Exercise 3

November 22, 2016

1 Assignment

In engineering, the stresses are one of the prinicipal variables which helps to predict the failure of a mechanical component, defined by a specific shape and material, and subjected to specific boundary conditions. The yield stress of a given material is the stress at which that material begins to deform plastically. This material property determines the limit of performance for mechanical components. In structural engineering this is a soft failure mode which doesn't normally cause catastrophic failure, but it can be consider as a limit value for the designing of mechanical components.

Finite element analysis helps to predict the stress distribution of a mechanical piece under certain boundary conditions. This is of great help for the engineer during the design process. As the cost of performing a computer simulation of a given mechanical component is low compared with experiments, the engineer has the oportunity to redesign several times a specific component and evaluate its performance until all usage requeriments are fullfiled.

Let's consider a hook made of steel, subjected to Dirichlet (fixed end) and Neumann (load) boundary conditions. The geometry, material properties and boundary conditions are given by Fig. 1.

- Run this problem using Ostero and postprocess the results with ParaView.
- Using ParaView, look for the point of maximum stress (to do this, split horizontally the Layout, create a SpreadSheet View and sort the SIGXX and SIGYY variable).

- Regarding steel, its yield point is about 50 MPa. Considering the geometry and boundary conditions given, will this particular case behave under desired engineering limits?
- If the answer is yes, explain why.
- If the answer is no, as you are the mechanical designer, propose two solutions. Take into account that the boundary conditions can not be modified. Which solution is the cheapest?

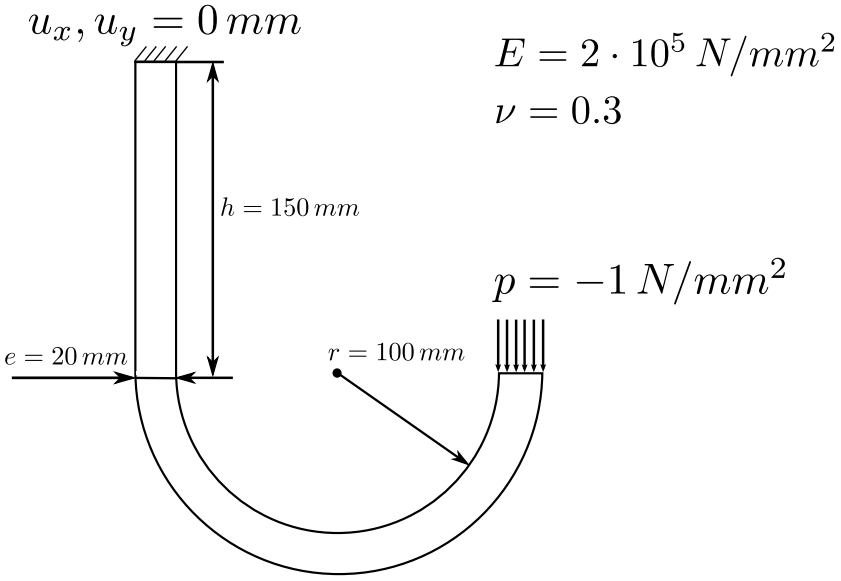

Figure 1: Geometry, material properties and boundary conditions.