Assignment in
The Finite Element Method FHLF01/FHLF10, 2018

Division of Solid Mechanics

The task is to write a finite element program and then use the program to analyse the thermo-mechanical behaviour of a heat exchanger during startup. The problem should be solved with Matlab along with suitable subroutines included in the CALFEM toolbox. The findings from the analysis should be presented in a well structured and well written report.

Problem description

In a plate heat exchanger, thin metal plates transfer heat between mediums (fluid or gas). The heat exchangers are constructed such that multiple plates are mounted in parallell with a thin channel in between each plate. Each medium then has an inlet and an outlet and is pressurized to flow through the enclosed channels (altering between every second plate) while heat is exchanged through the plates. The plates are geometrically designed to expose the mediums to a large surface area in order to optimize the efficiency of the heat exchanger. The complex design of the plates requires that engineers evaluate their thermo-mechanical performance in order to ensure that they can withstand operating conditions.

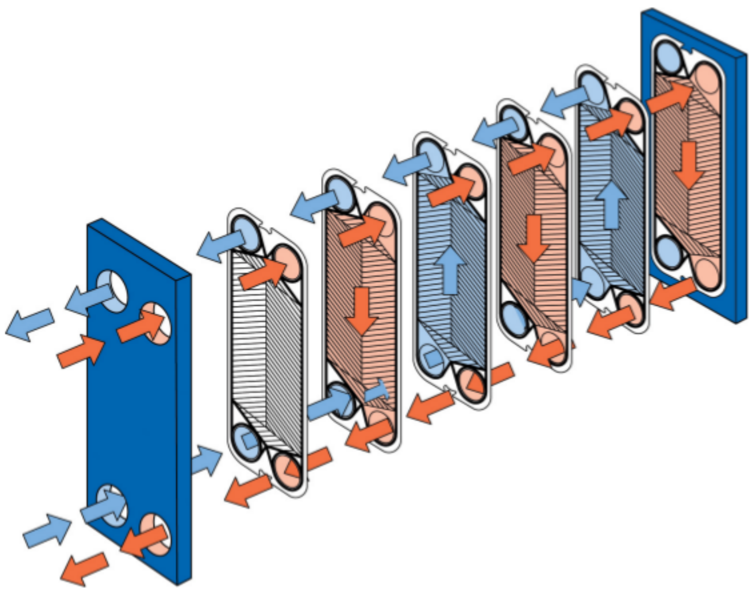

Figure 1: Sketch of a plate heat exchanger [1]. The arrows show the flow of the two different mediums.

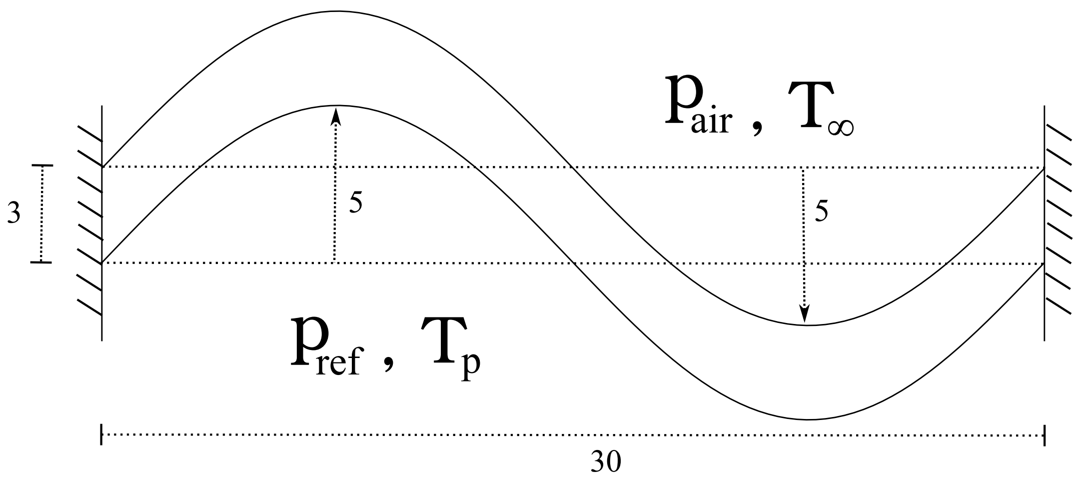

Figure 2: Sketch of the heat exchanger plate. (Dimensions in mm)

The assignment is to determine the heat flow and mechanical deformation of a plate within a heat exchanger during startup operation. To reduce the problem, only a small section of the original geometry is analyzed. A sketch of the geometrically simplified plate is presented in Fig. 2. The plate is shaped as a sinus curve of certain period and amplitude which can be calculated from the dimensions given in the sketch. Furthermore it is assumed that the left and the right side of the plate can be modelled as thermally insulated and clamped. The plate has an initial temperature denoted as T_0.

