Created in COMSOL Multiphysics 6.2

Angle Crack Embedded in a Plate

Introduction

In this model, a rectangular solid containing an inner crack is submitted to tension. The crack has an angle with respect to the load direction, which implies a mix of mode I and mode II loading on the crack. The energy release rate at the crack tips is calculated using the J-integral method. The stress intensity factors are also calculated and compared to reference values from the NAFEMS benchmark (Ref. 1).

Model definition

The geometry is a rectangle of size $2h \times 2b$ with a crack of length $2a$ at the center. The angle β between the crack and the vertical axis span three values: 90°, 67.5°, and 22.5°.

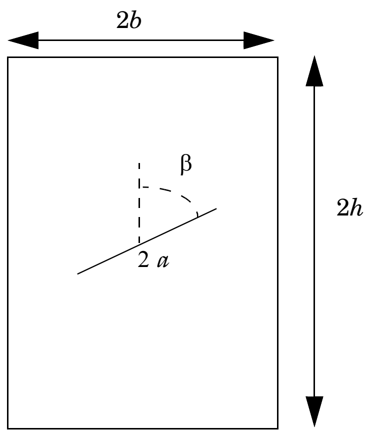


Figure 1: Crack geometry.

MATERIAL

As specified in the benchmark, the material is linear elastic with Young's modulus $E = 207$ GPa and Poisson's ratio $\nu = 0.3$.

LOADS AND CONSTRAINTS

A roller condition is applied on the bottom edge, and zero horizontal displacement is applied at the bottom-right corner to avoid rigid body motion. A uniform vertical load of $\sigma = 100$ MPa is applied on the top boundary.

J-INTEGRAL AND STRESS INTENSITY FACTORS

The energy release rate of a crack extension along the current direction of the crack can be calculated by the J-integral, which is calculated along a contour path around each crack tip:

$$J = \int_{\Gamma} W_s \mathbf{m} \cdot \mathbf{e}_1 - (\sigma \cdot \mathbf{m}) \cdot (\nabla \mathbf{u} \cdot \mathbf{e}_1) dl$$

Here, $\mathbf{e}_1$ is the unit direction vector of the crack, and $\mathbf{m}$ is the unit vector normal to the integration path.

The stress intensity factors K_I and K_{II} are calculated from the β_K ratio between mode I (opening) and mode II (sliding) displacement.

$$K_I = \sqrt{\frac{E^*}{1 + \beta_K^2} J}$$

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

Here, E^* is the equivalent Young's modulus. In 2D plane strain condition it is defined by

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

Results and Discussion

The stress plots show stress concentration at crack tips for the three angles (Figure 2-4).

