



THE UNIVERSITY OF
TENNESSEE
KNOXVILLE

Fracture Mechanics Term Project

Erdem Caliskan

Interaction of Inclusion and Crack by Determination of the Stress Intensity Factor

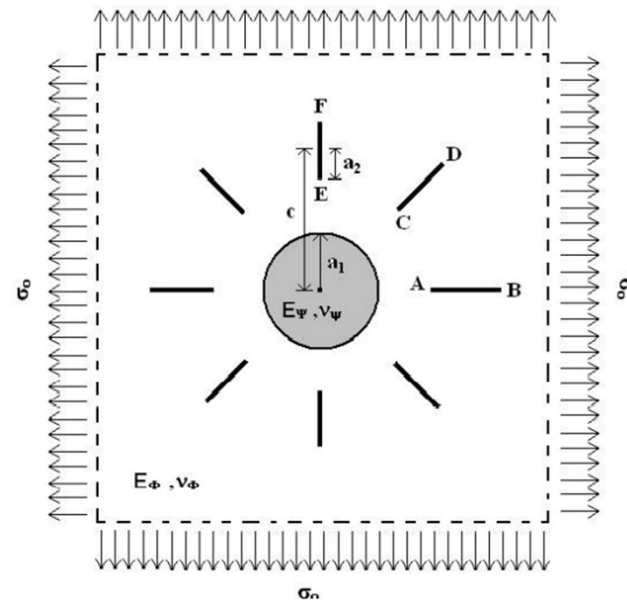


- In the following problem, which is illustrated in Figure 1, compute stress intensity factors at all crack tips denoted as A-F and fill out Table 1. These stress intensity factors are modified by the effect of an inclusion of different elastic material on the stress state around a crack in an infinite plate (with plane stress state) subjected to tensile loading. The cases without inclusion means that the whole domain has the material properties of the matrix. The Elastic Modulus of the matrix, $E\phi$, is equal to $10e3$ MPa. Use consistent units of N & mm, respectively for Force and Length, resulting in MPa for stress unit.



Tasks:

- 1. Compute J-integral for crack tip C.
- 2. Include figures of the partitioned area, mesh and the deformed shape around the crack region for distinct material case (E ratio of 10) with your final report.
- 3. Fill out table 1.



Assumptions

$$\nu_{\phi} = \nu_{\psi} = 1/3$$

$$\frac{E_{\psi}}{E_{\phi}} = 10.0$$

$$a_2 = a_1 = 1$$

$$\sigma_0 = 1.0$$

$$c = 3$$



Modeling the problem

- We will use COMSOL for the computational project.
- After creating a blank project, 2D simulation is selected. Then we select Solid Mechanics

The screenshot shows the 'Select Physics' dialog box in COMSOL. The 'Structural Mechanics' category is expanded, and 'Solid Mechanics (solid)' is selected. The right panel provides detailed information about the Solid Mechanics interface.

Select Physics

AC/DC
Acoustics
Chemical Species Transport
Electrochemistry
Fluid Flow
Heat Transfer
Radio Frequency
Structural Mechanics
Solid Mechanics (solid)
Plate (plate)
Beam (beam)

Solid Mechanics

The Solid Mechanics interface is intended for general structural analysis of 3D, 2D, or axisymmetric bodies. Typical results are displacements, stresses, and strains.

Plane stress, plane strain, or generalized plane strain assumptions can be used in a 2D analysis. In 2D axisymmetry, it is possible to include circumferential displacements.

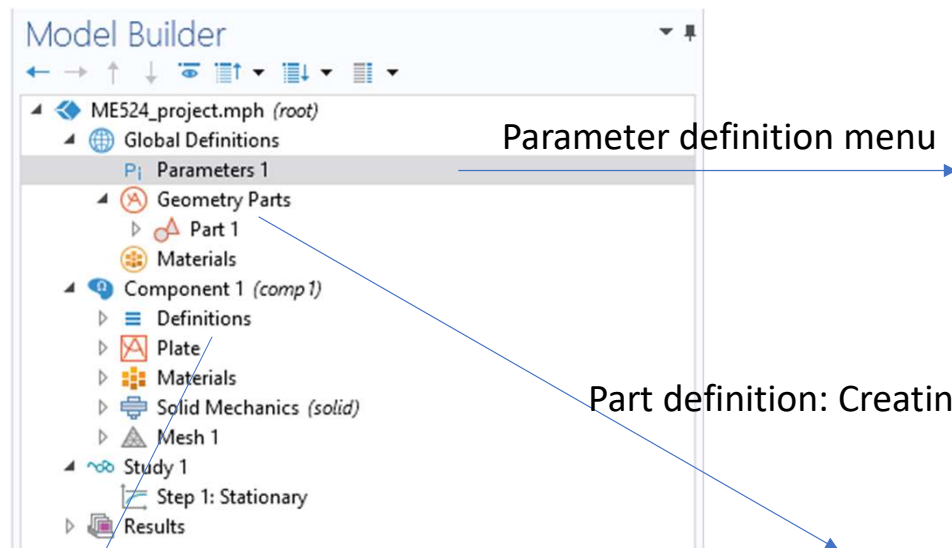
The Acoustics Module, MEMS Module, and Structural Mechanics Module add several features, for example geometric nonlinearity and advanced boundary conditions such as contact, follower loads, and nonreflecting boundaries.

With the Nonlinear Structural Materials Module or the Geomechanics Module, the interface is extended with, for example, material models for plasticity, hyperelasticity, creep, and concrete.



