FEM in Multiphysics, Homework 1

Petra Reisz, 11816863

November 12, 2023

1 Create a suitable mesh for the problem

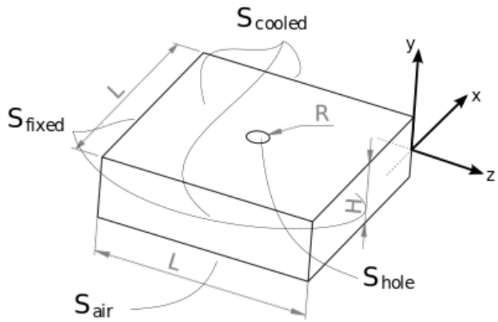


Figure 1: Setup of the geometry from assignment description.

1.1 Create hexahedral mesh and necessary regions, produce .cfs file

We wanted to create the setup as shown in 1. To create the according geometry, following steps were done in Coreform Cubit:

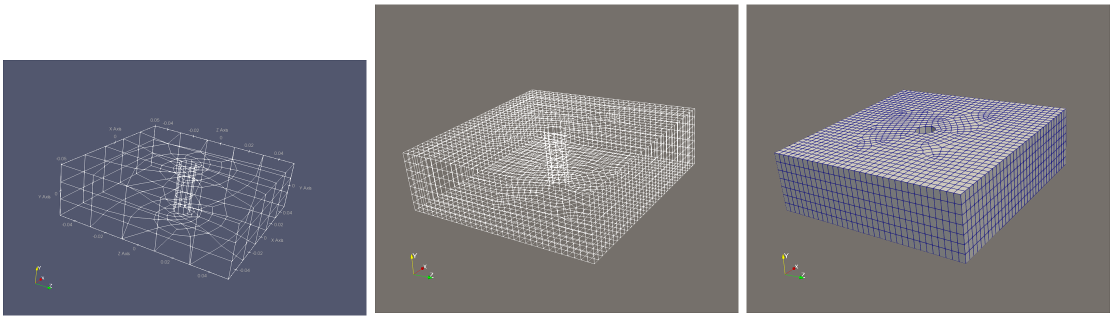
1. Creating a brick and a cylinder with the height/width/radius provided in the assignment description.
2. With boolean operation, the cylinder was subtracted from the brick, after it has been oriented accordingly (its circle has to be parallel to the x-z plane)
3. One assigns the surface names: *S_fixed_left, S_fixed_right, S_air, S_cooled, S_hole* as they are provided in Figure 1.
4. One creates the mesh. I choose an *adaptive*, textural mesh. The reason why I choose the adaptive mesh is that when solving PDEs with FE method, the regions with interior angle $> 180°$and strong 'sources' are considered as critical, and for this purpose adaptive meshing is recommended. I also set a maximum element size of 0.003 (that is 10% of H, intuitive value.) I did this because when leaving everything 'auto', the mesh was very coarse, see Figure 2).
5. One creates the block *'V_S235JR'* out of the regions.
6. One exports the file as *'geomHW1new.cdb'*
7. The cdb file can be converted to .cfs format by running: *cfs -g geometry* after modifying in the geometry.xml file the corresponding filename.

The Coreform Cubit file is also included in the report.

1.2 Paraview visualisation, nodes/elements

The visualized geometry and mesh can be seen in Figure 2. The number of nodes and elements can be extracted from the file *'geometry.info.xml'*. One has in this case:

- 10315 elements
- 9264 nodes



(a) Mesh, with adaptive meshing leaving all parameters auto

(b) Mesh, with adaptive meshing when setting ‘Maximum size‘=0.003 (WireFrame representation)

(c) Mesh, with adaptive meshing when setting ‘Maximum size‘=0.003 (surface representation)

Figure 2: Geometry and mesh

2 Modelling Assumptions

2.1 Modelling assumptions in the thermal model

1. Material: I assumed, a homogeneous,linear, material with isotropic and time -independent thermal behaviour, and set the parameters in *‘mat.xml‘* accordingly.

2. Analysis type: I assumed that the ‘excitation frequency‘ is a lot slower than the eigenfrequency of the system (for this one should actually do an eigenfrequency analysis, but we were provided the information that ‘drilling speed is low compared to the heat propagation‘. That justifies this assumption since the temperature has time to equilibrate until one drills deeper). For this reason, I choose analysis type ‘static‘.

