Assignment in
The Finte Element Method, 2017

Division of Solid Mechanics
Ralf Denzer, Sally Issa, Philip Oppermann

The aim of the project is to analyze the re-entry of the Apollo Command Module in the atmosphere. A Command Module is a cabin where the crew of three are housed together with the equipment needed for re-entering the atmosphere and parachutes for landing on the water, see Figure 1.

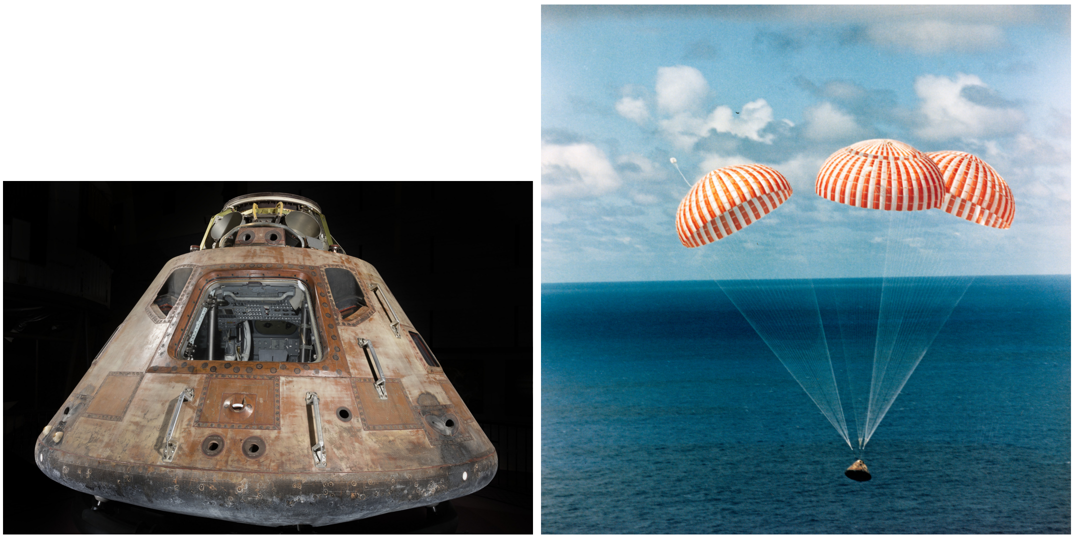

Figure 1: The Apollo Command Module (left). The Apollo Command Module landing on the water (right). Source: www.nasa.gov

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

- the Apollo Command Module under thermal load during re-entering in the atmosphere
- the Apollo Command Module under mechanical load as it parachutes close to the water surface.

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

Task 1 Apollo Command Module under thermal load

A strongly simplified version of the Apollo Command Module is shown in Figure 2. For simplicity we treat here only the two-dimensional cross section of the Command Module with thickness

$t = 1$ m. The Command Module is made of titanium.

On the line A-B we assume convection boundary conditions with a convection coefficient α_1 and a temperature at infinity $T_{1\infty} = 500°C$ which models the heat generated by friction in the atmosphere.

At the lines B-C, C-D and A-D we assume convection boundary conditions with a convection coefficient α_2 and a temperature at infinity $T_{2\infty} = -50°C$ which models the cooling by the cold atmosphere at high latitudes. At the inner boundary of the Command Module we assume a heat flux of zero.

In bottom of the Command Module we assume three cooling pipes with constant temperature $T_g = 100°C$ at the pipes' boundaries.

Determine the *transient* temperature evolution while the Command Module is entering the atmosphere. Assume that the initial temperature of the Command Module is $T_0 = -50°C$. 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, i.e. lines A-B, B-C, C-D, A-D, 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.

