Assignment in
The Finte Element Method, 2015

Division of Solid Mechanics

The task is to develop a finite element program in order to analyze a complex geometry exposed to thermal and mechanical loads. This problem should be solved using Matlab and suitable subroutines included in CALFEM.

Main task

To protect space shuttles from the heat caused by friction during re-entry to the atmosphere, the surface of the shuttle is covered with ceramic heat tiles. To keep the temperature down in a new concept concept, the tiles are cooled by a circulating fluid. The inlet temperature of the cooling fluid is T_{in} and due to the heat exchange the outlet temperature is T_{out}. To facilitate the cooling the fluid is pressurized to p_w. The pressure from the atmosphere is denoted p_o. It is assumed that the heat flux due to the friction against the atmosphere is $q_n = q_o$. Heat transport at the bottom of the tile is caused by convection with convection coefficient α_c and surrounding temperature T_∞. Due to symmetry, the sides of the tile are insulated. The material in the tile is assumed to be isotropic with Young's modulus E, Poisson's ratio ν, thermal expansion coefficient α and thermal conductivity k.

To be able to perform the analysis in CALFEM, the structure is modeled using plane strain conditions. In Fig. 1 the dimensions of the geometry, thermal and mechanical boundary conditions are shown. The cooling water is indicated by gray color. The contour of the top boundary is given by

$$y = 100 + 50\sin^2\left(\frac{3\pi x}{250}\right)[mm] \tag{1}$$

Problem formulation

Determine the transient temperature evolution when the shuttle is entering the atmosphere. Assume that the initial temperature of the tile is T_0. Plots showing the temperature evolution should be presented.

Determine the stationary temperature distribution and the corresponding stress distribution of the heat tile. The results should be presented using plots based on suitable extrapolations to nodal points. The stress analysis should be presented for thermal load, mechanical load and combined thermal and mechanical load. The numerical data used to define the problem is given in Table 1.

Numerical data

Table 1: Numerical data

$E = 100$ GPa	$\nu = 0.3$	$\alpha = 1 \cdot 10^{-6}$ 1/C°
$\alpha_c = 5000$ J/sm^{2}C°	$k = 45$ W/mC°	T_0=100 $C°$
T_{out}=50	T_{in}=5 $C°$	T_∞=25 $C°$
q_0= -250 $\cdot 10^3$W/m^2	$p_0 = 25$ MPa	$p_w = 1$ MPa
$c = 450$ J/(KgK)	$\rho = 192$ Kg/m^3	

Procedure

A fully implicit time integration scheme should be used. Note that the element function for forming, $\boldsymbol{C}^e$, is available at the course homepage. A suitable element is the linear triangular element. As a starting point, you have the strong formulation of the heat and the mechanical problems.