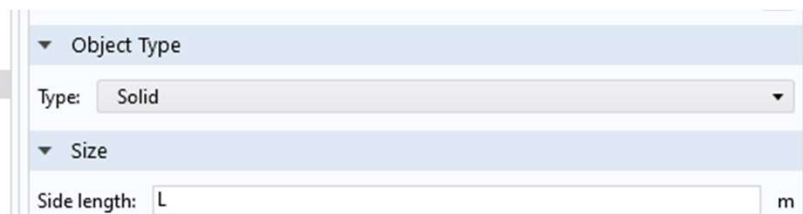
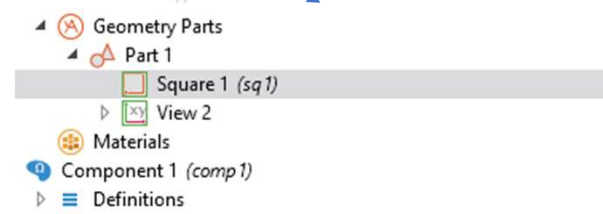
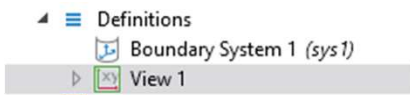
Defining the parameters

- Main menu shown below



Name	Expression	Value	Description
a	1[mm]	0.001 m	Half crack length
c	3[mm]	0.003 m	Crack center distance
E_phi	10e3[MPa]	1E10 Pa	E of the matrix
E_ksi	10*E_phi	1E11 Pa	E of the inclusion
nu_1	1/3	0.33333	Possion's ratio
sigma_0	1[MPa]	1E6 Pa	Far field load
L	15[mm]	0.015 m	Domain size

Definitions (coordinate system etc. generated automatically)





Geometric definitions

Following components are defined:

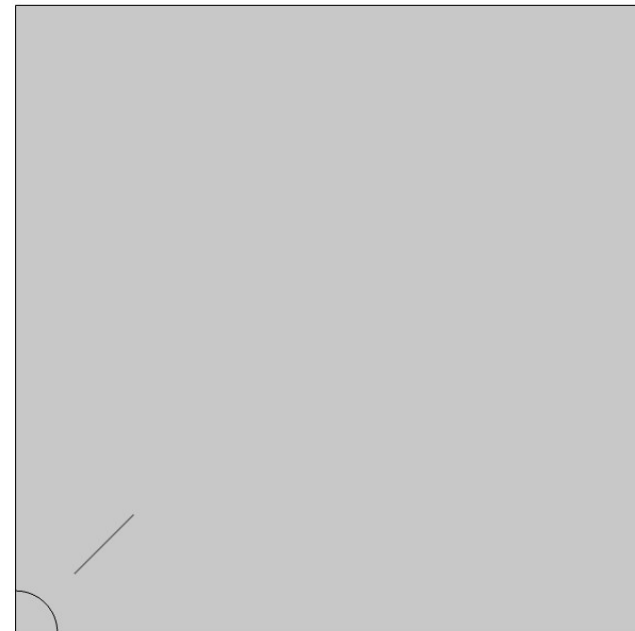
- Component 1 (*comp1*)
 - Definitions
 - Plate
 - Part Instance 1 (*pi1*)
 - Circular Arc 1 (*ca1*)
 - Point 1 (*pt1*)
 - Point 2 (*pt2*)
 - Line Segment 1 (*ls1*)
 - Rotate 1 (*rot1*)
 - Rotate 2 (*rot2*)
 - Line Segment 2 (*ls2*)
 - Point 3 (*pt3*)
 - Line Segment 3 (*ls3*)
 - Point 4 (*pt4*)
 - Line Segment 4 (*ls4*)
 - Line Segment 5 (*ls5*)
 - Line Segment 6 (*ls6*)
 - Point 5 (*pt5*)
 - Point 6 (*pt6*)
 - Line Segment 7 (*ls7*)
 - Line Segment 8 (*ls8*)
 - Line Segment 9 (*ls9*)
 - Point 7 (*pt7*)
 - Form Union (*fin*)

▼ Point

x:

y:

e.g. parameter usage





Material definitions

- Materials are defined using the parameters inputs we defined earlier

The screenshot displays the COMSOL Multiphysics Model Builder interface. On the left, the 'Model Builder' tree shows a project named 'ME524_project.mph' with a hierarchy including 'Global Definitions', 'Parameters 1', 'Geometry Parts', 'Part 1', 'Materials', 'Component 1 (comp 1)', 'Definitions', 'Plate', 'Materials', 'Matrix (mat1)', 'Basic (def)', 'Inclusion (mat2)', 'Basic (def)', 'Solid Mechanics (solid)', 'Mesh 1', 'Study 1', 'Step 1: Stationary', and 'Results'. The 'Matrix (mat1)' material is selected.

On the right, the 'Settings' window for the 'Matrix' material is shown. The 'Label' is 'Matrix' and the 'Name' is 'mat1'. Under 'Geometric Entity Selection', the 'Geometric entity level' is set to 'Domain' and 'Selection' is 'All domains'. A list shows two domains: '1 (overridden)' and '2'. The 'Material Properties' section is expanded, showing a list of property groups: Basic Properties, Acoustics, Electrochemistry, Electromagnetic Models, Equilibrium Discharge, External Material Parameters, Fluid Flow, Gas Models, Geometric Properties, Magnetostrictive Models, and Piezoelectric Models. The 'Material type' is set to 'Solid'. The 'Material Contents' table is also visible.

