plane stress case and a linear elastic material:

$$J = \frac{K_I^2}{E} \tag{1}$$

where E is Young's modulus.

ENERGY RELEASE RATE

For a linear elastic material it is possible to compute the value of the J-integral without using the path integrals. The reason is that its value equals the value of the energy release rate, G,

$$G = -\frac{1}{t}\frac{\partial U}{\partial a} \tag{2}$$

Here, U is the potential energy, a is the crack length, and t is the thickness. The potential energy of an elastic body is

$$U = \frac{1}{2}\int_{\Omega} \sigma:\varepsilon \, dV - \frac{1}{2}\int_{\Omega} (\mathbf{F_v} \cdot \mathbf{u} \, dV) - \int_{\partial\Omega} \mathbf{T} \cdot \mathbf{u} \, dS$$

The first term is the strain energy in the volume, and the second and third terms are the potential of the body load and prescribed tractions, respectively.

CRACK PROPAGATION

When subjected to a periodic load, the crack growth rate (in meters per load cycle) can be expressed by Paris' law:

$$\frac{da}{dN} = A(\Delta K_I)^m \tag{3}$$

Here, A and m are material parameters and ΔK_I is the range of the stress intensity factor. It is assumed that the load varies between 0 and 20 MPa, so that ΔK_I equals the computed K_I. Note that the value of the coefficient A depends on the unit for the stress intensity factor in a rather complex way, because of the power m, which in general is noninteger.

Results

Figure 2 shows the stress singularity at the crack tip.

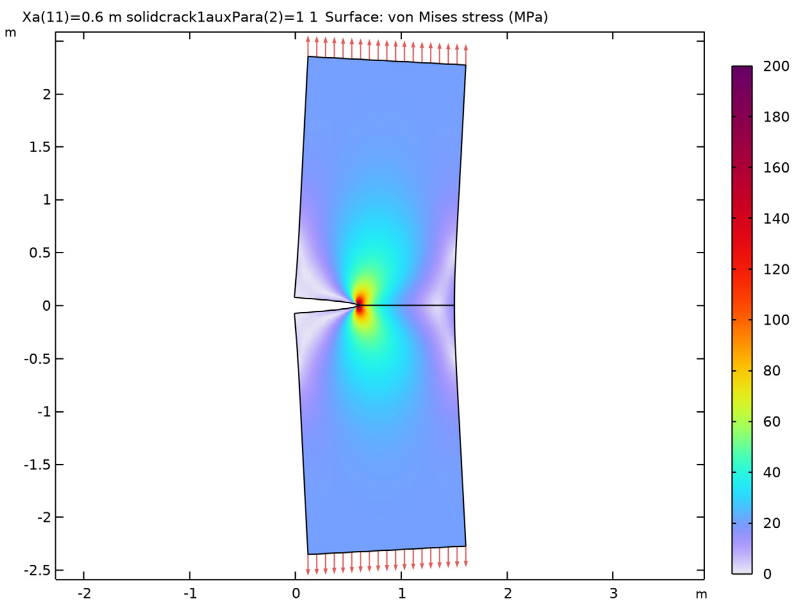

Figure 2: von Mises stress and the deformed shape of the plate when the crack length is 0.6 m. The displacement is exaggerated to illustrate the deformation under the applied load.

Based on Ref. 1 an analytical solution for the stress intensity factor is

$$K_{\mathrm{Ia}} = \sigma \cdot \sqrt{\pi \cdot a} \cdot f\left(\frac{a}{w}\right)$$

where $\sigma = 20$ MPa, and f is a correction factor given in Ref. 1.

$$f\left(\frac{a}{w}\right) = \frac{\sqrt{\frac{2w}{\pi a}\tan\frac{\pi a}{2w}}\left(0.752 + 2.02\frac{a}{w} + 0.37\left(1 - \sin\frac{\pi a}{2w}\right)^3\right)}{\cos\frac{\pi a}{2w}} \tag{4}$$

The expression in Equation 4 assumes that the plate is long, so that the height is significantly larger than the width. The stress field in Figure 2 indicates that this assumption is fulfilled.