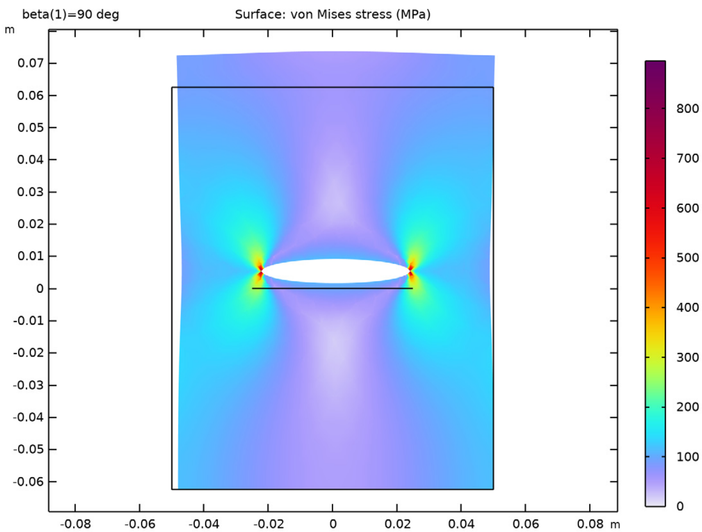


Figure 2: von Mises stress at crack angle β = *90°.*

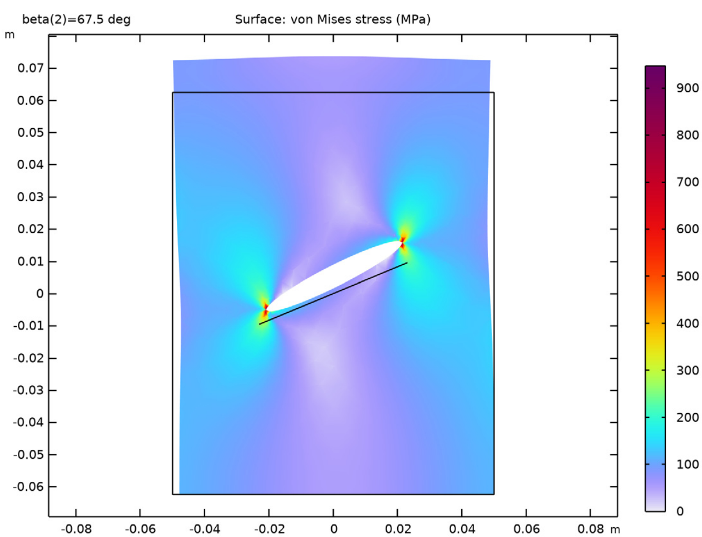


Figure 3: von Mises stress at crack angle β = *67.5°.*

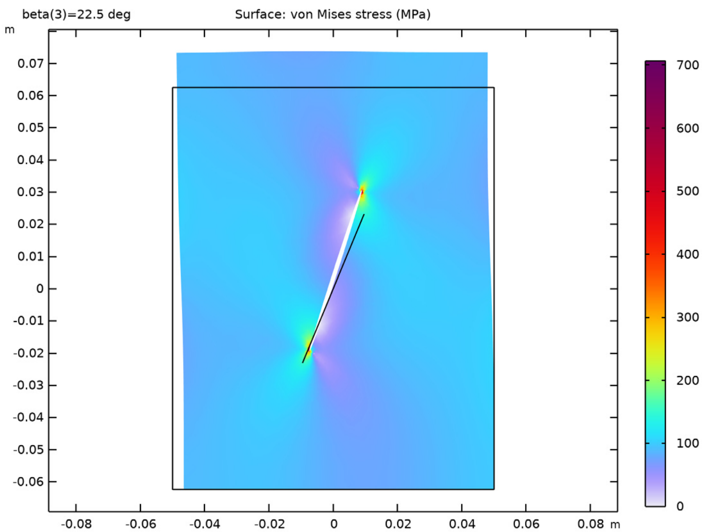


Figure 4: von Mises stress at crack angle β = *22.5°.*

The crack directions, J-integral paths, and J-integral values are also plotted by default (Figure 5-7). The value of J is maximum for the horizontal crack, and it decreases with the angle.

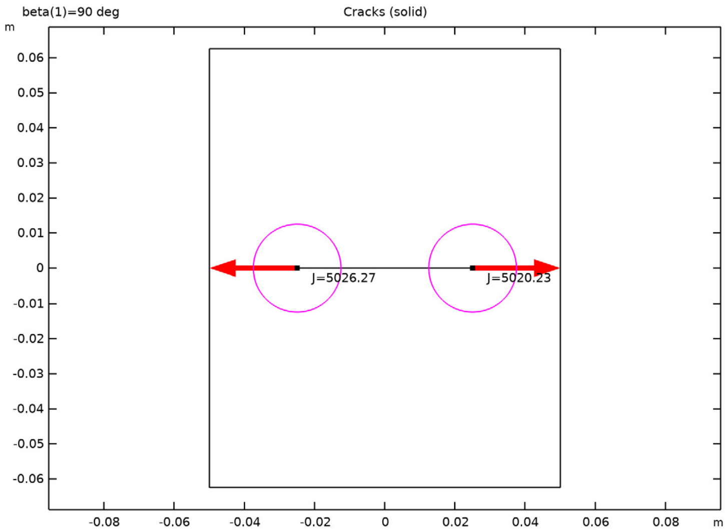


Figure 5: J-integral path and value at crack angle $\beta = 90°$.

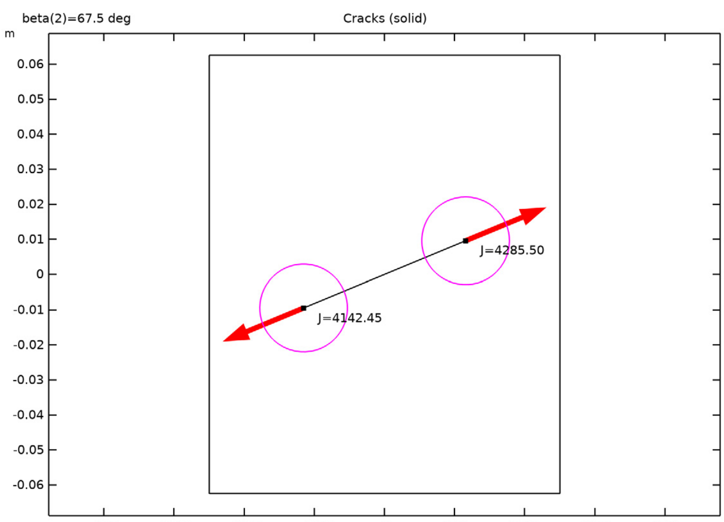


Figure 6: J-integral path and value at crack angle $\beta = 67.5°$.