3. Neumann boundary conditions: On the hole region we apply a pre-scribed heating power of $200W$, corresponding to a *Neumann BC* (one needs to calculate the flux by dividing this power with the area of the cylinder surface, when setting the boundary condition).

4. Robin boundary conditions: On the air- and water-cooled region we apply a pre-scribed bulk temperature and heat transfer coefficient, corresponding to a *Robin BC*.

5. Fourier law holds $\Longrightarrow$ no radiation, no convection

6. We have conservation of energy, described by the equation (strong form of PDE):

$$\rho c_m \frac{\partial T(x,t)}{\partial t} - \nabla \cdot (k \nabla T) = \dot{q}(x,t)$$

   where we neglect the first term with the time derivative, since we consider static analysis. T denotes the temperature in °C, ρ the density in kg/m^3, c_m the heat capacity of the considered material *S235JR* in J/K, k the heat conductivity in $W/(m \cdot K)$.

2.2 Assumptions in the mechanical model and for the coupling

1. Coupling: temperature is independent of the mechanical field (we can neglect strain heating since it would be only significant at high absolute temperatures, or for very fast stress variations). $\Longrightarrow$ Forward coupling of the temperature field on the displacement field due to thermal stresses. However, displacement field does not couple back.

2. We assume small strains, so we may use the linearised strain-displacement, strain-stress relationship. In solid mechanics analysis, this is the usual approach, expect dealing with extremely high loads leading to possibly non-linear behaviour of the material. Here it is not the case.

3. Material: linear, elastic, isotropic material behaviour, initially stress-free configuration, and define material parameters in *‘mat.xml‘* accordingly.

4. We can neglect traction between material and air/water.

5. We can apply static analysis, since we are interested in the equilibrium state of the of the initial geometry.
6. With the above assumptions one can derive the strong PDE: 6

$$-\nabla \cdot \sigma = g$$

with:

- $\sigma = C : s$ The stress tensor, and $s = \frac{1}{2}(\nabla u + \nabla u^T)$. C is the stiffness tensor, whose elements can be calculated from the Poisson number and E-modulus of the material.
- u denotes the displacement (vector quantity, in m)
- g denotes the volume force density, that is in our case (only considering thermal loads) $g = C : \kappa_{th} \Delta T$. Here κ_{th} denotes the thermal expansion (tensor) of the material S235JR in $1/K$. For the entries of κ_{th} one needs the specified 'Thermal expansion of S235JR' in the assignment description.

3 Drilling at normal operation

Obtain results by command ' cfs -p simulation.xml job '

3.1 Setup the simulation input.
This is specified in the file 'simulation.xml'. I wanted to obtain the following postprocessing results for the mechanical field:

- von Mises stress (element result)
- mechanical thermal stress (nodal result)
- mechanical thermal strain (nodal result)
- mechanical displacement (nodal result)

And for the thermal field:

- heat flux density (element result)
- Temperature (nodal result)
- heat flux on S_{air}, S_{cooled} (surface result)

3.2. Give the unit of the prescribed heat flux density q_s.
It is W/m^2.

3.3. Compute the heat flux density, which has to be prescribed at the source surface
We need to calculate

$$q_s = P/A_c$$

P is the produced heat power in W and A_c is the cylinder surface. $A_c = 2R\pi(H + R) = 0.0011m^2$. This yields for the heat flux density on the hole surface:

$$q_s = \frac{200W}{0.0011m^2} = 181818.1818W/m^2$$

3.4 - 3.5 Visualize the temperature distribution in Paraview (with only 10 colors in the palette) and the vector field of the heat flux density with a glyph-plot in the z-x plane
This can be seen in Figure 3 and 4.

3.6. Why is an output of heat flux density defined on elements and not at nodes? Can we compute a heat flux at a node on the reference element?
The heat flux is defined as (in the direction x_i):

$$q = -k\frac{\Delta T}{\Delta x_i}$$

We can also use instead of the difference operator Δ the differential operator. But for these cases, we either need a temperature difference in x_i (that can be calculated by the difference of neighbouring nodal temperature values on the element) or the spatial derivative of the temperature function (that is not not given, since we used Lagrangian finite elements that are C^0 continuous.) That is why we can only define heat flux density in the x, y, z-direction elementwise by taking the **difference** quotients of the nodal temperature values. We can also compute heat flux density on the reference element (coordinates η_i) by using the inverse transformation rule $\eta_i = \eta_i(x) \Longrightarrow q(\eta) = q(\eta(x))$.