The calculated stress intensity factors for the three different contours, tabulated in the evaluation group **Cracks (solid)**, show excellent agreement with the reference value. The largest deviation for any of the studied crack lengths and integration paths is less than 0.3%.

When using the built-in J-integral computation, you should ascertain that none of the contours used for the integration passes outside the computational domain, or encloses another crack. In the default plot **Cracks**, all integration paths are visualized. In Figure 3 and Figure 4 the contours are shown for the shortest and longest cracks, respectively.

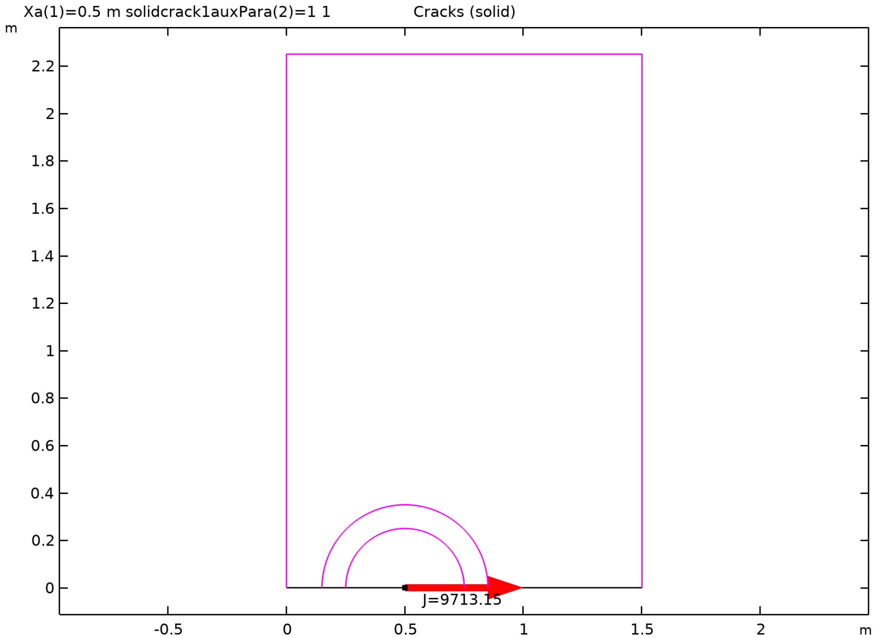

Figure 3: The contours used for integration when a = 0.5 m.

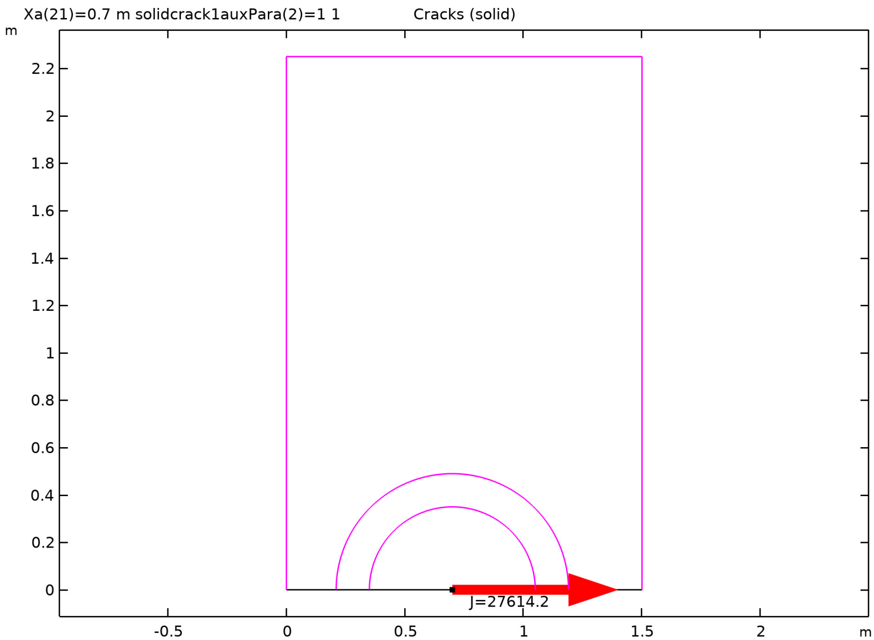

Figure 4: The contours used for integration when a = 0.7 m.

