

**Name: Nguyen Xuan Binh**

**Student ID: 887799**

## **Fracture Mechanics Assignment 5**

### **A? Computational problem**

A 3d printed polymer component has developed a crack as shown in Fig. 1. You have been asked to predict the maximum load  $P$  that the structure can support.

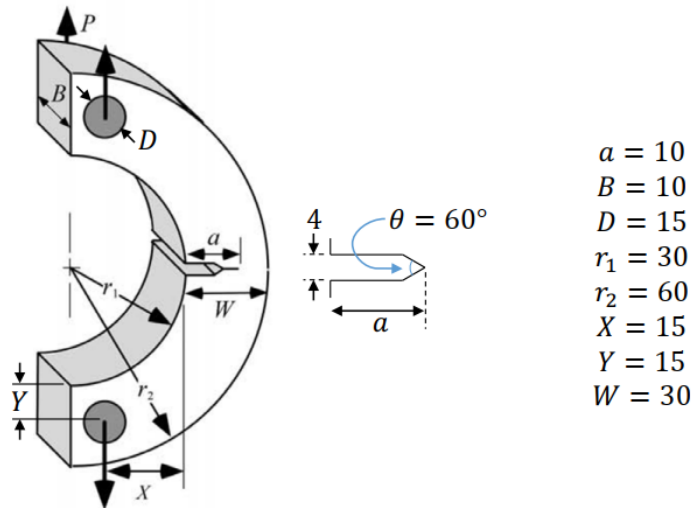


Figure 1: Geometry of the component. All dimensions are in mm.

Experiments have been done to measure the material properties of this 3d printed polymer. First, a tensile test was done using a standard dogbone geometry with dimensions shown in Fig. 2a. Second, a fracture toughness test was conducted on a Compact Tension (CT) sample with dimensions given in Fig. 2b. The load and extension measured during each test are given in separate *.xlsx* files on myCourses.

### **A? Part 1: material properties**

First, calculate the material properties of the polymer.

1. Plot the stress-strain curve of the material using the tensile test results provided in the *.xlsx* file and the dimensions shown in Fig. 2a. Use the stress-strain curve to determine the Young's modulus  $E$  of the material.

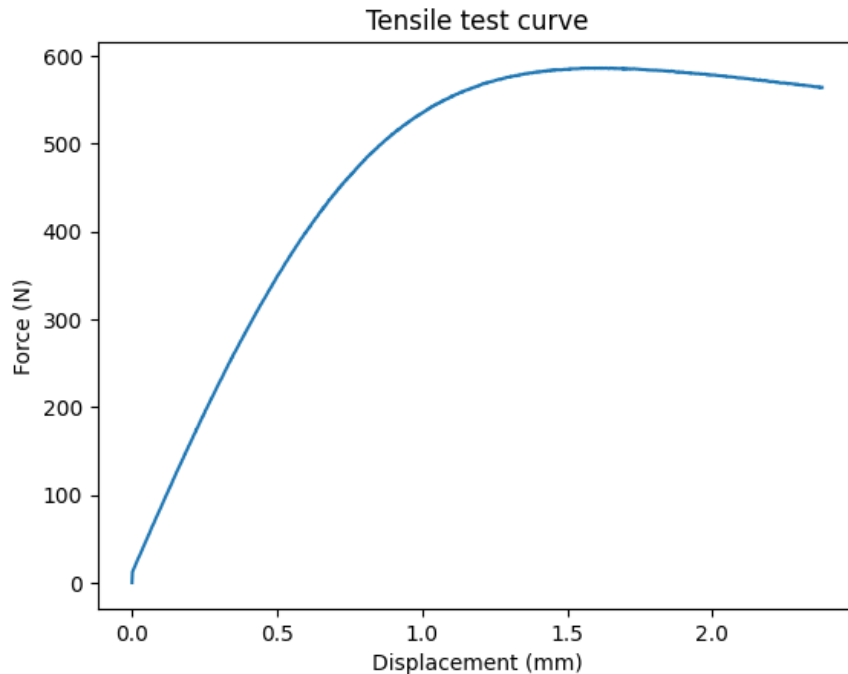
In this exercise, I use Python for data processing and calculations

First, we can plot the stress strain curve recorded in the Tension\_test.xlsx file

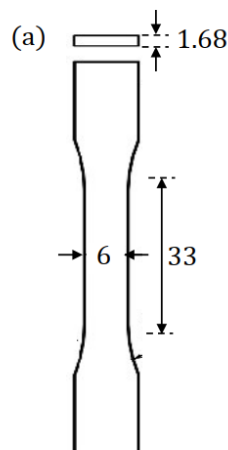
```
# Read from Tensile_test.xlsx
df = pd.read_excel('Tensile_test.xlsx')
forceTensile = df['Force (N)']
displacementTensile = df['Extension (mm)']

# Plot stress strain curve
```

```
plt.title('Tensile test curve')
plt.plot(displacementTensile, forceTensile)
plt.xlabel('Displacement (mm)')
plt.ylabel('Force (N)')
```