Property	Variable	Value	Unit	Property group
<input checked="" type="checkbox"/> Poisson's ratio	nu	nu_1	1	Basic
<input checked="" type="checkbox"/> Young's modulus	E	E_phi	Pa	Basic
<input checked="" type="checkbox"/> Density	rho	1	kg/m ³	Basic
<input type="checkbox"/> Young's modulus	E	mat1.de...	Pa	Young's modulu
<input type="checkbox"/> Poisson's ratio	nu	mat1.de...	1	Young's modulu



Physics definitions

- Component 1 (comp1)
 - Definitions
 - Plate
 - Materials
 - Matrix (mat1)
 - Basic (def)
 - Inclusion (mat2)
 - Basic (def)
 - Solid Mechanics (solid)
 - Linear Elastic Material 1
 - Free 1
 - Initial Values 1
 - Symmetry 1
 - Symmetry 2
 - Boundary Load 1
 - Boundary Load 2
 - Crack 1
 - J-Integral 1
 - J-Integral 2
 - J-Integral 3
 - J-Integral 4
 - Crack 2
 - J-Integral 1
 - J-Integral 2
 - J-Integral 3
 - J-Integral 4
 - Crack 3
 - J-Integral 1
 - J-Integral 2
 - J-Integral 3
 - J-Integral 4

Generated by default

The screenshot shows the 'Settings' window for a 'Symmetry' definition. The 'Boundary Selection' section is set to 'Manual' and lists five items: 1, 3, 4 (overridden), and 5. The 'Graphics' window shows a rectangular plate with a vertical crack on the left edge. A blue arrow points from the 'Symmetry 1' definition in the tree to the 'Symmetry' settings window, and another blue arrow points from the 'Symmetry 1' definition to the crack in the graphics window.

e.g. symmetry definition

Overwritten since we defined a crack to that boundary

The screenshot shows the physics definition toolbar with various categories and their respective icons. The 'Crack' icon is highlighted in yellow. The categories and their items are:

- Boundaries**: Crack, Boundary Load, Symmetry
- Solid Mechanics**: Boundary Load, Fixed Constraint, Prescribed Displacement, Roller
- Connections**: Rigid Connector, Attachment, Periodic Condition
- Mass, Spring, and Damper**: Spring Foundation, Added Mass, Thin Elastic Layer, Thin-Film Damping
- More Constraints**: Symmetry, Antisymmetry, Prescribed Velocity, Prescribed Acceleration, Free
- More**: Crack, Low-Reflecting Boundary

Defining crack



Settings
Crack

Label: Crack 1

Boundary Selection

Selection: Manual

8

Override and Contribution

Coordinate System Selection

Coordinate system: Global coordinate system

Crack Definition

Crack surface: Slit

Crack Orientation

Crack orientation: From crack

Crack Definition

- Crack surface: Slit
- Slit
- From geometry
- Symmetric

Graphics

mm

15

14

13

12

11

10

9

8

7

6

5

4

3

2

1

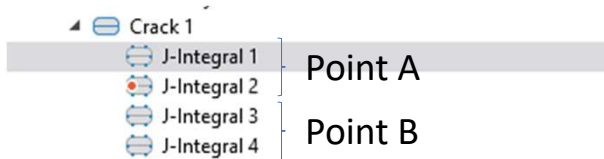
0

We select the edge that corresponds to a crack

Slit has been selected for this crack. Other two are symmetric



J-integral definition



The reason for having two J-integrals, we have two different radius for the contour integral

For this case it is 0.5 of the crack size,

We also calculate for 0.9 crack size (results were very similar, 0.5 results have been used)

Settings
J-Integral

Label: J-Integral 1

Point Selection

9

Override and Contribution

Equation

J-Integral

Integration path:
Circular

Radius of contour integral:
 r_{Γ} solid.crack1.crackSize*0.5 m

Stress Intensity Factors

Averaging region

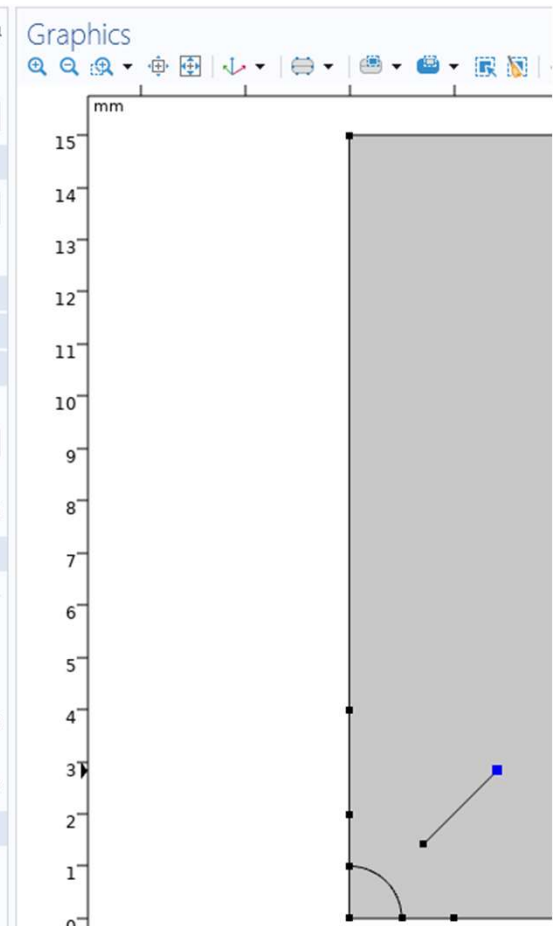
$$K_{I,II} = \frac{1}{c-b} \int_b^c \tilde{K}_{I,II} d\xi$$

