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.

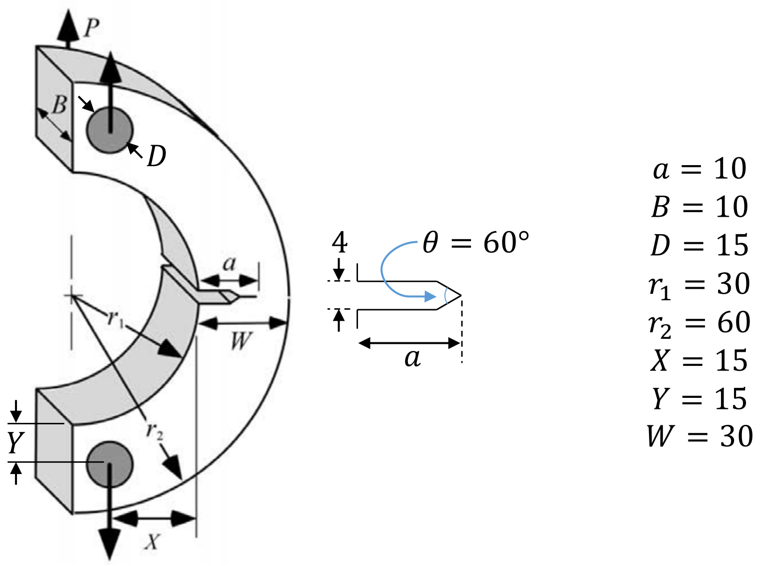

Figure 1: Geometry of the component. All dimensions are in mm.

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.

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).

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.

Then, to help you create your FE model, I have prepared a number of tutorial videos that will take you

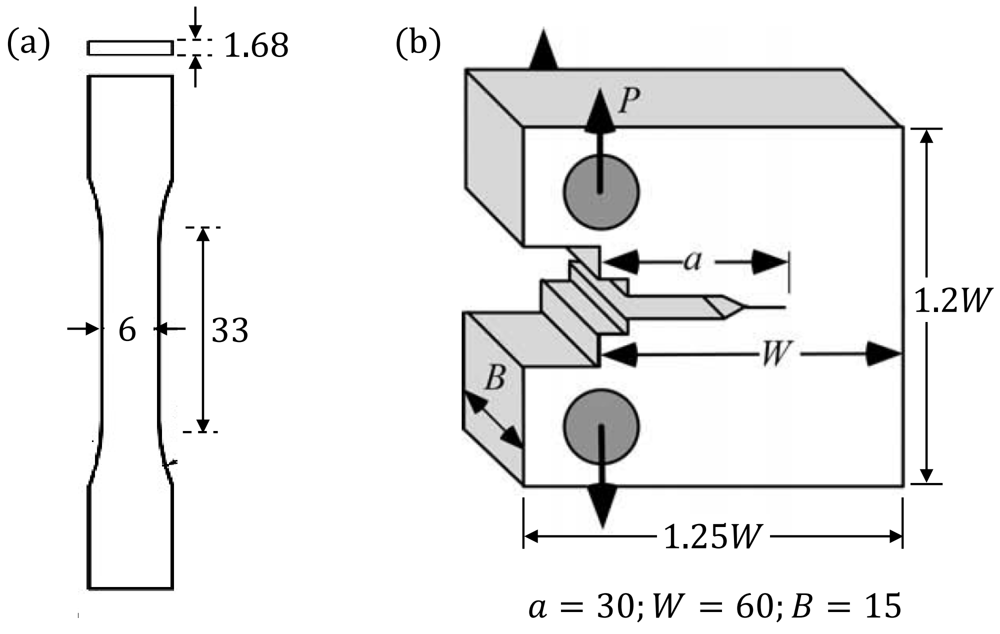

Figure 2: Specimens used to measure the material properties of the 3d printed polymer: (a) dogbone specimen for tensile tests and (b) CT sample for fracture toughness tests. All dimensions are in mm.

through each step of the process. These are available here: https://aalto.cloud.panopto.eu/Panopto/Pages/Sessions/List.aspx?folderID=f639915e-a855-49fa-b11e-aa4d00

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.

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}.

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?

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 $P = 1\,\mathrm{N}$.

Number of contours	K_I (Pa$\sqrt{\mathrm{m}}$)
n_1	K_1
n_2	K_2
n_3	$\dots$
$\dots$	$\dots$