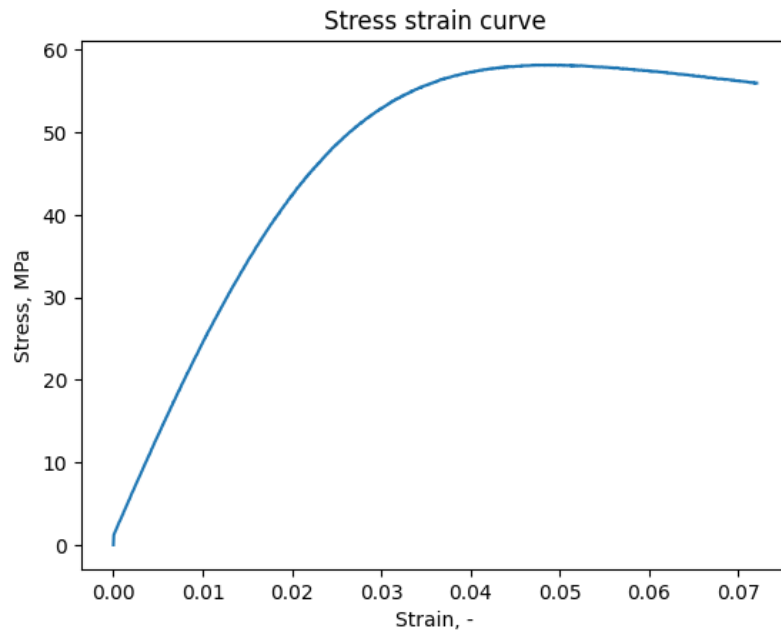


Force displacement curve cannot derive the Young's modulus, so the geometry of the dogbone specimen needs to be considered. Stress is obtained by dividing the force by the cross-sectional area, and strain is obtained by dividing displacements by original length



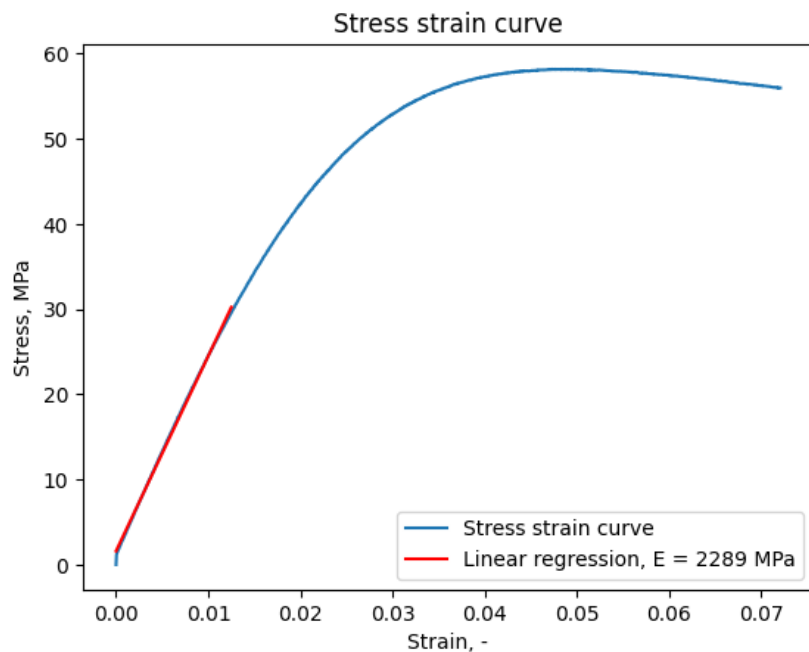
```
# Cross section properties
width = 6 # mm
thickness = 1.68 # mm
# Original length
L0 = 33 # mm
area = width * thickness # mm^2
stress = forceTensile / area # N/mm^2 = MPa
strain = displacementTensile / L0 # mm/mm = dimensionless
```

```
# Plot stress strain curve
plt.title('Stress strain curve')
plt.plot(strain, stress)
plt.xlabel('Strain, -')
plt.ylabel('Stress, MPa')
```



I proceed to fit a linear regression to the first 300 datapoints to measure the slope of the regression, which is the Young's modulus. The result is

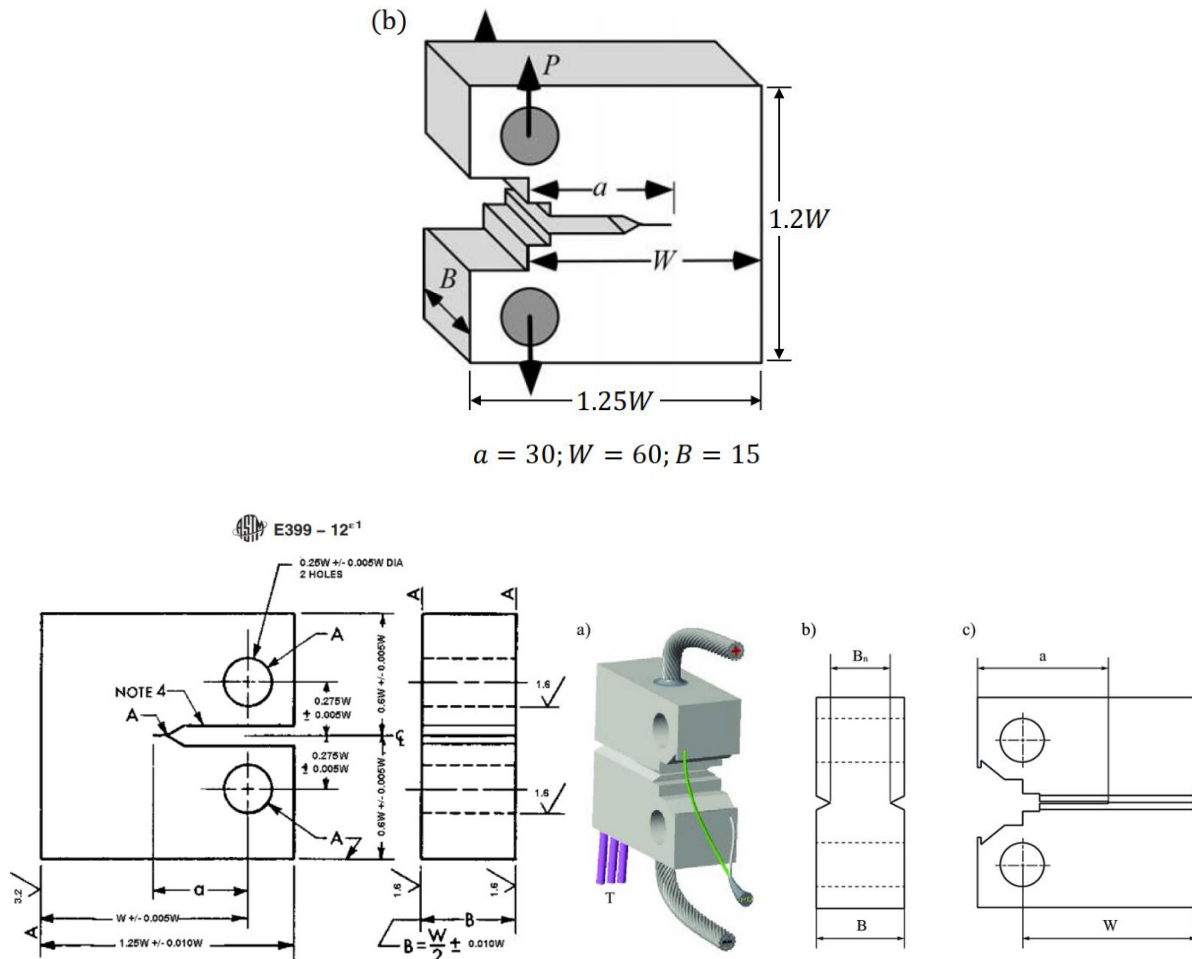
Youngs modulus: 2289 MPA (ANSWER)