Lower bound:
b 0.2*solid.crack1.crackSize m

Upper bound:
c 0.5*solid.crack1.crackSize m

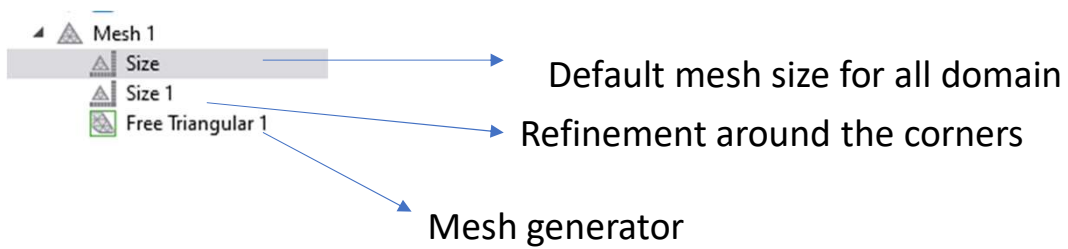
Integration

All definitions are defined by default, We only changed 0.5 to 0.9 in the J-integral 2 and 4

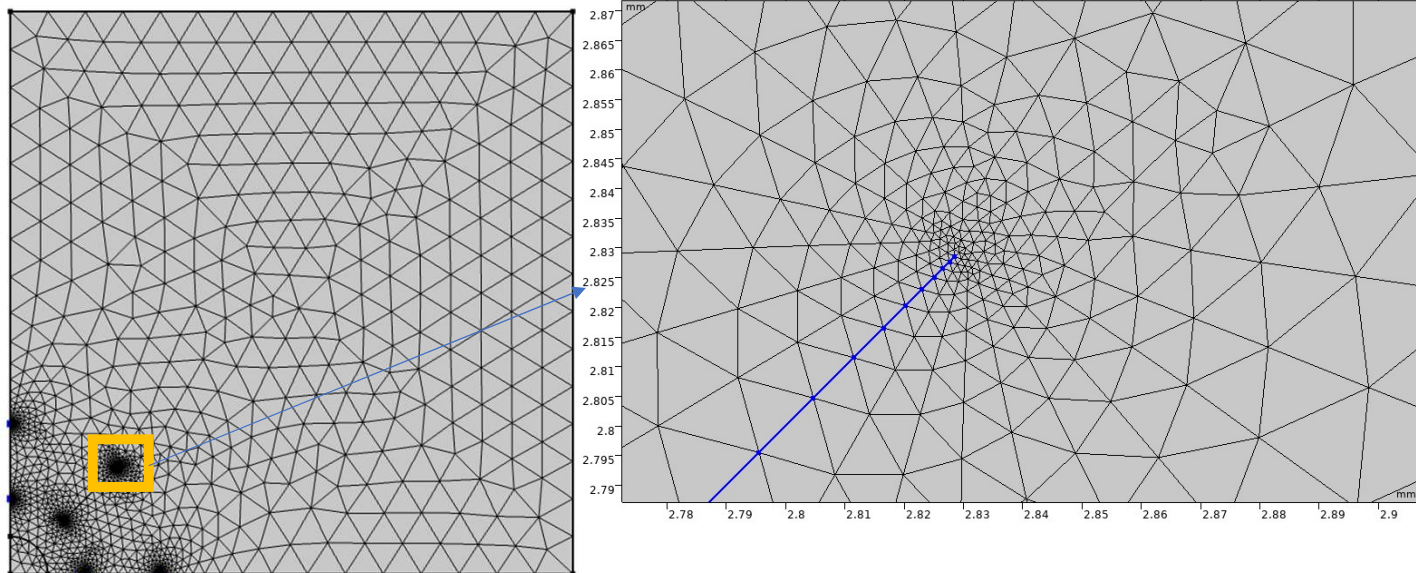




Meshing



Resulting mesh



Settings

Size

Build Selected Build All

Label: Size 1

Geometric Entity Selection

Geometric entity level: Point

Selection: Manual

3
4
7
8
9
10

Element Size

Calibrate for:

General physics

Predefined Extremely fine

Custom

Element Size Parameters

Maximum element size: 0.01 mm

Minimum element size: From sequence mm

Maximum element growth rate: From sequence

Curvature factor: From sequence

Resolution of narrow regions: From sequence



Solver configuration

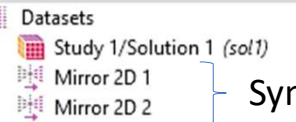
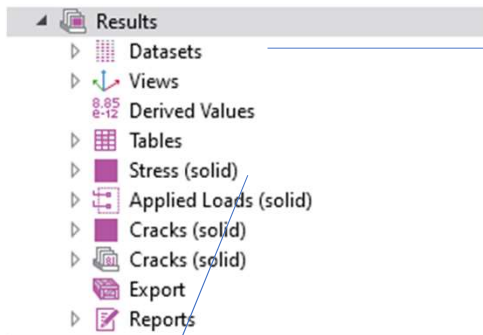
Study 1

- Step 1: Stationary
 - Solver Configurations
 - Solution 1 (sol1)
 - Compile Equations: Stationary
 - Dependent Variables 1
 - Displacement field (comp1.u)
 - Stationary Solver 1
 - Direct
 - Advanced
 - Fully Coupled 1

All parameters are default in this section.