3.7. What do you notice when looking at the flux vectors / what does the direction of the vectors tell us?
They show 'away' from the heat source, meaning, that heat flows from here in the direction to the boundaries of the brick (see Figure 4). On the surfaces S_{cooled}, S_{air} heat is transported away by air/water in order to keep the prescribed reference temperature of 20 °C.

3.8. What is the highest temperature, where does it occur? Is this temperature safe for the structure in terms of material transformations?
We can find and visualize these values (node values) in Paraview.

- Max temperature: 82.439 °C. The location is next the drilling on the air cooled (less cooling due to smaller heat transfer coefficient!) side. It absolutely makes sense to have the maximum temperature here, since on this side we have less cooling, plus we apply here the heating power (caused by drilling). It can be seen in Figure 3 on the left side. It is a safe temperature for steel, that has a melting temperature of couple of more that 1000 Celsius and a max. operating temperature of around 400 °C (`https://matmatch.com/materials/minfm33118-en-10025-2-grade-s235jr-as-rolled-condition-ar-`).
- Min temperature: 42.4626 °C. The location is at the edge on the water cooled (more cooling due to bigger heat transfer coefficient!) side. It absolutely makes sense to have the minimum temperature here, since on this side we have more cooling plus we are far away from heating power. It can be seen in Figure 3 on the right side.

3.9. How much heat is transferred due to the water cooling and how much due to the air? How much is this in percentage?
We divide by $200W$, what is the total heating power, to obtain the percentage:

- Transferred heat due to water: $150.227W \Longrightarrow \frac{150.227W}{200W} \cdot 100\% = 75.1135\%$
- Transferred heat due to air: $5.6552W \Longrightarrow \frac{5.6552W}{200W} \cdot 100\% = 2.8276\%$

The amount of transferred heat can be found in the output-history files *'jobname-heatFlux-surfRegion-S_air/cooled.hist'*

4 Drilling without water cooling

Obtain results by command ' cfs -p simulationNoWater.xml jobNoWater '

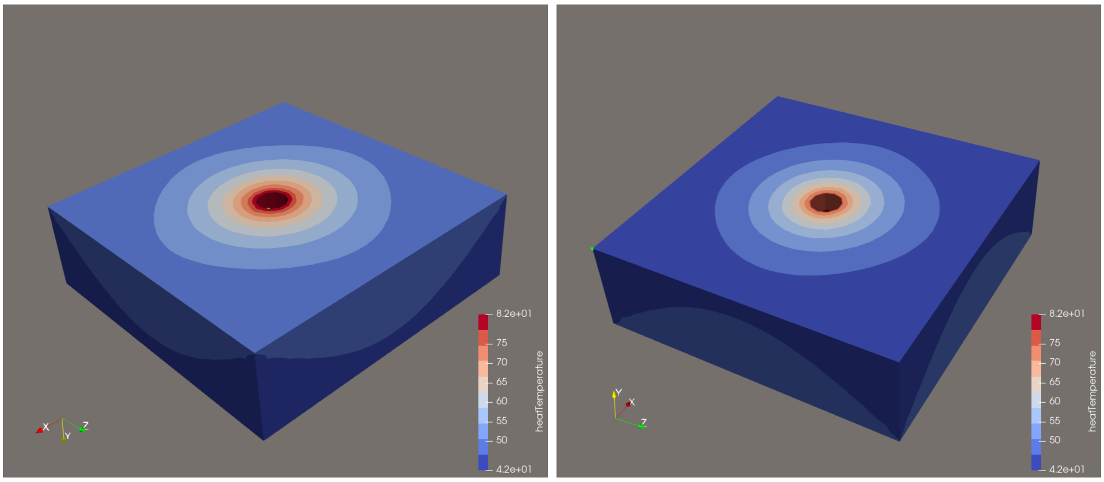
4.1 - 4.3. Which boundary condition do you need to change? What kind of BC do you set now? Visualize the temperature distribution in Paraview! What is the highest temperature / where does it occur? Is it safe in terms of material transformations?
We need to change the BC where we have heat transfer coefficient "600" to the $\alpha_{air} = 20W/(m^2K)$. By this we assume that the metal is covered from both sides by air (thereby we have no water cooling on the one side). The temperature distribution can be seen in Figure 5. The maximum nodal temperature is 476.814 °C and is located inside the drilled hole. If the temperature is the highest here, we expect the biggest thermal strain s_{th} (and thereby stress σ_{th}) as well in this region because:

$$s_{th} = \kappa_{th}(T - T_r)$$

.

