Assignment in
The Finte Element Method, 2018

Division of Solid Mechanics
Ralf Denzer, Philip Oppermann, Simon Pettersson

The aim of the project is to analyze a rocket engine during firing, see Figure 1.

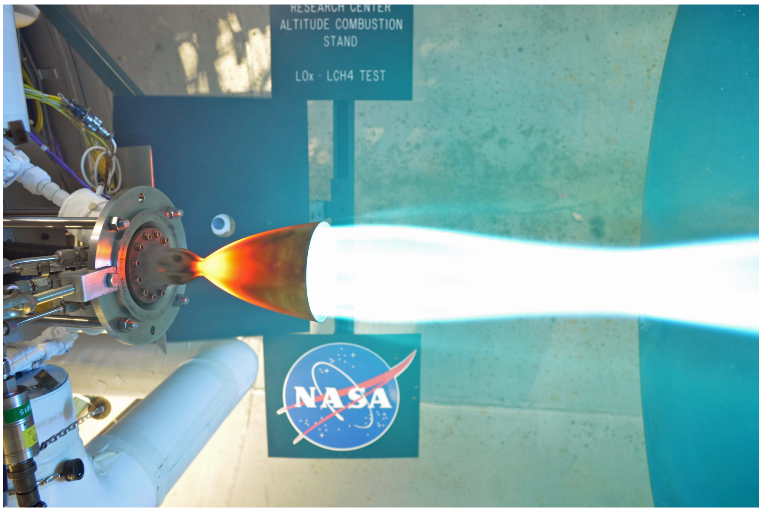

Figure 1: Firing Up Rocket Engine Test, Source: www.nasa.gov

The aim is to develop *two* finite element programs in order to analyze

- the rocket engine under thermal load from the starting of the engine.
- the rocket engine under mechanical load during firing.

These problems should be solved using Matlab. The mesh is already provided and linear triangular elements have been used. Thus, it is *not necessary* that you define your own mesh! You can use the Matlab program skeletons provided during the Computer Exercise sessions to solve the assignment. Do **not** use subroutines included in CALFEM.

Task 1 Rocket engine under thermal load

A strongly simplified version of the rocket engine is shown in Figure 2. We treat here only the two-dimensional cross section of the rocket engine with thickness $t = 1$ mm. We assume that the rocket engine is made of titanium.

Along the line F-G-H-I we assume convection boundary conditions with a convection coefficient α_1 and a temperature at infinity $T_{\infty 1} = 2000°\mathrm{C}$ which models the heat generated by fuel burning.

At the lines A-B-C-D-E-F and I-J-K-L-M-N we assume convection boundary conditions with a convection coefficient α_2 and a temperature at infinity $T_{\infty 2} = 20°\text{C}$ which models the cooling by surrounding air in the lab. At the boundary line A-N we assume a constant temperature of $T_{A-N} = -150°\text{C}$ due to the inflow of cooled fuel.

Determine the *transient* temperature evolution when firing up the rocket engine. Choose a time integration interval $t \in [0, T_{\text{end}}]$ such that the temperature field is quasi-stationary at $t = T_{\text{end}}$. Assume that the initial temperature of the rocket engine is $T_0 = 20°\text{C}$. A fully implicit time integration scheme shall be used. Note that the element function for forming the element heat capacity matrix $\boldsymbol{C}^e$ is available at the course homepage.

Hint: For simplicity start by solving the stationary heat problem with constant temperature on the convective boundary in order to check that the code is working. Later on you can implement the convection boundary conditions. As a last step the transient heat problem can be solved.

