

1.1 Plate under uniform radial tension

Given is the tensile test plate with a center hole for the numerical analysis of the limit states. The geometry, the material properties and the loading of the structure are given in figure 1.1. The elastic-plastic behavior of the structure shall be modeled and analysed with Abaqus.

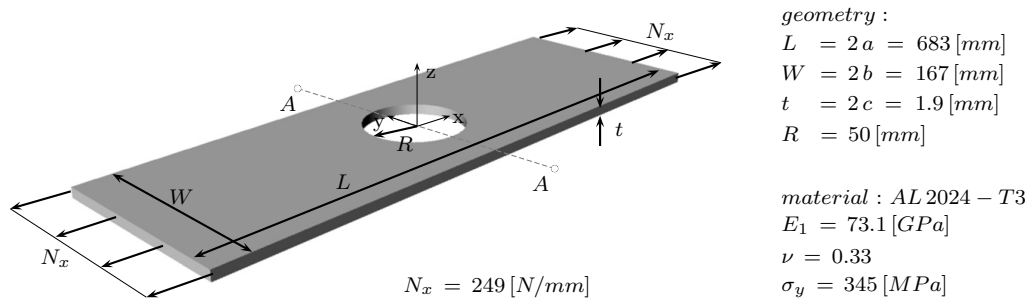


Figure 1.1: Tensile test plate model with a hole.

Solve the problem using the following steps:

1. model the problem in Abaqus exploiting the structural properties of the test plate.
 - choose two suitable element types based on different mathematical models for comparison of the numerical results of 2 to 5.
 - explain briefly your choice, the advantages of your choice and possible limitations.
 - give a precise summary of the model properties with regard to the chosen parameters, the boundary conditions and the applied yield criterion. Assume a bilinear isotropic hardening model with $E_2 = 0.1 E_1$.
2. how is the stress distribution along the cutting line A-A?
 - run your model for the given loading and various mesh densities.
 - show the convergence behaviour of your model under uniform mesh refinement.
 - show the stress distribution in the cutting line A-A.
3. show the stress distribution around the hole. Determine the stress concentration factor in the elastic regime from your results
4. show the stress-strain relation for a meaningful location and compare the two models of 1. Comment on the results.
5. apply the maximum load, elastically unload and determine the residual stresses in your model along A-A and around the hole.