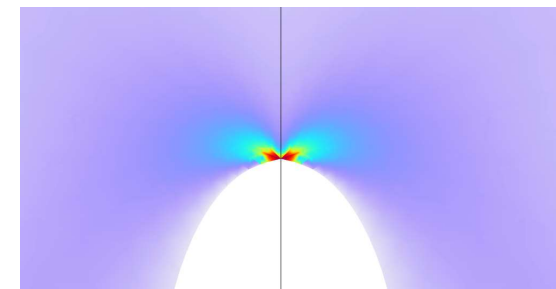
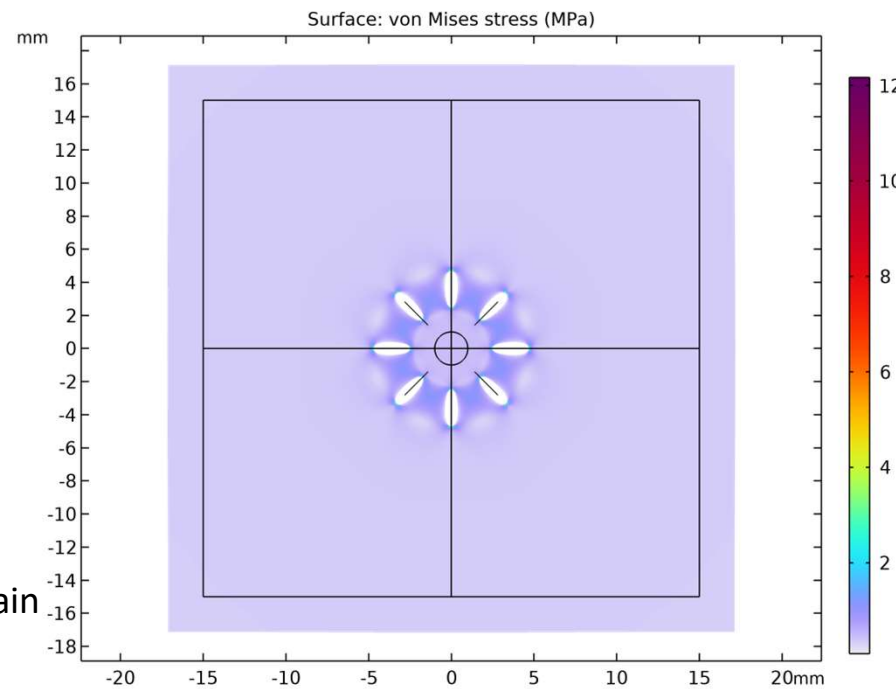
Results