The two different ways of computing the energy release rate, and thus K_I, are compared in Figure 5. As can be seen, these two methods give essentially the same values.

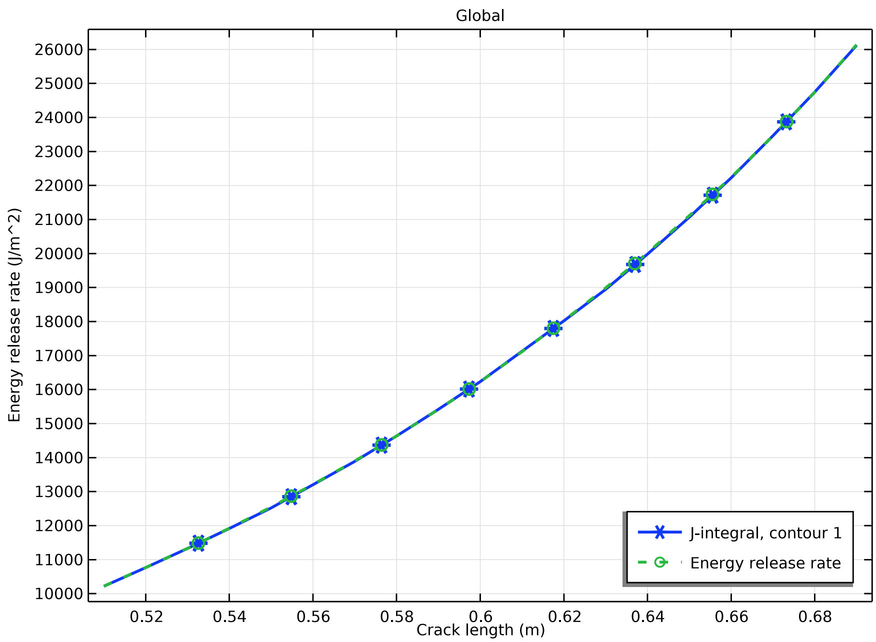

Figure 5: J-integral compared with energy release rates computed using numerical differentiation.

Finally, the crack growth speed can be investigated. In Figure 6, the crack growth speed is shown as function of the crack length. The dependence is quite strong: an increase in crack length from 0.5 m to 0.7 m (40%), increases the crack growth rate by a factor of 5. According to the constants used in Paris' law, the crack growth rate is proportional to the stress intensity factor raised to the power of 3.1. As can be seen from the previous results, the stress intensity factor increases strongly with the crack length, and this combination results in an increasing crack growth rate.

In practice, Paris' law may not be applicable when K_I approaches the critical value K_{Ic}.