However, this result varies greatly depending on the number of initial points. If 400 datapoints are taken, Young's modulus would be 2180 MPa.

2. Calculate the fracture toughness  $K_{Ic}$  of the material using the experimental data. The formula to compute  $K_{Ic}$  for the CT specimen is given in Appendix A4 of ASTM E399 (which is included on myCourses).

The formula to compute  $K_{Ic}$  is in page 18-19 in the ASTM E399 file. The formulas are based on the compact specimen geometry in Appendix A4



NOTE 1—Surface finishes in  $\mu\text{m}$ .

NOTE 2—A surfaces shall be perpendicular and parallel to within  $0.002 W$  TIR.

NOTE 3—The intersection of the crack starter notch tips with the two specimen surfaces shall be equally distant from the top and bottom edges of the specimen within  $0.005 W$ .

NOTE 4—Integral or attachable knife edges for clip gage attachment to the crack mouth may be used (see Figs. 3 and 4).

NOTE 5—For starter notch and fatigue crack configuration see Fig. 5.

NOTE 6— $1.6 \mu\text{m} = 63 \mu\text{in.}$ ,  $3.2 \mu\text{m} = 125 \mu\text{in.}$

**FIG. A4.1 Compact C(T) Specimen—Standard Proportions and Tolerances**

**A4.5.3 Calculation of  $K_Q$** —Compact specimen  $K_Q$  is calculated in SI or inch-pound units of  $\text{Pa}\sqrt{\text{m}}$  ( $\text{psi}\sqrt{\text{in.}}$ ) as follows (see Note A4.2):

$$K_Q = \frac{P_Q}{\sqrt{B B_N} \sqrt{W}} \cdot f\left(\frac{a}{W}\right) \quad (\text{A4.1})$$

where:

$$f\left(\frac{a}{W}\right) = \frac{\left(2 + \frac{a}{W}\right) \left[ 0.886 + 4.64 \frac{a}{W} - 13.32 \left(\frac{a}{W}\right)^2 + 14.72 \left(\frac{a}{W}\right)^3 - 5.6 \left(\frac{a}{W}\right)^4 \right]}{\left(1 - \frac{a}{W}\right)^{3/2}} \quad (\text{A4.2})$$

for which:

$P_Q$  = force as determined in 9.1.1, N (lbf),

$B$  = specimen thickness as determined in 8.2.1, m (in.),

$B_N$  = specimen thickness between the roots of the side grooves, as determined in 8.2.1, m (in.),

$W$  = specimen width (depth) as determined in A3.4.1, m (in.), and

$a$  = crack size as determined in 8.2.3 and A4.4.1, m (in.).

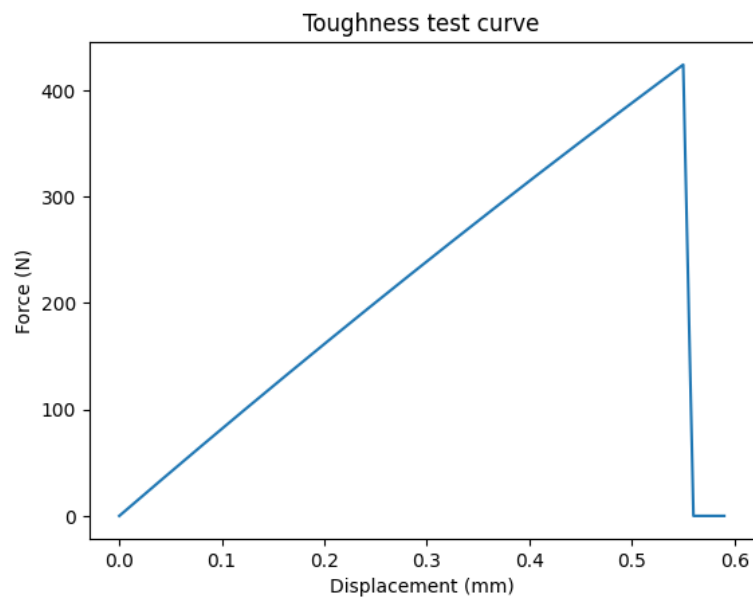
NOTE A4.1—Example: for  $a/W = 0.500$ ,  $f(a/W) = 9.66$ .

NOTE A4.2—This expression for  $a/W$  is considered to be accurate within 0.5 % over the range  $0.2 \leq a/W \leq 1$  (18, 21).