Symmetries are recovered visually by defining mirrors

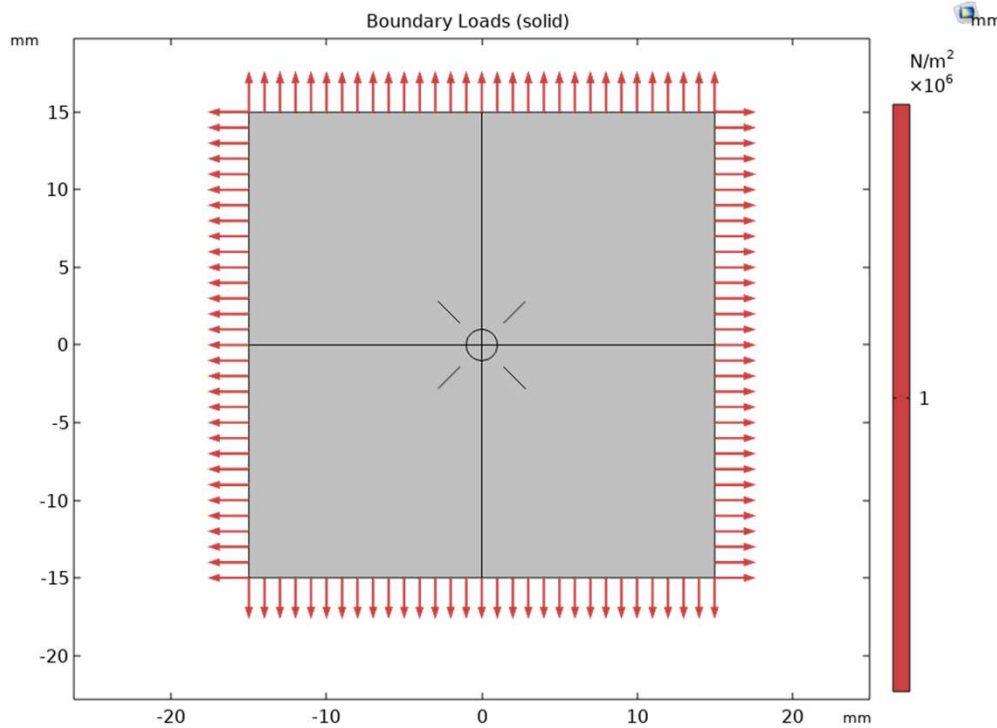


Mirror is selected to plot the domain

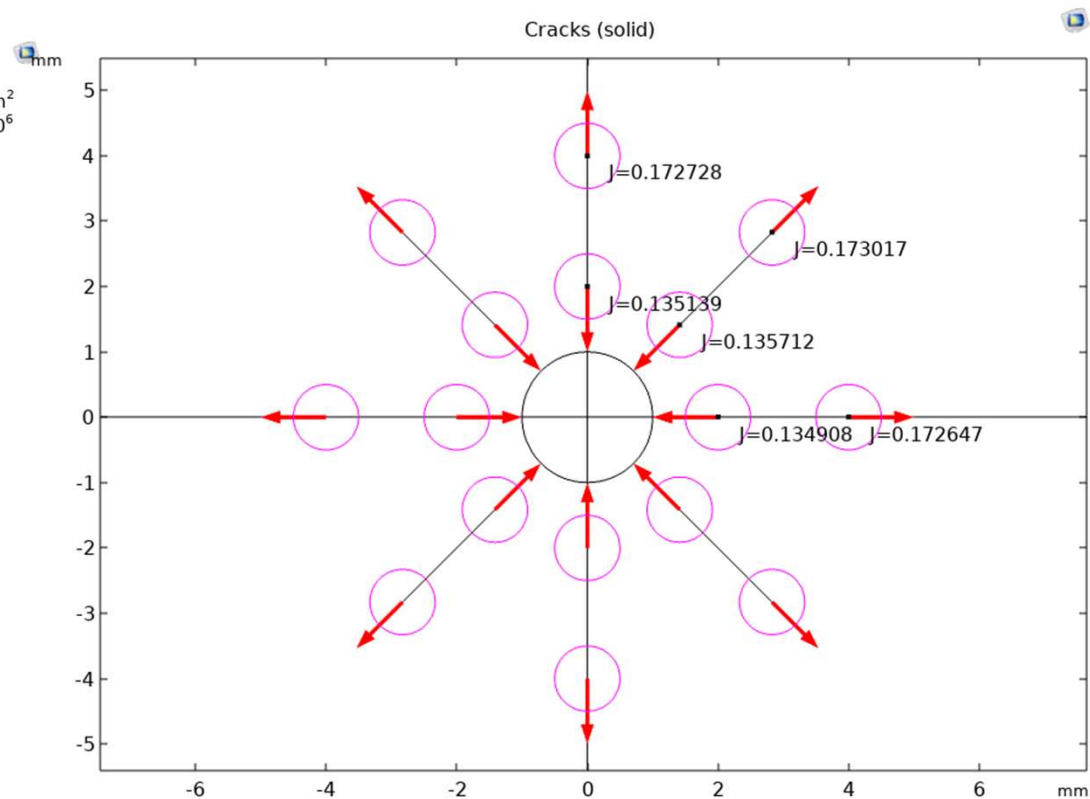




Boundary Loads and cracks

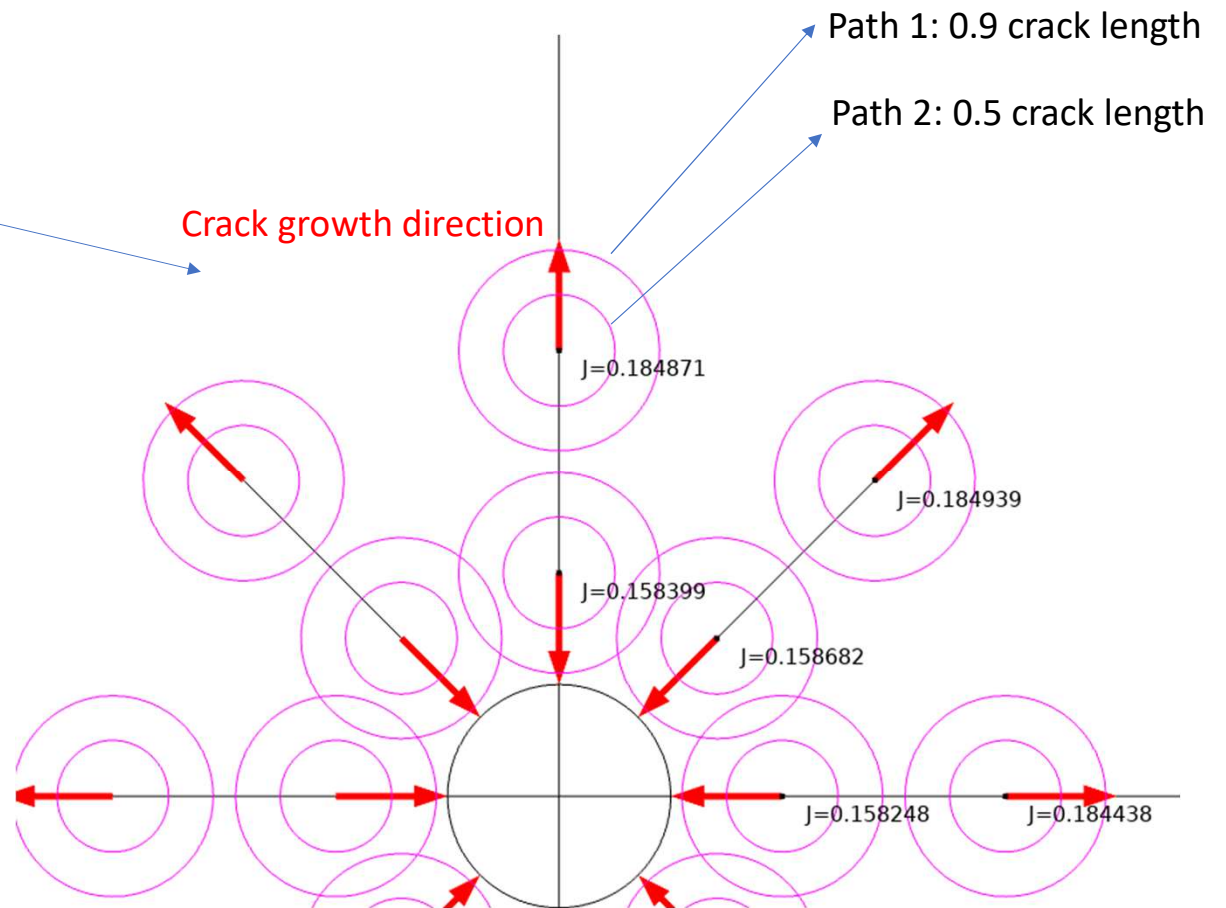
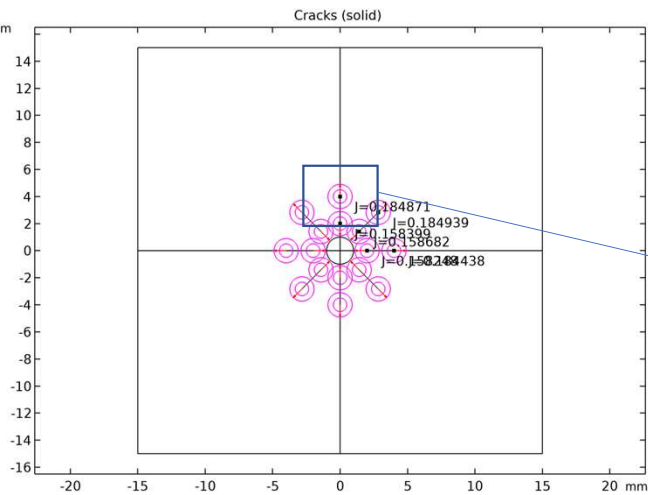