The plots of the stress distribution is based on the stress at the nodal points. The extrapolation of the stress in the elements to the nodal points can be done by taking the mean value of the stresses in the elements connected to a node. The following Matlab code can be used:

```
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 associated with the temperature problem.

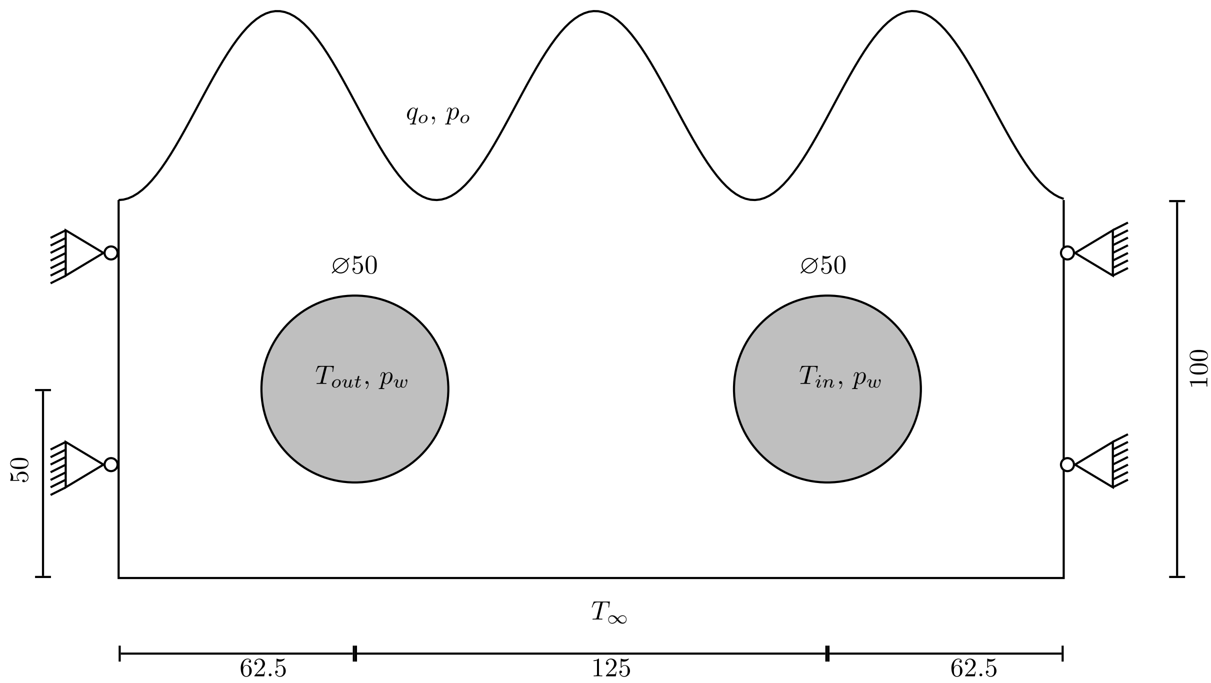

Figure 1: Cross section of heat tile (Dimensions in mm)

The geometry can be generated and meshed using the matlab toolbox *pdetool*. A suggestion for the procedure is the following. First mesh the heat tile as given in Fig. 1, but make the top boundary flat. A m-file can then be generated with PDE-tool. The m-file contains two vectors where the x and y coordinates of the vertices to the polygons in the mesh can be identified. The coordinates for the top boundary can then be inserted into these vectors to generate the top part of the mesh.

Report

A fundamental ingredient in all research is that it should be possible to regenerate the results obtained based on the report. In the present situation this implies that the appended matlab code should only be considered as supporting material. Moreover, note that one variable for grading the report is the structure of the computer code, i.e. you should choose suitable names for variables etc. A suitable structure for the report is:

- **Introduction**: Description of the problem, geometry etc. Keep this section as short as possible.
- **Procedure**: How the problems are solved (weak formulation, application of boundary conditions, thermal strains etc.). Note that you are encouraged to make references to text-books etc. It is important to carefully present all calculations that are not available in the litterature.
- **Results**: Present the results in illustrative figures and/or tables. Note that the results should be commented such that the reader can not missunderstand the results (correct labels, units, figure texts etc.)
- **Disscusion**: A disscusion of the results. You might want to discuss sources of errors and accuracy in this section.
- **Computer Code**: Note that the code should be easy to follow and all declared variables should have intuitive names and so on.

A well structured report briefly containing all steps from the strong formulation to the FE formulation is to be returned to the Div. of Solid Mechanics no later than **May 26 at 16.00**. The reader of the report is assumed to have the same knowledge level as the author. If the report contains theoretical errors, the report is returned in order to be corrected. It is possible to obtain up to 5 points which are augmented to the points obtained at the exam in May 2015. The assignment should be approved no later than 2015-06-12. You should submit your report in PDF format to FHLF01@solid.lth.se, FHLF10@solid.lth.se or FHL064@solid.lth.se. In addition to your report you should also attach your m-files in the email. Moreover, a paper version should also be handed in to the division of Solid Mechanics. Note that the bonus points obtained are only valid for the examination in May 2015.

Collaboration

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