Assignment in
The Finite Element Method
2010

Division of Solid Mechanics

The task is to write a finite element program and analyse various aspects of a structure. The CALFEM-package should be used and a number of additional subroutines have to be written.

Main task

The most common manufacturing technique when large quantities of a plastic parts is produced is injection moulding. Melted plastic is injected at high pressure into a frame made of steel with the density ρ and specific heat capacity c. In this case the frame consists of two identical tools, one upper and one lower half. The task is to analyze the temperature and stress distribution in such a tool. Sketches of the tool is shown in Fig. 1. In order to avoid leakage, a pressure P_{tool} is applied, squeezing the tools together. The surface that the pressure P_{tool} acts on and the surface that is in contact with the rigid structure have constant temperature T_g. The melted plastic is injected at the pressure $P_{plastic}$. The heat transferred from the melted plastic to the tool is approximated as a constant heat flow of magnitude q_n.
Both the temperature and stress problem is symmetric with respect to the x-axis. The tool is assumed to be long in the z-direction i.e. the problem can be approximated as a 2D problem. Horizontal movement of the tool is prevented since it is attached to a rigid structure at the right hand side (the hatched part in Fig. 1a), the surface in contact with the rigid structure have a constant temperature T_g . The surrounding air is assumed to have the temperature T_∞ and the heat transport between the tool and the surrounding air is due to convection (convection coefficient α_c). The convection take place on the boundaries marked with dashed lines in Fig. 1a. The tool material is assumed to be isotropic and linear elastic with Young's modulus E, Poisson's ratio ν, thermal expansion coefficient α, thermal conductivity k. The initial temperature of the tool is T_∞.

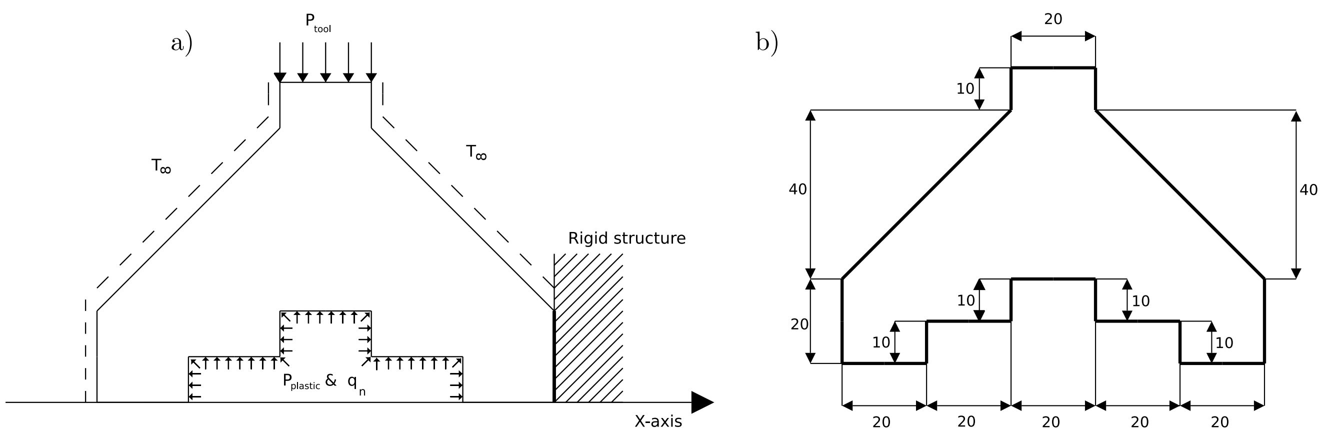

Figure 1: Sketches of the tool, the dimensions in b) is in mm.

The tool may be modeled using plane stress conditions. In Fig. 1 the dimensions of the geometry, thermal and mechanical loading are shown. It is highly recommended that you solve the computer lab before you start solving this assignment.

Problem formulation

a) Create and plot the geometry.

b) Determine how the temperature field varies during the molding process. The temperature field should be presented after 1 and 5 minutes using contour plots.

c) Determine the effective stress (von Mises) field after 5 min. The stress analysis should be presented for thermal load, mechanical load and combined thermal and mechanical load.

Procedure

The analysis should be performed in CALFEM. A suitable element is the linear triangular element. As a start point, you have the strong formulations. At least 50 elements should be used to perform the analyze. A well structured report (preferably not more than 6 pages excluding appendix) should returned to the Div. of Solid Mechanics no later than **2010-05-19 16.00**. The report should contain a description of the problem, as well as the solution procedure. The results from the calculations should be presented in form of illustrative figures and tables. The program code should be included as an Appendix and be generously commented and well structured. When writing the report it can be assumed that the reader has basic knowledge of the finite element method and solid mechanics. The reader can also be assumed to have knowledge of the problem description but it has been a while since he/she dealt with this type of analysis. After reading the report, the reader should be able to reproduce all the relevant code without using the included program. If the report contains errors, the report is returned in order to be corrected.

Collaboration

The task should be solved in groups of *two* or *individually*. For further details see www.solid.lth.se and navigate to the course page.

Useful hints

The nodal extrapolation of the von Mises effective stress can be done according to:

```
for i=1:size(coord,1)
   [c0,c1]=find(Edof(:,2:4)==i);
   Seff_nod(i,1)=sum(Seff_el(c0))/size(c0,1);
end
```

where *Seff_nod* and *Seff_el* is the von Mises effective stress at the nodal points and in the elements, respectively. *Edof* is the topology matrix related to the temperature problem.

Numerical data for main task

$E = 210\ GPa$	ν=0.3	α=1 $\cdot 10^{-5}\ 1/K$
k = 45 W/(m K)	α_c=500 W/m^2K	T_∞=25 ^oC
$P_{plastic}$=50 MPa	T_g=25 ^oC	P_{tool}=300 MPa
q_n=2$\cdot 10^5\ W/m^2$	$\rho = 7800 kg/m^3$	$c = 500 J/(kg\ K)$

Table 1: Numerical data