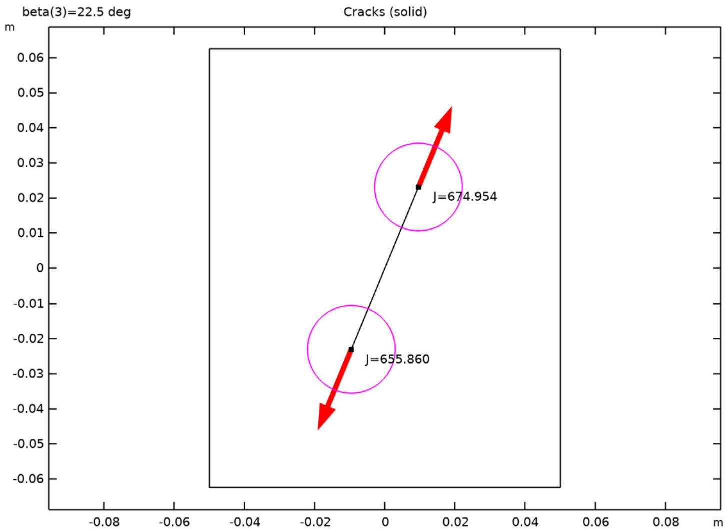


Figure 7: J-integral path and value at crack angle $\beta = 22.5°$.

The values of the stress intensity factors can be compared to values reported in Ref. 1. The stress intensity factors K_I and K_{II} are given relative to $K_0 = \sigma\sqrt{\pi a}$. The results can differ slightly depending on the platform used to build the mesh and compute the solution.

TABLE 1: COMPARISON BETWEEN COMPUTED AND REFERENCE STRESS INTENSITY FACTORS.

VARIABLE	90°	67.5°	22.5°
KI/K0, Reference	1.200	1.030	0.190
KI/K0, Left tip	1.206	1.028	0.185
KI/K0, Right tip	1.206	1.052	0.189
KII/K0, Reference	0	0.370	0.405
KII/K0, Left tip	0.012	0.377	0.392
KII/K0, Right tip	-0.012	0.367	0.400

The computed stress intensity factors are in agreement with the values reported in Ref. 1. For slanted cracks the results at the crack tips differ from each other. The difference can be explained by the fact that one side of the solid block is submitted to a roller condition, while a boundary load is applied to the other side, which makes the loading nonsymmetric.

One can see that for $\beta = 90°$ the crack mode is opening only, since K_{II} is zero. When the angle is decreased the sliding mode II appears and becomes more and more important. For β = 22.5° mode II is dominant, since $K_{II} > K_I$. This is in good accordance with the plots of opening and sliding displacements along the crack, as plotted in Figure 8.

Figure 8: Opening and sliding displacement along crack.

Reference

1. H. Pang and R. Leggatt, "*2D Test Cases in Linear Elastic Fracture Mechanics, part 3.4: Angle crack embedded in a plate*," NAFEMS, 1992.

Application Library path: `Structural_Mechanics_Module/Fracture_Mechanics/angle_crack_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
b	50[mm]	0.05 m	Half width
h0	1.25*b	0.0625 m	Half height
a	b*0.5	0.025 m	Half crack length
beta	90[deg]	1.5708 rad	Crack angle
load	100[MPa]	1E8 Pa	Applied load
K0	load/1[N/m^2]* sqrt(pi*a/1[m])	2.8025E7	Target stress intensity factor

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 `2*b`.
4 In the **Height** text field, type `2*h0`.
5 Locate the **Position** section. From the **Base** list, choose **Center**.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.
2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **x** text field, type `-a`.