In this case,  $B = B_N$  according to the geometry given in the exercise

First, I plot the toughness test curve

```
df = pd.read_excel('Toughness_test.xlsx')
forceToughness = df['Force (N)']
displacementToughness = df['Extension (mm)']
plt.title('Toughness test curve')
plt.plot(displacementToughness, forceToughness)
plt.xlabel('Displacement (mm)')
plt.ylabel('Force (N)')
```



In the graph, the force is applied until the fracture occurs and the force is dropped to 0.

```
# Maximum force at fracture moment
P = max(forceToughness)
print(f"Critical force: {round(P)} N")
```

Critical force at fracture moment:

```
Critical force: 424 N
```

Then, we can apply the formula above:

```
# Compact tension specimen
a = 0.030 # m
W = 0.06 # m
# We have B = Bn
B = 0.015 # m
Bn = 0.015 # m

def f(a, W):
    aW = a / W
    nominator = (2 + aW) * (0.886 + 4.64 * aW - 13.32 * aW**2 + 14.72 * aW**3
- 5.6 * aW**4)
    denominator = (1 - aW) ** (3/2)
    return nominator/denominator

# Return Kic of unit Pav(m)
def Kic_equation(P, B, Bn, a, W):
    return P/(np.sqrt(B * Bn) * np.sqrt(W)) * f(a, W)

# The fracture toughness Kic of the material is:
Kic = Kic_equation(P, B, Bn, a, W)
print(f"Fracture toughness: {round(Kic) * 1e-6} MPav(m)")
```

The fracture toughness is:

```
Fracture toughness: 1.11485 MPav(m)
```

(ANSWER)

## A? Part 2: creating the Finite Element model

The next step is to create a Finite Element (FE) model of the part shown in Fig. 1. This will be done with the commercial software Abaqus. Please consult *Abaqus.pptx* (on myCourses) for instructions on how to access Abaqus.

These are the model and output files of the FE model of the part shown in Figure 1

Name	Date modified	Type	Size
model	28/05/2023 17.48	CAE File	1 024 KB
model.rec	28/05/2023 18.47	REC File	3 KB
Ring	28/05/2023 18.47	MS-DOS Applicati...	4 KB
Ring.dat	28/05/2023 18.47	DAT File	11 KB
Ring.inp	28/05/2023 18.47	INP File	42 KB
Ring.ipm	28/05/2023 18.47	IPM File	4 KB
Ring	28/05/2023 18.47	Text Document	1 KB
Ring	28/05/2023 18.47	Outlook Item	9 KB
Ring	28/05/2023 18.47	ODB File	397 KB
Ring.prt	28/05/2023 18.47	PRT File	25 KB
Ring	28/05/2023 18.47	STA File	1 KB

Originally, I thought the commercial software refers to the license version, but it turns out to be Abaqus's student version. Therefore, the output files above can only be loaded in Student's version and not the Full License version

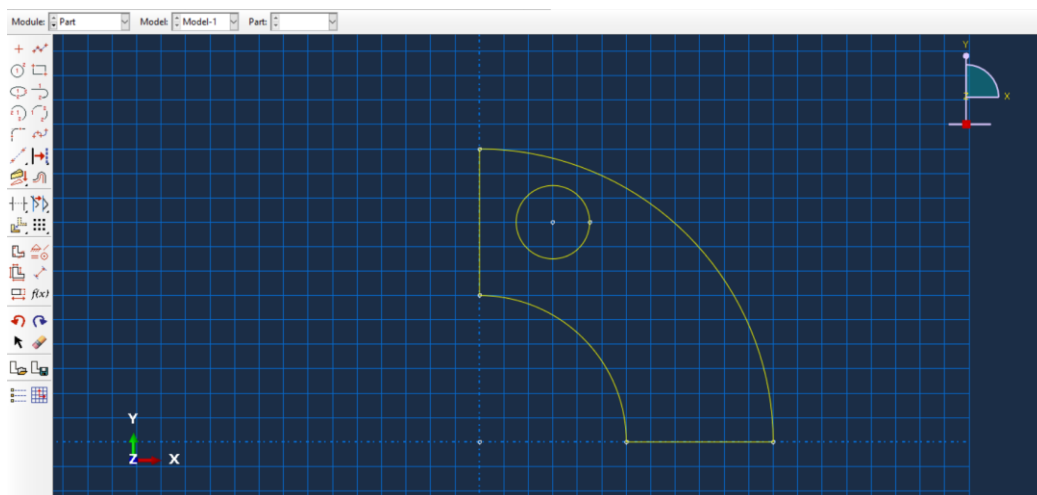
## A? Part 3: report

For this assignment, please submit a short report (max 3 pages) including the following sections:

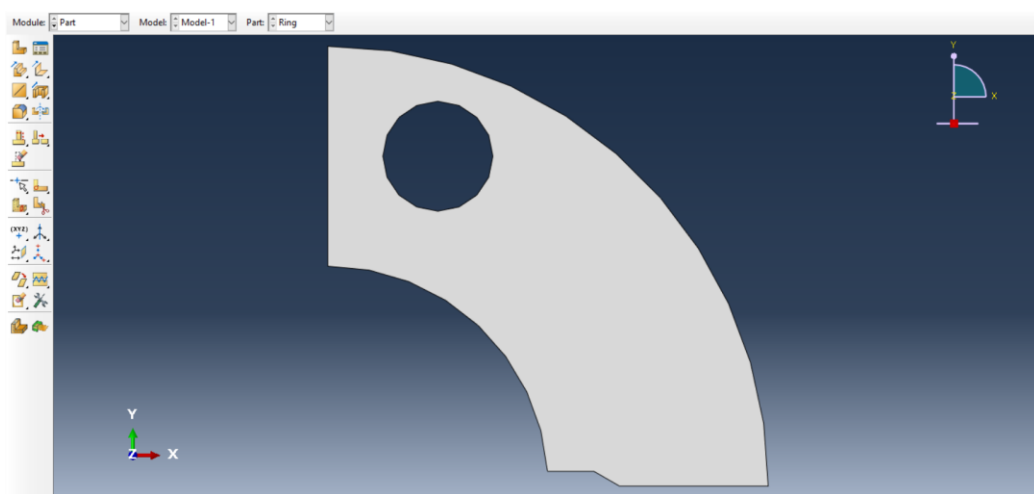
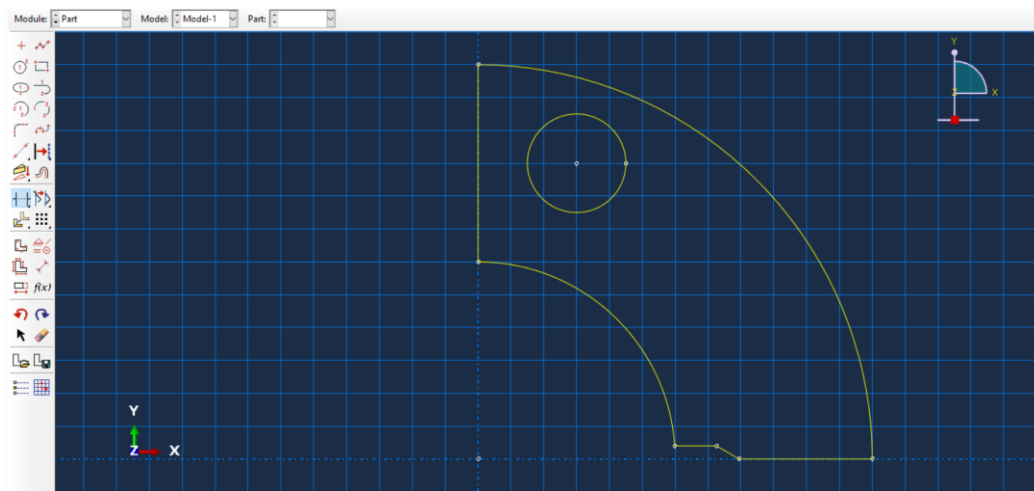
**Modelling approach** Here, describe the geometry modelled, boundary conditions, material properties, the type of elements, mesh size and the analysis (plane stress/strain). You can use figures to make your description clearer. With this description someone should be able to recreate the model in Abaqus.

### Geometry modelling

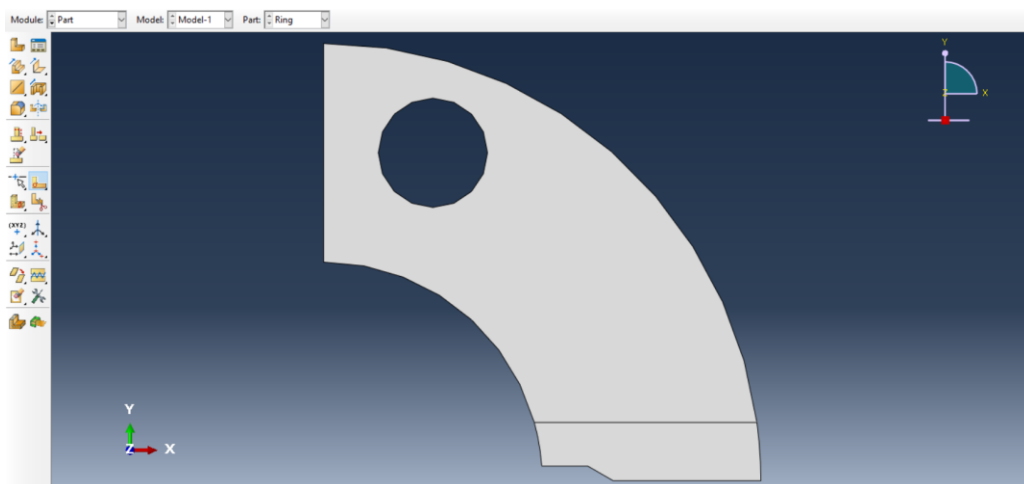
Creating geometry without the crack using shell planar 2D sketch. Sketch drawing size of 0.2 (200 mm). We only need to model half of the specimen as it is Y-symmetrical



Add the crack at the lower end. The angle of the crack is 30 degrees



Partitioning Face sketch. This is helpful for creating different seed sizes for the two separate regions, as the region near the boundary conditions should have finer mesh.



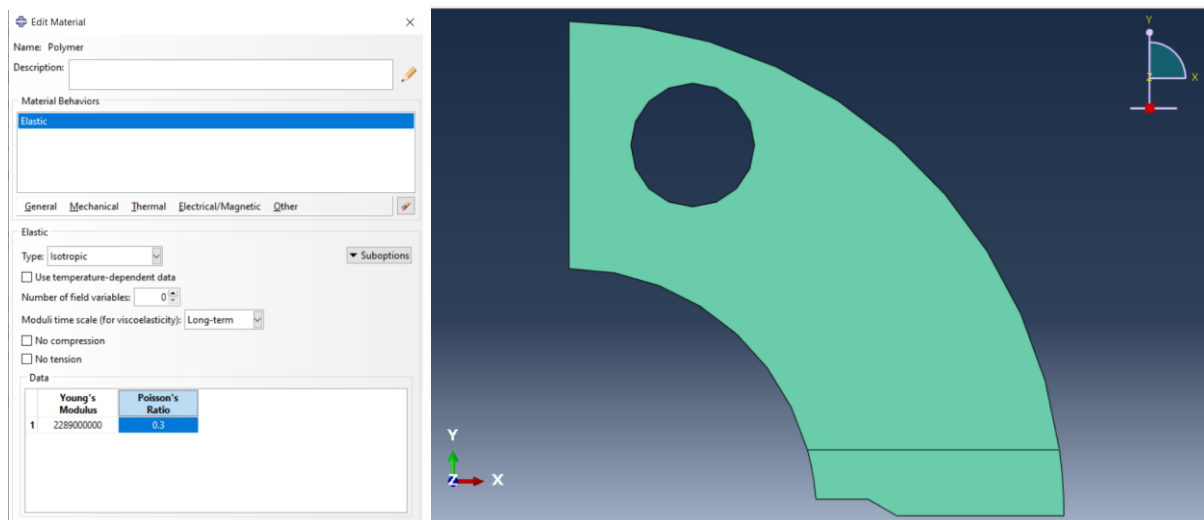


## Polymer material definition and section assignment

Elasticity: Young's Modulus of  $2.289 \times 10^9$  Pa and Poisson's ratio of 0.3