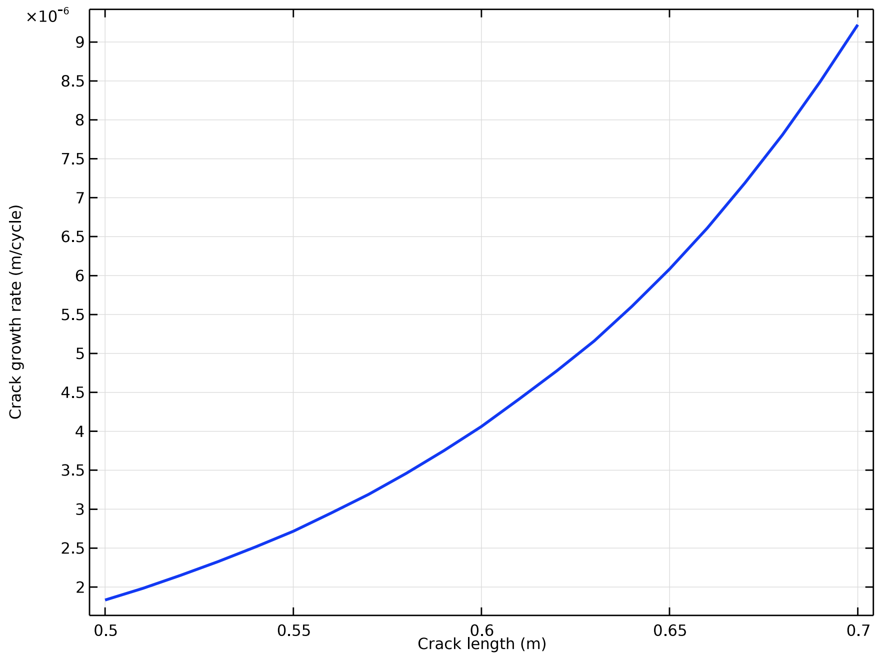

Figure 6: Crack propagation rate as function of the crack length.

Notes about the COMSOL Implementation

In this analysis you compute the J-integral for three different contours traversing three different regions around the crack tip. Note that the boundaries along the crack are not included in the J-integral because they do not give any contribution to the J-integral. This is due to the following facts: for an ideal crack, dy is zero along the crack faces, and all traction components are also zero ($T_i = 0$) as the crack faces are not loaded.

To evaluate the J-integral along three different paths, you need to add one **Crack** node with three **J-integral** subnodes. Since the **Symmetric** option is used in the **Crack** node, the values of the J-integrals and stress intensity factors automatically take the full structure into account. When you add a **J-integral** node, the K_{I}, K_{II}, and (in 3D) K_{III} stress intensity factors are also computed.

To evaluate the energy release rate, you need to add **Virtual Crack Extension** subnode under the **Crack** node. The subnode has an action button **Create Deformed Geometry and Study** to set up the deformed geometry and stationary study with sensitivity node. Since the

Symmetric option is used in the **Crack** node, the values of the energy release rate automatically take the full structure into account.

Reference

1. H Tada, P.C. Paris, and G.R. Irwin, *The Stress Analysis of Cracks Handbook*, Del Research Corp, Press, 1973.

Application Library path: `Structural_Mechanics_Module/Fracture_Mechanics/single_edge_crack`

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 **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 Click **Load from File**.

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

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)** 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
`dadN`	`AP*(solid.crack1.jint1.KI/1e6)^mP`		`Crack growth rate (m/cycle)`

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 **Width** text field, type `Wp`.

4 In the **Height** text field, type `Hp`.

5 Click **Build All Objects**.

Add a point at the crack tip.

Point 1 (pt1)

1 In the **Geometry** toolbar, click **Point**.

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

3 In the **x** text field, type `Xa`.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

MATERIALS

Steel

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Steel` in the **Label** text field.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	`206[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)

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

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

3 From the list, choose **Plane stress**.

4 Locate the **Thickness** section. In the d text field, type `Th`.

Symmetry 1

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

2 Select Boundaries 2 and 4 only.

It does not matter whether you select boundary 2 as a symmetry boundary or not. If selected, it will be overridden when the **Crack** node is added.

Boundary Load 1

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

2 Select Boundary 3 only.

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

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

`0`	x
`20[MPa]`	y

Prescribed Displacement 1

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

Suppress rigid body motion.

2 Select Point 4 only.

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

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

Crack 1

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

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

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

4 Select Boundary 2 only.

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

6 Select Point 3 only.

J-Integral 1

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

2 Right-click **J-Integral 1** and choose **Duplicate**.

J-Integral 2

1 In the **Model Builder** window, click **J-Integral 2**.

2 In the **Settings** window for **J-Integral**, locate the **J-Integral** section.

3 In the r_Γ text field, type `solid.crack1.crackSize*0.7`.

Crack 1

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

J-Integral 3

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

2 In the **Settings** window for **J-Integral**, locate the **J-Integral** section.

3 From the **Integration path** list, choose **On edges**.

4 Select Point 3 only.

5 Locate the **Integration Path** section. Click to select the **Activate Selection** toggle button.

6 Select Boundaries 1, 3, and 5 only.

To compute the energy release rate, add a **Virtual Crack Extension** feature. Use the action buttons in the feature to generate the deformed geometry and study.

