Exercise no. 2 of the Finite Element Method I. course

Aim of the exercise

The exercise illustrates an important feature of the finite element method - the mesh convergence. Some ABAQUS mesh refinement techniques will be shown.

Analyse the stress-strain response of the part which is sketched below by using ABAQUS. Assume a 2D model with element size equal to 5 mm for the preliminary analysis and identify critical location based on the maximum equivalent stress. Then prepare at least two different versions of the model mesh with smaller elements and observe differences in the maximum stress. Use material with E = 210 000 MPa, ν = 0.3 and a single concentrated force varying in time as shown in the graph below. Consider the fixed time increment 0.25 to see how ABAQUS treats time-varying loads.