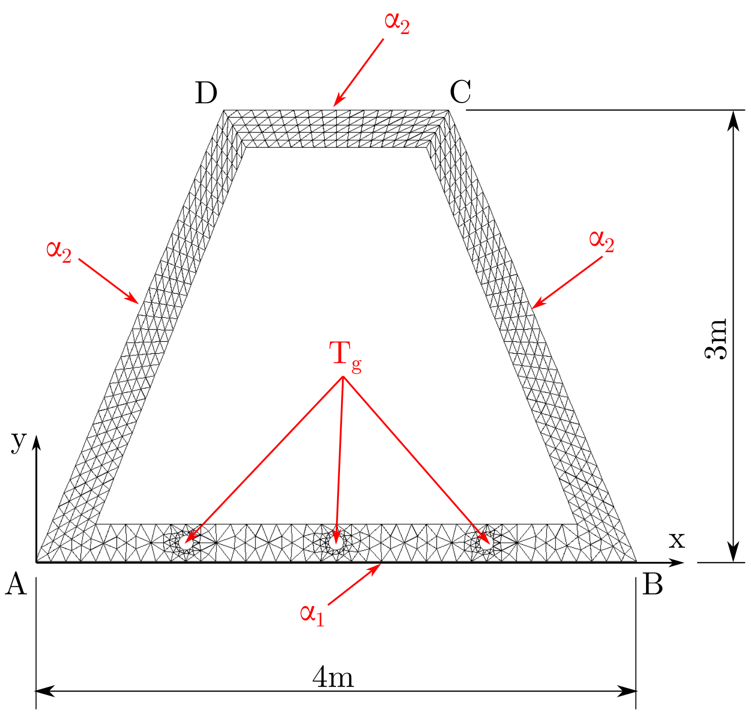

Figure 2: Thermal conditions of the Apollo Command Module

The following table gives numerical data

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

Task 2 Apollo Command Module under mechanical load

In this task we analyze the Command Module under quasi-static mechanical loading as it parachutes close to the water surface, see Figure 3. For simplicity we treat here only the two-dimensional cross section of the Command Module with thickness $t = 1$ m and assume a plane strain state. The Command Module is loaded by a parabolic pressure, see Fig. 2, stemming from the surrounding air. The maximal pressure in the center is measured as $p_{max} = 0.1$ MPa. It is assumed that the upper part, i.e. line C-D, is supported by roller supports while in point D we assume one fixed support. These mechanical boundary conditions should model the drag forces induced by the parachutes cables. All other surfaces are assume to be free of loads. Furthermore, the Command Module is loaded by gravity $g = 9.81$ m/s^2.

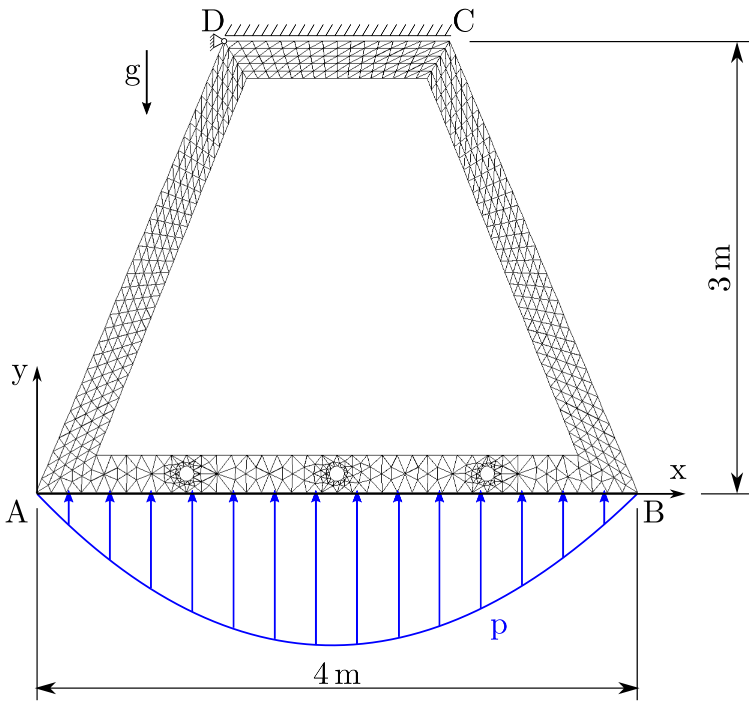

Figure 3: Mechanical conditions of the Apollo Command Module

The following table gives numerical data

$\rho = 4540$ kg/m^3	mass density of titanium
$E = 110$ GPa	Young's modulus of titanium
$\nu = 0.36$	Poisson's ratio of titanium
$p_{max} = 0.1$ MPa	maximum pressure

Develop a finite element program in order to analyze the von Mises effective stress distribution in the Module Command.

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 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 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 Division 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 2017. The assignment should be approved no later than 2017-06-12. You should submit your report in PDF format to 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 is only valid for the examination in May 2017.

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.