Crack 1

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

Virtual Crack Extension 1

1 In the **Physics** toolbar, click **Attributes** and choose **Virtual Crack Extension**.

To create a parametric study, use options in the feature's **Advanced** section. To see this section, activate advanced physics settings as follows.

2 Click the **Show More Options** button in the **Model Builder** toolbar.

3 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.

4 Click **OK**.

5 In the **Settings** window for **Virtual Crack Extension**, click to expand the **Advanced** section.

6 From the **Add parametric sweep** list, choose **Yes**.

7 In the **Parameters** table, enter the following settings:

Index	Parameter name	Parameter value list	Parameter unit
1	Xa	range(0.5,da,0.7)	m

8 Click **Automated Model Setup** in the upper-right corner of the **Sensitivity** section. From the menu, choose **Create Deformed Geometry and Study**.

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Finer**.

4 Locate the **Sequence Type** section. From the list, choose **User-controlled mesh**.

Size 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size 1**.

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

3 Click the **Custom** button.

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

5 Select the **Maximum element size** check box. In the associated text field, type `0.01`.

6 Click **Build All**.

VIRTUAL CRACK EXTENSION STUDY

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

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

RESULTS

Mirror 2D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 2D**.

2 In the **Settings** window for **Mirror 2D**, locate the **Axis Data** section.

3 In row **Point 2**, set **X** to `1`.

4 In row **Point 2**, set **Y** to `0`.

5 Locate the **Data** section. From the **Dataset** list, choose **Virtual Crack Extension Study/ Solution 2 (solidcrack1solp)**.

Stress (solid)

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

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

3 From the **Dataset** list, choose **Mirror 2D 1**.

4 From the **Parameter value (Xa (m))** list, choose **0.6**.

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Surface 1

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

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

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

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

5 In the **Maximum** text field, type `200`.

Stress (solid)

Right-click **Results>Stress (solid)>Surface 1** and choose **Arrow Line**.

Boundary Load

1 In the **Settings** window for **Arrow Line**, type `Boundary Load` in the **Label** text field.

2 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Load>solid.F_Ax,solid.F_Ay - Load (spatial frame)**.

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

4 Locate the **Arrow Positioning** section. From the **Placement** list, choose **Mesh nodes**.

5 Locate the **Coloring and Style** section.

6 Select the **Scale factor** check box. In the associated text field, type `1e-8`.

7 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

8 Clear the **Color and data range** check box.

9 Clear the **Color** check box.

10 Clear the **Arrow scale factor** check box.

Color Expression 1

1 Right-click **Boundary Load** and choose **Color Expression**.

2 In the **Settings** window for **Color Expression**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Load>solid.F_A_Mag - Load magnitude - N/m²**.

3 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Gradient**.

4 From the **Top color** list, choose **Red**.

5 From the **Bottom color** list, choose **Gray**.

6 Clear the **Color legend** check box.

Deformation 1

Right-click **Boundary Load** and choose **Deformation**.

Line 1

1 In the **Stress (solid)** toolbar, click **Line**.

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

3 In the **Expression** text field, type 1.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

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

7 Click to expand the **Quality** section. From the **Smoothing** list, choose **None**.

Deformation 1

In the **Stress (solid)** toolbar, click **Deformation**.

Line 1

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

2 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

3 Clear the **Color** check box.

4 Clear the **Color and data range** check box.

Deformation

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

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** check box. In the associated text field, type 200.

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

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

Check the placement of integration contours. In particular, ensure that circular paths remain inside the domain.

ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Add Predefined Plot** to open the **Add Predefined Plot** window.

2 Go to the **Add Predefined Plot** window.

3 In the tree, select **Virtual Crack Extension Study/Solution 2 (solidcrack1solp)> Solid Mechanics>Cracks (solid)**.

4 Click **Add Plot** in the window toolbar.

RESULTS

Cracks (solid)

1 In the **Cracks (solid)** toolbar, click **Plot**.

2 In the **Settings** window for **2D Plot Group**, click **cycle_plot_level**.

3 Click **Plot First**.

Check if there are any differences between the results for the three contours by adding predefined evaluation group. That could indicate a too coarse mesh.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

2 In the tree, select **Virtual Crack Extension Study/Solution 2 (solidcrack1solp)> Solid Mechanics>Fracture Mechanics Results (solid)**.