Plasticity: no plasticity for polymer material

After that, we assign the material to a section. Then we assign the model to the section

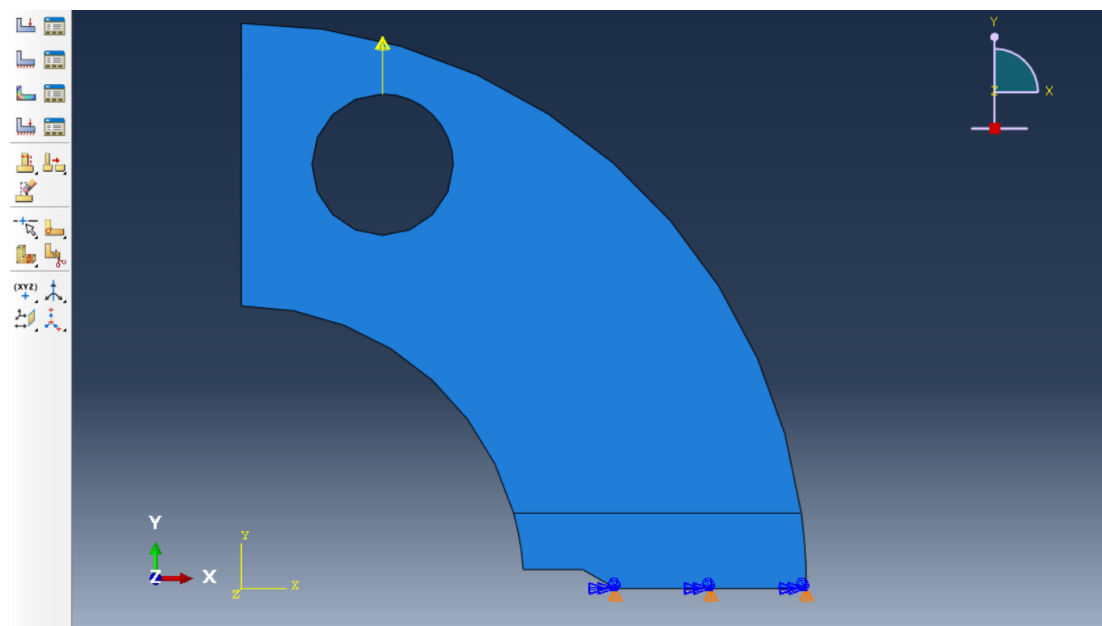


## Create assembly, add time step, loading definition and boundary conditions

Time step: choosing default settings

Loading: apply the load on as a concentrated mechanical force in upward direction (Y-axis) on the top of the circular nod

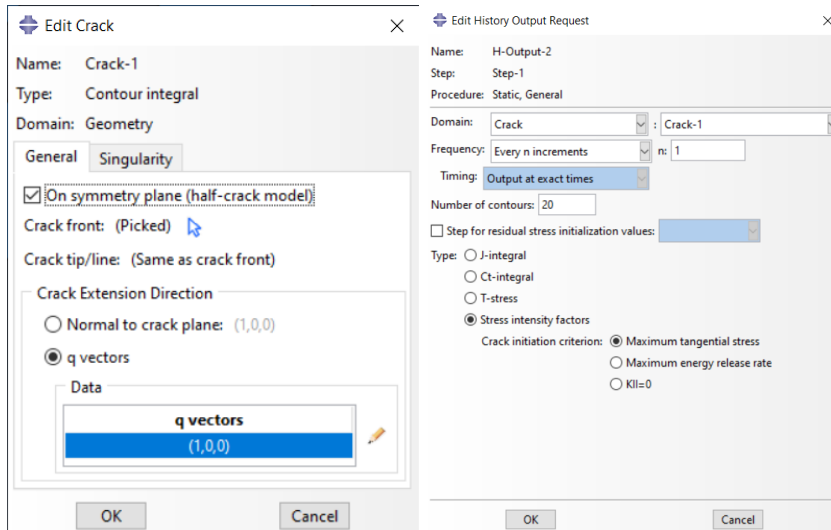
Boundary condition: set the line at the lower end of the model as the boundary condition (Ysymm option)



## Define the crack and history output request.

The crack is defined by Interaction > Special tab

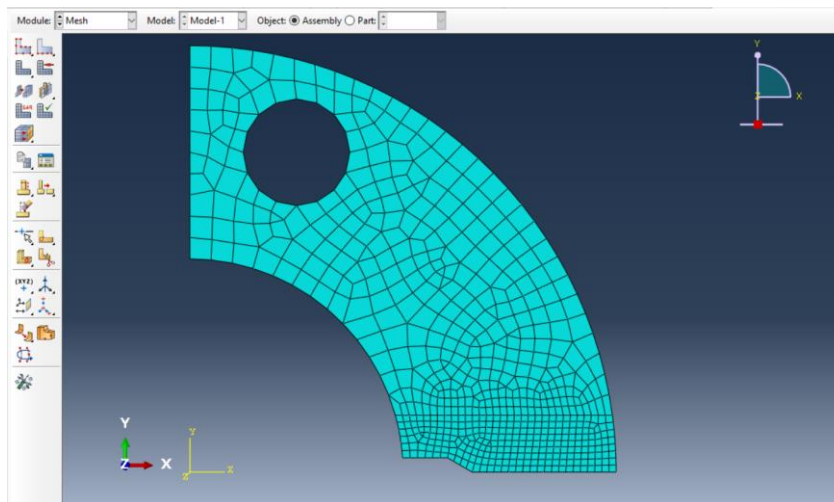
After definition of the crack, it will appear in Assembly > Engineering Features in the model tree. Then we proceed to add the history output request like below, which defines the number of contours for stress intensity factors. We choose number of contours = 20