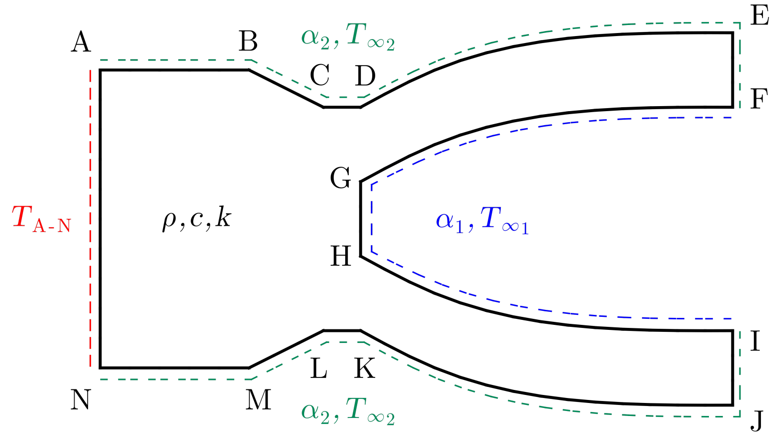

Figure 2: Sketch of the simplified rocket engine.

The following table gives numerical data

$\rho = 4540$ kg/m^3	mass density of titanium
$c = 520$ J/(kg $\cdot$ K)	mass specific heat capacity of titanium
$k = 22$ W/(m $\cdot$ K)	thermal conductivity of titanium
$\alpha_1 = 120$ W/(m$^2 \cdot$ K), $\alpha_2 = 20$ W/(m$^2 \cdot$ K)	convection coefficients

Task 2 Rocket engine under mechanical load

In this task we analyze the rocket engine under quasi-static mechanical loading as it is firing, see Figure 3. For simplicity we treat here only the two-dimensional cross section of the rocket engine with thickness $t = 1$ mm and assume a plane strain state. Thermal strains are not considered. The rocket engine is loaded by a constant pressure $p_{\max}$ along the line F-G-H-I stemming from the fuel burning. The line A-N is clamped. All other surfaces are assumed to be free of loads. Furthermore, the rocket engine is subjected to the gravity field $g = 9.81$ m/s^2.

The following table gives numerical data

Develop a finite element program in order to analyze the von Mises effective stress distribution in the rocket engine. Additionally, compute the resultant force at the clamped line A-N.

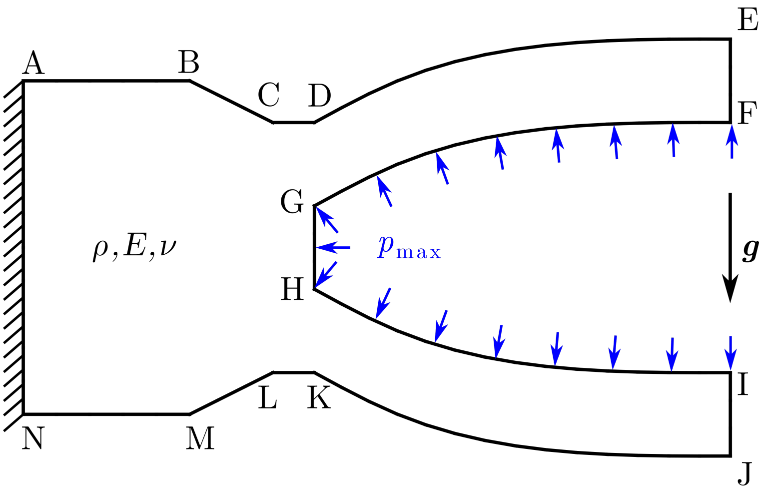

Figure 3: Mechanical conditions of the rocket engine.

$\rho = 4540$ kg/m^3	mass density of titanium
$E = 110$ GPa	Young's modulus of titanium
$\nu = 0.36$	Poisson's ratio of titanium
$p_{\max} = 10$ MPa	maximum pressure

Hint: The contour 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 node points and in the elements, respectively. *Edof* is the connectivity (or topology) matrix associated with a problem that has only one degree of freedom per node, like e.g. a temperature problem.

Report

A fundamental part 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 and comments of the computer code, i.e. you should choose suitable names for variables etc. A common 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 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 etc.)

- **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 briefly containing all steps from the strong formulation to the FE formulation is to be returned to the Division of Solid Mechanics no later than **May 22 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/June 2018. The assignment should be approved no later than 2018-06-12. You should submit your report in PDF format to FHLF20@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[1]. Note that the bonus points obtained are only valid for the examination in May/June 2018.

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.

[1]You find a box close to the printer of the Division where you can post your report.