3 Click **Add Plot** in the window toolbar.

4 In the **Home** toolbar, click **Add Predefined Plot** to close the **Add Predefined Plot** window.

Since this is a pure Mode I case, remove the evaluation of the Mode II stress intensity factors.

RESULTS

Stress Intensity Factors, Mode 2

1. In the **Model Builder** window, expand the **Results>Fracture Mechanics Results (solid)** node.
2. Right-click **Stress Intensity Factors, Mode 2** and choose **Delete**.

Stress Intensity Factors, Mode 1

Add a comparison between the values from the second integration path and the reference stress intensity factor.

1. In the **Model Builder** window, under **Results>Fracture Mechanics Results (solid)** click **Stress Intensity Factors, Mode 1**.
2. In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.
3. In the table, enter the following settings:

Expression	Unit	Description
`K1r`	`1`	`Reference stress intensity factor`
`(solid.crack1.jint2.KI-K1r)/K1r*100`	`1`	`Percent difference from reference value`

Fracture Mechanics Results (solid)

1. In the **Model Builder** window, click **Fracture Mechanics Results (solid)**.
2. In the **Settings** window for **Evaluation Group**, locate the **Data** section.
3. From the **Parameter selection (solidcrack1auxPara)** list, choose **Last**.
4. In the **Fracture Mechanics Results (solid)** toolbar, click **Evaluate**.

Compare the J-integral with the energy release rate.

J-Integral and Energy Release Rate

1. In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `J-Integral and Energy Release Rate` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Virtual Crack Extension Study/Solution 2 (solidcrack1solp)**.
4. From the **Parameter selection (Xa)** list, choose **Manual**.

5 In the **Parameter indices (1-21)** text field, type `range(2,20)`.

6 From the **Parameter selection (solidcrack1auxPara)** list, choose **Last**.

Global 1

1 Right-click **J-Integral and Energy Release Rate** and choose **Global**.

2 In the **Settings** window for **Global**, 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.J - J-integral - J/m²**.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
solid.crack1.jint1.J	J/m^2	J-integral, contour 1
solid.crack1.vce1.G	J/m^2	Energy release rate

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Xa**.

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

6 From the **Width** list, choose **2**.

7 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

8 From the **Positioning** list, choose **Interpolated**.

9 Click to expand the **Legends** section. Find the **Include** subsection. Clear the **Solution** check box.

J-Integral and Energy Release Rate

1 In the **Model Builder** window, click **J-Integral and Energy Release Rate**.

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

3 Select the **x-axis label** check box. In the associated text field, type `Crack length (m)`.

4 Select the **y-axis label** check box. In the associated text field, type `Energy release rate (J/m^2)`.

5 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

6 In the **J-Integral and Energy Release Rate** toolbar, click **Plot**.

Crack Growth Rate

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Virtual Crack Extension Study/ Solution 2 (solidcrack1solp)**.

4 From the **Parameter selection (solidcrack1auxPara)** list, choose **Last**.

Global 1

1 Right-click **Crack Growth Rate** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
dadN	1	Crack growth rate (m/cycle)

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Xa**.

5 Locate the **Coloring and Style** section. From the **Width** list, choose **2**.

Crack Growth Rate

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

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** check box. In the associated text field, type `Crack length (m)`.

6 Select the **y-axis label** check box. In the associated text field, type `Crack growth rate (m/cycle)`.

7 Locate the **Legend** section. Clear the **Show legends** check box.

8 In the **Crack Growth Rate** toolbar, click **Plot**.

Compute the total number of cycles needed for driving the crack from 0.5 m to 0.7 m.

Number of cycles

1 In the **Results** toolbar, click 8.5 **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, type `Number of cycles` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Virtual Crack Extension Study/ Solution 2 (solidcrack1solp)**.

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

Expression	Unit	Description
1/dadN	1	

5 Locate the **Data Series Operation** section. From the **Transformation** list, choose **Integral**.

6 Click **Evaluate**.