$$\sigma_{th} = C : s_{th}$$

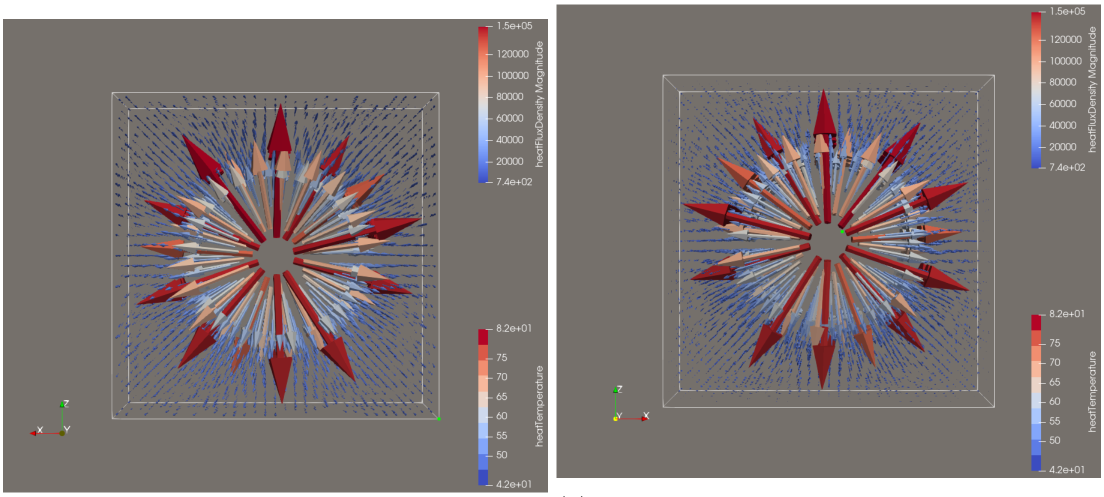
The above statement is justified by figure 6. It is not safe if a straight, undeformed form of the hole is required (high strain) also it leads to high stress in the inner region of the material that can be unsafe.



(a) Temperature distribution from the view of the air-cooled side and the maximum nodal temperature value indicated with green dot

(b) Temperature distribution from the view of the water-cooled side and the minimum nodal temperature value indicated with green dot

Figure 3: Temperature distribution



(a) Heat flux density from the view of the water-cooled side

(b) Heat flux density from the view of the air-cooled side and the minimum nodal temperature value indicated with green dot

Figure 4: Heat flux density

Besides, the max. service temperature of this material is prescribed as 391.1 °C (`https://matmatch.com/materials/minfm33118-en-10025-2-grade-s235jr-as-rolled-condition-ar-`). This means, this maximal temperature value is not safe anymore.

4.4. How much heat is now transfered due to the air? How much is this in percentage?
Now we have heat transfer on both sides by air: is $72.274W$ on the upper (previously water -cooled), and $72.275W$ on the lower side. That means

$$\frac{72.274W + 72.275W}{200W} \cdot 100\% = 72.27\%$$

of the heating power is transferred by air.

5 Heat-mechanic coupling

Obtain results by command ‘ cfs -p simulation.xml job ‘

I considered heat-mechanic coupling in the following way: on the same mesh, I did two analysis steps. In the first one, only the temperature distribution was calculated (this was discussed in the previous