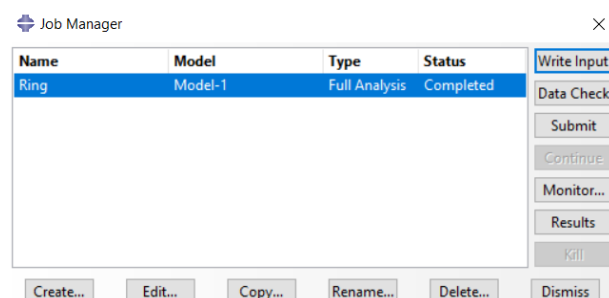
## Meshing the specimen

The top part above the line partition: seed edges of 0.003 (mm) for coarse mesh

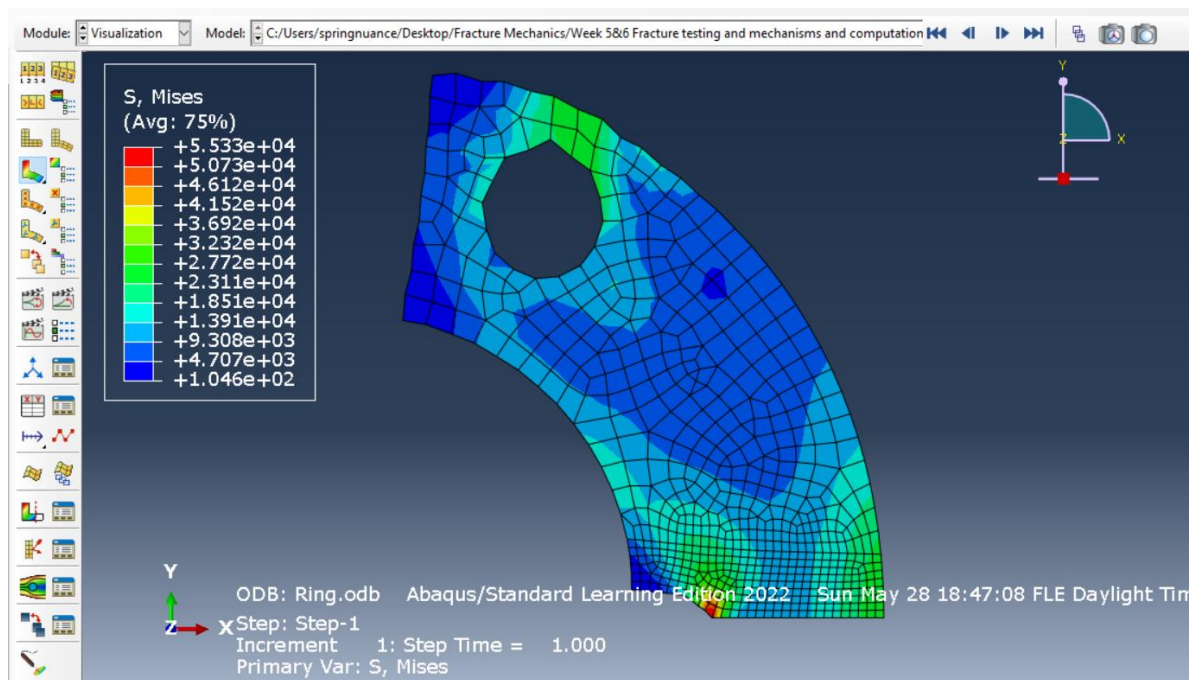
The below part under the line partition: seed edges of 0.001 (mm) for fine mesh



## Submitting the simulation job



## Result visualization, such as von Mises stress and strain



**Results** In this section, report how the prediction of the stress intensity factor  $K_I$  varies with the number of contours used to compute the contour integral. You can use a table, like the one shown in Table 1, or a figure to report your results. Based on your results, choose which contour will be used to make your prediction of the maximum load  $P_{max}$ . Present your calculation for  $P_{max}$ .

Table 1: Values of stress intensity factor  $K_I$  for selected number of contours used to compute the contour integral. All results are for an applied load  $P = 1\text{N}$

Number of contours	$K_I (Pa\sqrt{m})$
C01	3926.92
C02	5092.67
C03	5188.54
C04	5229.28
C05	5287.29
C06	5322.84
C07	5335.68
C08	5338.94
C09	5342.16
C10	5345.00
C11	5246.52
C12	5123.87
C13	5085.03
C14	5127.16
C15	5188.79
C16	5124.87
C17	5253.49
C18	5332.69
C19	5332.69
C20	4104.00

We can see the value of stress intensity factor fluctuates around 5250-5350  $Pa\sqrt{m}$ . I will choose the sixth contour as the stress intensity factor for testing

To find  $P_{\max}$ , we can try different magnitude of the loading such that the stress intensity factor at C06 reaches 1114850  $Pa\sqrt{m}$ . Since the LEFM is assumed, the stress intensity factor is linearly proportional to the loading magnitude. Therefore, we can run a few simulations to check the slope of the regression

P (N)	$K_I (Pa\sqrt{m})$ at C06
1N	5322.84
250N	1.33071E+06
500N	2.66142E+06
750N	3.99213E+06
1000N	5.32284E+06

```
# Linear regression between stress intensity factor KI and applied force P
P = np.array([1, 250, 500, 750, 1000])
KI = np.array([5322.84, 1.33071E+06, 2.66142E+06, 3.99213E+06, 5.32284E+06])
# fit linear regression
P = P.reshape(-1, 1)
KI = KI.reshape(-1, 1)
reg = LinearRegression()
reg.fit(KI, P)
# Slope
print(f"Slope of linear regression between P and KI: {reg.coef_[0][0]}")
```

Slope of linear regression between P and KI: 0.000187869633503919

```
# Predict Pmax at stress intensity factor KI = KIc = 1114850 Pa√(m)
KIc = 1114850
Pmax = reg.predict([[KIc]]).item()
print(f"Maximum force at fracture moment: Pmax = {round(Pmax, 2)} N")
```

Maximum force at fracture moment: Pmax = 209.45 N (ANSWER)

We can now plug in Pmax in Abaqus to verify if  $K_I$  is near  $K_{Ic} = 1114850 Pa\sqrt{m}$

Name: P209		
	X	Y
1	1	1.11487E+06

It is equal to the critical stress intensity factor, so we can conclude that  $P_{\max} = 209.45N$  which will cause the specimen to fracture

**Discussion** Compare your value of  $P_{max}$  to an analytical estimation based on the formulas in Appendix A6 of ASTM E399. Are both values close? what could explain the variation? Discuss the limitations of this FE model: is there anything that is present in reality but absent from the FE model? How could this FE model be improved?