J-integral at point C = 0.13571 J/m²



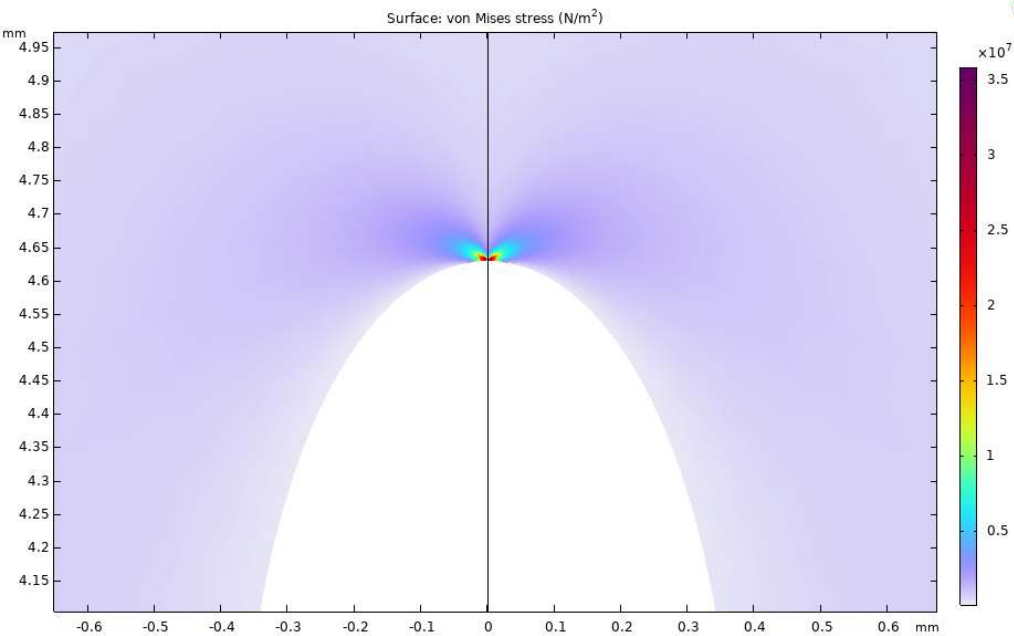


Cracks



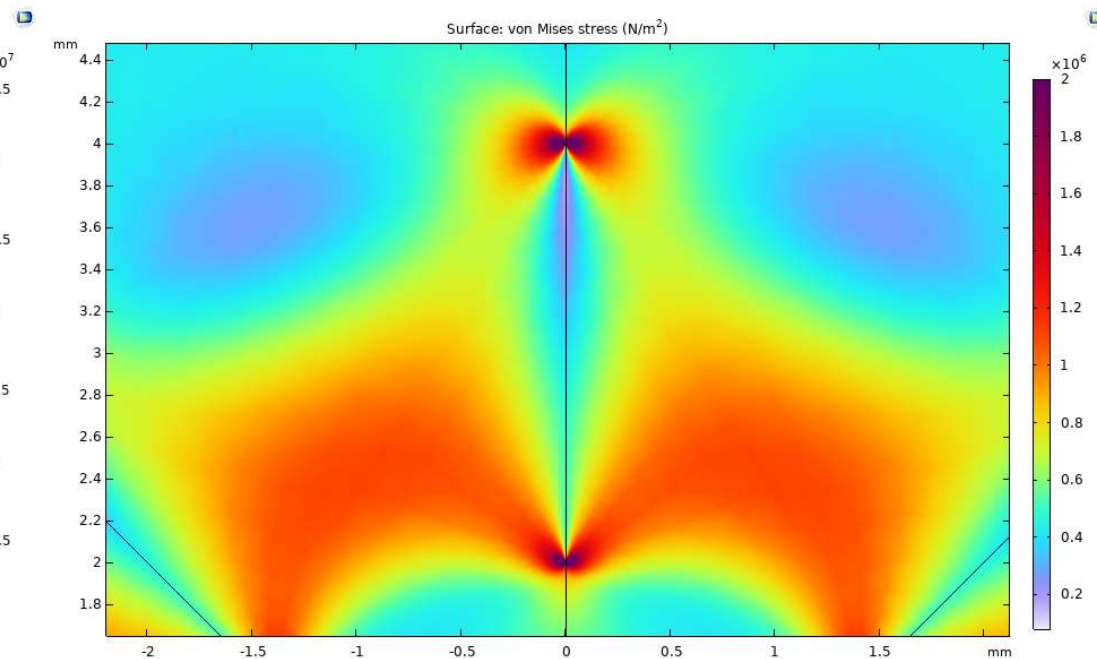


Stress plot around crack



Deformation is scaled

Far field v-M stress = 0.33 MPa
v-M stress at the angled crack = 26.7 Mpa
v-M stress at the vertical crack = 36.3 MPa