5 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

6 In the **x** text field, type `a`.

Rotate 1 (rot1)

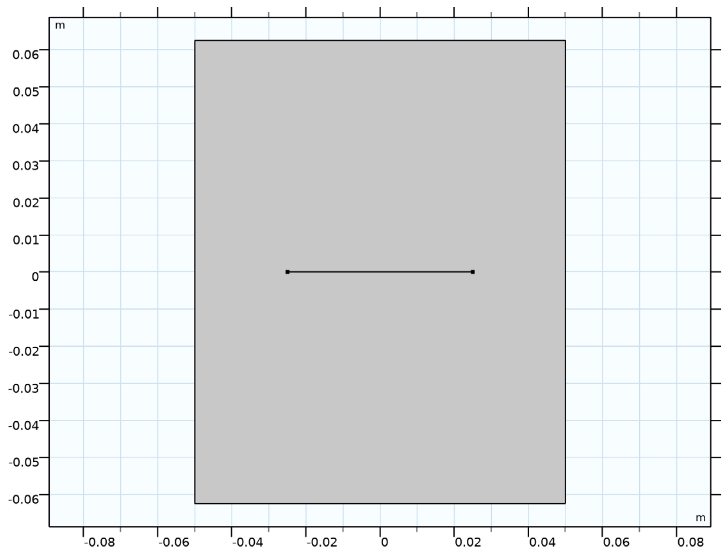
1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Select the object **ls1** only.

3 In the **Settings** window for **Rotate**, locate the **Rotation** section.

4 In the **Angle** text field, type `90-beta`.

5 Click **Build All Objects**.



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	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)

Crack 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose the boundary condition **More>Crack**.

2 Select Boundary 4 only.

Add two **J-Integral** features to evaluate J-integrals at both crack tips.

J-Integral 1

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

Crack 1

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

J-Integral 2

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

2 Select Point 4 only.

Roller 1

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

2 Select Boundary 2 only.

Prescribed Displacement 1

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

2 Select Point 5 only.

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

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

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
load	y

MESH 1

The mesh is automatically refined at crack tips. Edit the generated meshing sequence to apply a custom size.

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

2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

3 From the list, choose **User-controlled mesh**.

Size 1

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

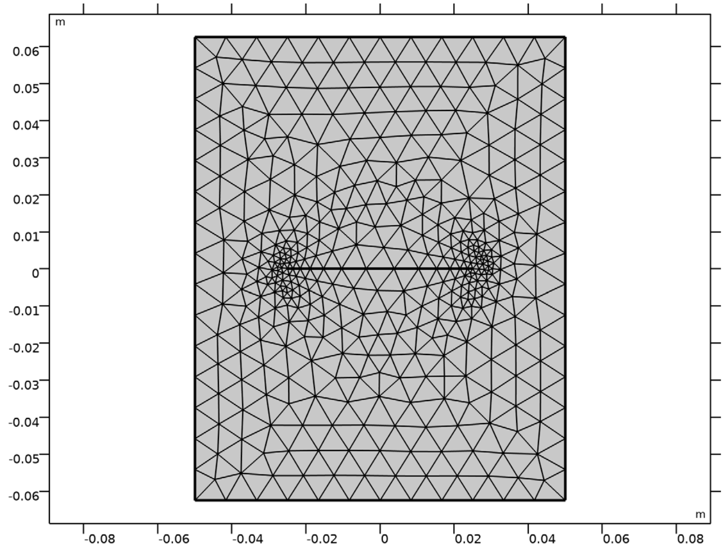
2 In the **Settings** window for **Size**, locate the **Element Size** section.

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 `a/20`.

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



STUDY 1

Add a parametric sweep to change the crack angle.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, click to select the cell at row number 1 and column number 1.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
beta (Crack angle)	90 67.5 22.5	deg

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

RESULTS

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

5 Click **Plot First** to display the results for the first angle.

6 Click **Plot Next** several times to display the results for all the angles.

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 **Study 1/Parametric Solutions 1 (sol2)>Solid Mechanics>Cracks (solid)**.

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

5 In the tree, select **Study 1/Parametric Solutions 1 (sol2)>Solid Mechanics> Fracture Mechanics Results (solid)**.

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

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

RESULTS

Cracks (solid)

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

2 In the **Settings** window for **2D Plot Group**, click **Plot First** to display the results for the first angle.

3 Click **Plot Next** several times to display the results for all the angles.

Stress Intensity Factors, Mode I

1 In the **Model Builder** window, expand the **Results>Fracture Mechanics Results (solid)** node, then click **Stress Intensity Factors, Mode I**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`solid.crack1.jint1.KI/K0`	`1`	`Stress intensity factor, mode I [crack1/jint1]`
`solid.crack1.jint2.KI/K0`	`1`	`Stress intensity factor, mode I [crack1/jint2]`

Stress Intensity Factors, Mode 2

1 In the **Model Builder** window, click **Stress Intensity Factors, Mode 2**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`solid.crack1.jint1.KII/K0`	1	Stress intensity factor, mode II [crack1/jint1]
`solid.crack1.jint2.KII/K0`	1	Stress intensity factor, mode II [crack1/jint2]

4 In the **Fracture Mechanics Results (solid)** toolbar, click **=** **Evaluate**.

Stress (solid)

In the **Model Builder** window, under **Results** right-click **Stress (solid)** and choose **Duplicate**.