Formula in the appendix

A6.5.3 Calculation of  $K_Q$ —Arc-shaped tension specimen  
 $K_Q$  is calculated in SI or inch-pound units of  $\text{Pa}\sqrt{\text{m}}$  ( $\text{psi}\sqrt{\text{in.}}$ ) as follows (see Note A6.3):

$$K_Q = \quad (\text{A6.4})$$

$$\frac{P}{B\sqrt{W}} \left( 3 \frac{X}{W} + 1.9 + 1.1 \frac{a}{W} \right) \left[ 1 + 0.25 \left( 1 - \frac{a}{W} \right)^2 \left( 1 - \frac{r_1}{r_2} \right) \right] \cdot f \left( \frac{a}{W} \right)$$

where:

$$f \left( \frac{a}{W} \right) = \frac{\sqrt{\frac{a}{W}}}{\left( 1 - \frac{a}{W} \right)^{3/2}} \left[ 3.74 - 6.30 \frac{a}{W} + 6.32 \left( \frac{a}{W} \right)^2 - 2.43 \left( \frac{a}{W} \right)^3 \right] \quad (\text{A6.5})$$

for which:

$P_Q$  = force as determined in 9.1.1, N (lbf),  
 $B$  = specimen thickness as determined in 8.2.1, m (in.),  
 $X$  = loading hole offset as determined in A6.4.1, m (in.),  
 $W$  = specimen width (depth) as determined in A6.4.1, m (in.),  
 $a$  = crack size as determined in 8.2.3 and A6.4.1.1, m (in.), and  
 $r_1/r_2$  = ratio of inner-to-outer radii as determined in A6.4.1.

```
# Arc-shape tension specimen
a = 0.01 # m
B = 0.01 # m
r1 = 0.03 # m
r2 = 0.06 # m
W = 0.03 # m
X = 0.015 # m

KIc = 1114850 # Pa√(m)
def f(a, W):
    aW = a / W
    firstTerm = np.sqrt(aW)/(1-aW)**(3/2)
    secondTerm = 3.74 - 6.30 * aW + 6.32 * aW**2 - 2.43 * aW **3
    return firstTerm * secondTerm

# Return KI of unit Pa√(m)
def KQ_equation(P, B, X, r1, r2, a, W):
    firstTerm = P / (B * np.sqrt(W))
    secondTerm = 3 * (X/W) + 1.9 + 1.1 * (a/W)
    thirdTerm = 1 + 0.25 * (1 - a/W) ** 2 * (1 - r1/r2)
    fourthTerm = f(a, W)
    return firstTerm * secondTerm * thirdTerm * fourthTerm
```

```
# Return KI of unit Pav(m), no other params except P
def KQ_equation_P(P):
    firstTerm = P / (B * np.sqrt(W))
    secondTerm = 3 * (X/W) + 1.9 + 1.1 * (a/W)
    thirdTerm = 1 + 0.25 * (1 - a/W) ** 2 * (1 - r1/r2)
    fourthTerm = f(a, W)
    return firstTerm * secondTerm * thirdTerm * fourthTerm - Kic
```

```
# KQ at Pmax = 209.45
KQ = KQ_equation(Pmax, B, X, r1, r2, a, W)
print(f"Stress intensity factor KQ at Pmax: {round(KQ)} Pav(m)")
```

Stress intensity factor KQ at Pmax: 1148519 Pav(m)

We can see the predicted Pmax produces  $K_Q$  very close to the critical stress intensity factor  $K_{Ic}$ . We can also reverse the function to find Pmax given  $K_Q$

```
sol = optimize.root_scalar(KQ_equation_P, bracket =
[0,1000], method='brentq')
print(f"Pmax given Kic is {round(sol.root,2)} N")
```

Pmax given Kic is 203.31 N

The analytical solution is close to the numerical one above, which is  $P_{max} = 209.45$ . Therefore, we can conclude that the FE model in Abaqus is quite reliable. However, the gap of 5N in Pmax prediction is also noticeable. Several factors can explain the variation between the analytical and numerical results:

1. Simplified assumptions: analytical estimations rely on theoretical models that may not fully capture all microstructure complexities present in the tested specimen.
2. Material Behavior: the FE model in Abaqus only features elasticity properties without any other specifications. In reality, there could be also thermality or conductivity
3. Mesh Discretization: The FE model's accuracy is influenced by the mesh seeds. If the mesh is not fine enough, it can lead to deviations from the true numerical result

Limitations of the FE model in Abaqus:

1. Mesh sensitivity: The accuracy of the results can be sensitive to the mesh density. Especially the student version's version is limited to only 1000 elements on the mesh. Additionally, only squares mesh can support stress intensity factor, so if we choose different mesh geometry, the FE model would not work
2. Difficult to work with fix mode scenarios: this FE model has  $K_I$  and  $K_{II}$  defined separately, but if we want mix mode, then it is harder to quantify the effect

How to improve the FE model:

1. Refine Mesh: Create denser mesh under the line partition near the boundary conditions to give better predictions for mode I stress intensity factors
2. Define more material properties, such as damage for elastomers, thermal and electrical conductivity. This will better accurately capture the behavior of the polymer