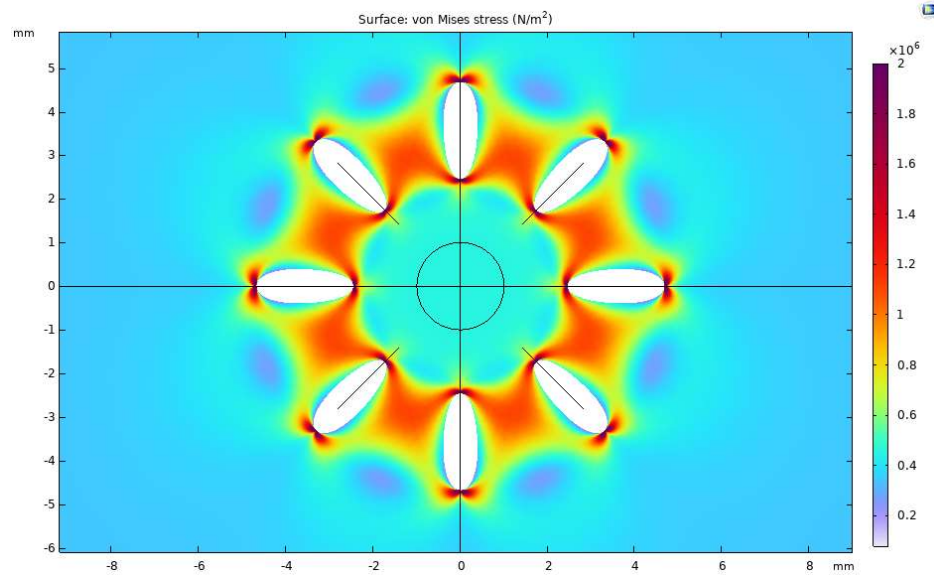


Deformation is not scaled

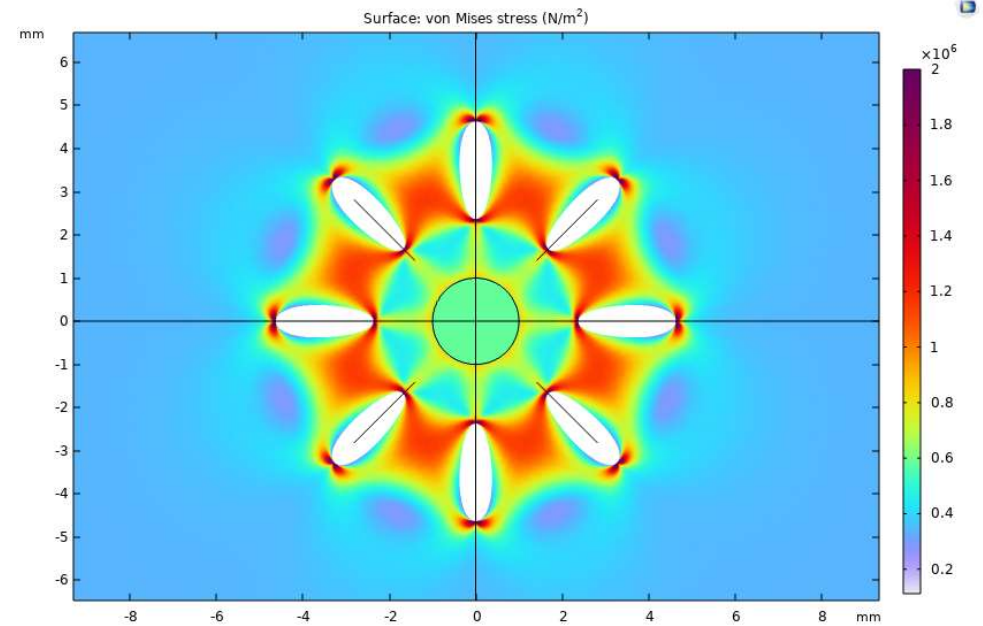


Stress plots

Matrix only



Matrix + Inclusion





Matrix only results

Crack tip	J-integral (J/m ²)	K _I (MPa/mm ^{1/2})	K _{II} (MPa/mm ^{1/2})
A	0.15826	42195	0
B	0.18492	45611	0
C	0.15904	42299	0.31425
D	0.18537	45667	-0.83114
E	0.1585	42227	0
F	0.185	45621	0

↓ Numerical error

Matrix + Inclusion results

Crack tip	J-integral (J/m ²)	K _I (MPa/mm ^{1/2})	K _{II} (MPa/mm ^{1/2})
A	0.13491	38958	0
B	0.17265	44071	0
C	0.13571	39074	0.28458
D	0.17302	44118	-0.83295
E	0.13514	38991	0
F	0.17273	44082	0

Stiffer inclusion resulted lower stress intensity factors.