Stress, Multiple Angles

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

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

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

4 Locate the **Color Legend** section. Clear the **Show legends** check box.

5 Click to expand the **Plot Array** section. Select the **Enable** check box.

Surface 1

1 In the **Model Builder** window, expand the **Stress, Multiple Angles** node, then click **Surface 1**.

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 From the **Parameter value (beta (deg))** list, choose **90**.

Line 1

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

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

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

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

5 From the **Color** list, choose **Black**.

6 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

7 From the **Parameter value (beta (deg))** list, choose **90**.

8. Click to expand the **Title** section. From the **Title type** list, choose **None**.
9. Click to expand the **Plot Array** section. Clear the **Belongs to array** check box.
10. Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.
11. Clear the **Color** check box.
12. Clear the **Color and data range** check box.
13. Clear the **Height scale factor** check box.
14. Clear the **Tube radius scale factor** check box.

Deformation 1

Right-click **Line 1** and choose **Deformation**.

Selection 1

1. In the **Model Builder** window, right-click **Line 1** and choose **Selection**.
2. Select Boundaries 1–3 and 5 only.
3. In the **Stress, Multiple Angles** toolbar, click **Plot**.

Line 1, Surface 1

1. In the **Model Builder** window, under **Results>Stress, Multiple Angles**, Ctrl-click to select **Surface 1** and **Line 1**.
2. Right-click and choose **Duplicate**.

Line 2, Surface 2

1. In the **Model Builder** window, under **Results>Stress, Multiple Angles**, Ctrl-click to select **Surface 2** and **Line 2**.
2. In the **Settings** window for **Surface**, locate the **Data** section.
3. From the **Parameter value (beta (deg))** list, choose **67.5**.
4. Click to expand the **Title** section. From the **Title type** list, choose **None**.
5. Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Line 2

1. In the **Model Builder** window, click **Line 2**.
2. In the **Settings** window for **Line**, locate the **Data** section.
3. From the **Parameter value (beta (deg))** list, choose **67.5**.
4. Locate the **Plot Array** section. Select the **Belongs to array** check box.
5. Select the **Manual indexing** check box.
6. In the **Index** text field, type 1.

Line 2, Surface 2

1 In the **Model Builder** window, under **Results>Stress, Multiple Angles**, Ctrl-click to select **Surface 2** and **Line 2**.

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

3 Right-click and choose **Duplicate**.

Line 3, Surface 3

1 In the **Model Builder** window, under **Results>Stress, Multiple Angles**, Ctrl-click to select **Surface 3** and **Line 3**.

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

3 From the **Parameter value (beta (deg))** list, choose **22.5**.

4 Click to expand the **Plot Array** section. Select the **Manual indexing** check box.

5 In the **Index** text field, type `2`.

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

Line 3

1 In the **Model Builder** window, click **Line 3**.

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

3 From the **Parameter value (beta (deg))** list, choose **22.5**.

4 Locate the **Plot Array** section. In the **Index** text field, type `2`.

Stress, Multiple Angles

1 In the **Model Builder** window, click **Stress, Multiple Angles**.

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

3 From the **View** list, choose **New view**.

4 In the **Stress, Multiple Angles** toolbar, click **Plot**.

Crack Displacement

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 Displacement` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol2)**.

Line Graph 1

1 Right-click **Crack Displacement** and choose **Line Graph**.

2 Select Boundary 4 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>Cracks>Crack displacement - m>solid.crack1.jint1.delta_u1 - Crack opening displacement**.

4 Click to expand the **Legends** section. Select the **Show legends** check box.

5 Find the **Prefix and suffix** subsection. In the **Prefix** text field, type `Opening, \beta =`.

Line Graph 2

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

2 Select Boundary 4 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>Cracks>Crack displacement - m>solid.crack1.jint1.delta_u2 - Crack sliding displacement**.

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

5 From the **Color** list, choose **Cycle (reset)**.

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

7 Find the **Prefix and suffix** subsection. In the **Prefix** text field, type `Sliding, \beta=`.

8 In the **Crack Displacement** toolbar, click **Plot**.