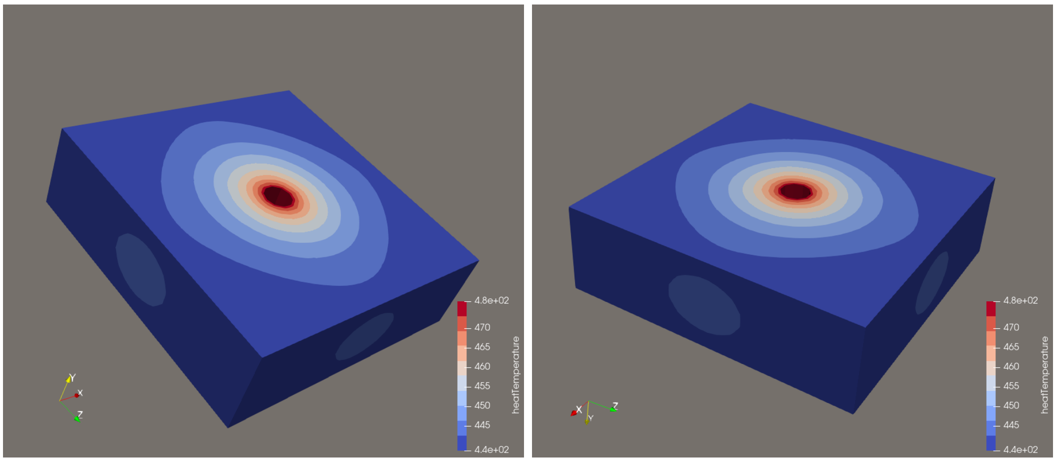


Figure 5: Temperature distribution from the 'up'- and 'down' side of the brick.(Symmetrc, both-sided air cooling)

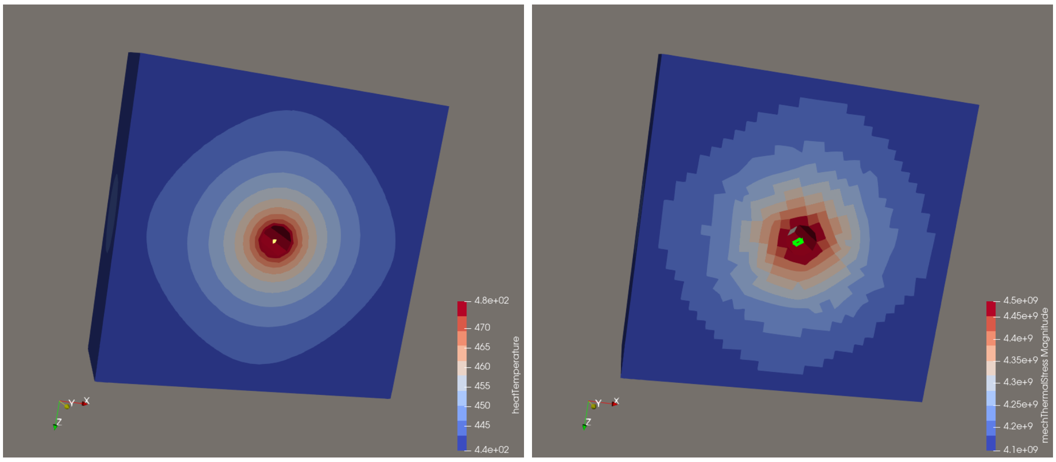


Figure 6: The node with maximum temperature is indicated with a yellow dot on the left. The element with maximum thermal stress is marked with green on the right.(Symmetrc, both-sided air cooling)

sections). In the analysis step 2, the mechanic field was calculated. This could be done due to the assumption that we only have forward coupling: the temperature field causes thermal strains and has hereby mechanic effects, but there is no/negligible effect of the mechanical field on the temperature distribution.

5.1. How shall the (mechanic) boundary condition be chosen in order to fit the description above?
We can implement the condition of clamped sides by choosing Dirichlet type boundary conditions on the surfaces S_{fixed_left} and S_{fixed_right}.

5.2-5.3 Visualize the deformed structure (mind the number of colours). What is the maximum deflection, and where does it occur?
The deformed structure can be seen in Figure 7.

The maximum displacement occurs on the non-cooled side, in the middle of the non-clamped edge. (higher Temperature$\Longrightarrow$ higher expansion/strain).