Air is flowing at the upper side of the plate, generating a heat flow q_n which can be modelled using Newton's convection law, i.e. $q_n = a_{air}(T - T_\infty)$. As the air flows it applies a pressure to the upper side of the plate denoted as p_{air}.

A regriferant is flowing at the lower side of the plate, the flow is assumed to generate a time-dependent temperature boundary condition during the start up operation. The temperature at the lower side of the plate is assumed to be

$$T_p(t) = \begin{cases} (T_{ref} - T_0) f(t) + T_0, & \text{if } t \leq t_p \\ T_{ref}, & \text{if } t > t_p \end{cases} \tag{1}$$

where T_0 is the initial temperature (same as the plate) and T_{ref} is the operating temperature after the start up time t_p. The monotonic function $f(t)$ describes the increase in temperature during start up and is given by

$$f(t) = \left(\frac{t}{t_p}\right)^3 \left(6\left(\frac{t}{t_p}\right)^2 - 15\frac{t}{t_p} + 10\right) \tag{2}$$

The flow of the refrigerant generates a pressure to the lower side of the plate given by p_{ref}.

Due to the heat generation, thermal stresses will be introduced in the structure. A linear elastic material behaviour, together with the assumption of plane strain condition is assumed. The plate is made of 316 stainless steel and all necessary material properties are presented in Tab. 1. All necessary information for the initial and boundary conditions are presented in Tab. 2.

Problem formulation

- Determine how the temperature distribution changes with respect to time and find the stationary temperature distribution. The temperature distribution should be presented as a contour plot at times $t = 2.5s$, $t = 5s$, $t = 7.5s$ as well at stationary conditions.

- Determine the effective von Mises stress field and the corresponding displacement field at stationary conditions. The von Mises stress field should be presented as a contour plot which clearly visualizes the stress gradients through the plate, this can be obtained by changing the maximum and minimum values of the colorbar in the contour plot.

Hint: The von Mises stress is defined as:

$$\sigma_{eff} = \sqrt{\sigma_{xx}^2 + \sigma_{yy}^2 + \sigma_{zz}^2 - \sigma_{xx}\sigma_{yy} - \sigma_{xx}\sigma_{zz} - \sigma_{yy}\sigma_{zz} + 3\tau_{xy}^2 + 3\tau_{xz}^2 + 3\tau_{yz}^2}$$

Procedure

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

The contour plots of the stress distribution are based on the stress at the nodal points. The stress of the nodal points can be approximated by taking the mean value of the stresses in the elements connected to a node. The following Matlab code can be useful:

```
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. Note that here the topology matrix (Edof) is associated with the temperature problem.

Numerical data

Young's modulus	$E = 193$ GPa
Poisson's ratio	$\nu = 0.25$
Density	$\rho = 7990$ Kgm^{-3}
Thermal conductivity	$k = 16.2$ Wm^{-1}K^{-1}
Specific heat capacity	$c_p = 500$ JKg^{-1}K^{-1}
Thermal expansion coefficient	$\alpha = 1.6$e-5 K^{-1}

Table 1: Material properties of 316 Stainless steel.

Surrounding temperature, air	$T_\infty = 25\ C^\circ$
Convection coefficient, air	$a_{air} = 1000$ Wm^{-2}K^{-1}
Pressure, air	$p_{air} = 10$ MPa
Pressure, refrigerant	$p_{ref} = 50$ MPa
Initial temperature, steel & refrigerant	$T_0 = 25\ C^\circ$
Operating temperature, refrigerant	$T_{ref} = 50\ C^\circ$
Start up time	$t_p = 10\ s$

Table 2: Initial and boundary conditions.

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 e.t.c. A suitable structure for the report is:

- **Introduction**: Description of the problem, geometry e.t.c. Keep this section as short as possible.
- **Procedure**: How the problems are solved (weak formulation, application of boundary conditions, thermal strains e.t.c.). Note that you are encouraged to make references to textbooks e.t.c. It is important to carefully present all calculations that are not available in the literature.
- **Results**: Present the results in illustrative figures and/or tables. Note that the results should be commented such that the reader can not misunderstand the results (correct labels, units, figure texts e.t.c.)
- **Discussion**: A discussion 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 containing all steps from the strong formulation to the FE formulation shall be returned to the Division of Solid Mechanics no later than **2018-05-25 16.00**. Matlab/CALFEM files (appendix) should be well commented. The reader of the report is assumed to have the same knowledge level as the author. If the report contains major programming or 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 2018. The assignment should be approved no later than 2018-06-11. You should submit your report in PDF format to FHLF01@solid.lth.se or FHLF10@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 is only valid for the examination in May 2018.

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 homepage.

References

[1] Henrik Forsbäck and Joel Johansson, *Simulation and testing of GPHE channel plates during assembling*, Master Dissertation, TFHF-5161, Lund University, 2011