Max. Displacement (magnitude): $2.667 \cdot 10^{-5} m$

5.4-5.5 Visualize the von-Mises stress. What is the maximum von-Mises stress and where does it occur? Can it be a problem for the structure?
The von - Mises stresses can be seen in Figure 8. Its value is $2.383 \cdot 10^{8} Pa$. The *tensile strength* of the material is about $340MPa$. Tensile strength refers to the amount of load or stress that a material can handle until it stretches and breaks. In this consideration the maximal von Mises stress of $2.383 \cdot 10^{8} Pa$ is safe. However the yield strength is about $260MPa$. The yield strength is the stress point at which a material becomes permanently deformed, providing a useful approximation of that material's elastic

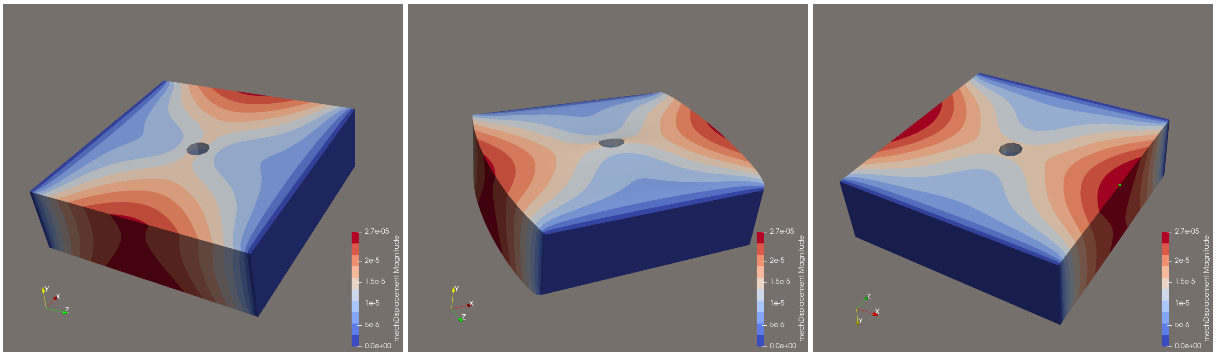


Figure 7: Magnitude of mechanical displacement (scale in m). Also scaled up to visualize how the deformation looks like (middle). The node with maximum magnitude of displacement is indicated with a green dot.

limit. Before reaching the yield point, the material will deform elastically, but it will always revert to its original shape upon removal of the applied stress. Once the yield point is exceeded, a small fraction of the deformation experienced will become permanent and irreversible. If the application requires that the material is completely undeformed, then, the value of $2.383 \cdot 10^8 Pa > 260MPa$ can be considered unsafe. The values tensile / yield strength and the definitions are taken from `https://matmatch.com/materials/minfm33118-en-10025-2-grade-s235jr-as-rolled-condition-ar-`.

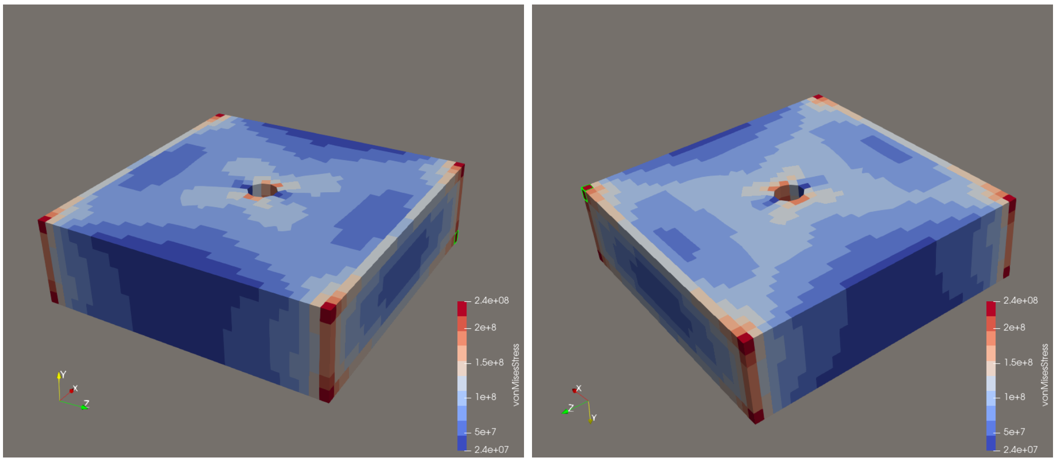


Figure 8: Von-Mises (scale in Pa) stresses on the water-cooled (left), air-cooled (right) side and the element experiencing the maximal von-Mises stress value indicated with a green marking.