1 Problem description

In the task we consider a laminar flow in the channel illustrated in Fig.1. The fluid enters the domain with the velocity $u_{\text{inlet}}(y)$ at the temperature $T_{\text{inlet}} = 300\text{K}$ and is heated up with several heaters equidistantly placed at the bottom surface. The heating is applied at the fixed temperature $T_{\text{heater}} = 400\text{K}$. The initial temperature in the domain is $T_{\text{initial}} = T_{\text{inlet}} = 300\text{K}$. The remaining parts of the channel walls are completely insulated ($q'' = 0$). The fluid is assumed to be Newtonian and incompressible.

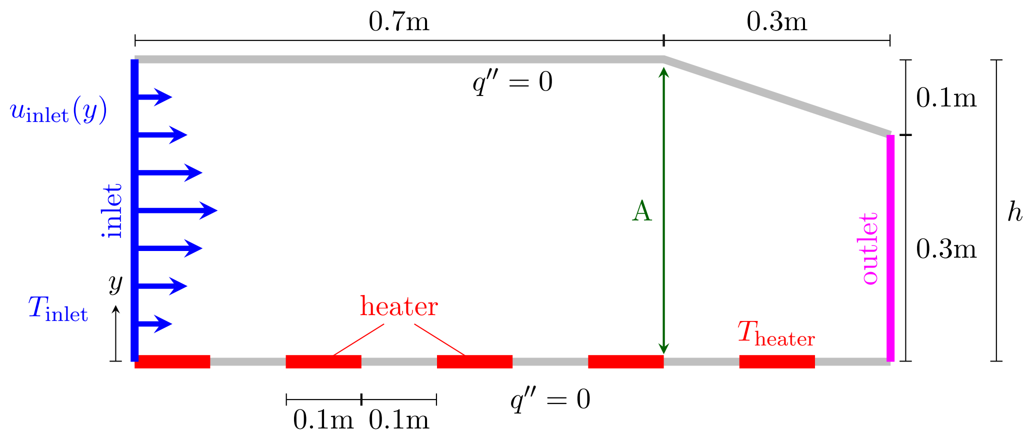

Figure 1: Schematic of the considered configuration.

2 Task description

In order to take into account all properties of the described flow configuration create a new simulation and implement the following modifications:

1. For the representation of the temperature field add the energy equation into pisoFoam solver:

$$\frac{\partial T}{\partial t} + \nabla \cdot (uT) = \nabla \cdot \left(\frac{\nu}{Pr} \nabla T \right) + \frac{(\nabla u : \tau)}{C_p}, \tag{1}$$

   with $Pr = 0.7$ and $C_p = 1000\ \text{m}^2\,\text{s}^{-2}\,\text{K}^{-1}$ and τ is defined as:

$$\tau = \nu(\nabla u + \nabla u^T) \tag{2}$$

2. Write a boundary condition for the bottom surface using `groovyBC` or `codestream`. This boundary condition applies fixed temperature at the heater surfaces and zero gradient temperature conditions at the walls in between.
   Hint: You can use the periodic function $f(x) = \sin(\pi x / L_{heater})$ to distinguish between wall surface and heater surface. If $f(x) > 0$ the location belongs to a heater, else it is a wall.

3. Apply the laminar inlet velocity with `foamNewBC`:

$$u_{\text{inlet}}(y) = U_0 \left(\frac{4y}{h} \left(1 - \frac{y}{h}\right) \right)^n, \tag{3}$$

   where $U_0 = 1.5\text{m}\,\text{s}^{-1}$, $n = 2$ and h is the channel height (see Fig. 1) are the input entries.

4. Write a new model for the description of viscosity dependency (`viscosityModels`) as:

$$\nu(x, y) = min(max(\nu_{min}, \nu_{ref} - (T(x, y) - T_{ref})\dot{\nu}), \nu_{max}), \tag{4}$$

   where $\nu_{ref} = 0.3\mathrm{m}^2\,\mathrm{s}^{-1}$, $\nu_{min} = 0.001\mathrm{m}^2\,\mathrm{s}^{-1}$,$\nu_{max} = 0.5\mathrm{m}^2\,\mathrm{s}^{-1}$, $\dot{\nu} = 1 \times 10^{-3}\mathrm{m}^2\,\mathrm{s}^{-1}\,\mathrm{K}^{-1}$ and $T_{ref} = 300\mathrm{K}$.

5. Write a run-time processing function using `swak4Foam`, which returns the pressure drop between the inlet and outlet. Moreover, computing the fluid bulk temperature at section A and plotting it over time. The bulk temperature is defined as:

$$T_b = \frac{\int_A Tu \ \mathrm{dA}}{\int_A u \ \mathrm{dA}} \tag{5}$$

Choose appropriate schemes and solver settings and run the simulation to reach the steady-state.

3 Oral examination

For the oral examination we ask you to prepare a presentation based on your implementation describing your solution for every particular subtask. The final examination (30 minutes) will start with this presentation followed by a discussion and question parts. Please note that we might interrupt you during your talk with questions. We expect a presentation duration of 10-20 minutes with 1-2 slides for each subtask. Please try to focus on the most important/difficult subtasks.

4 Submission details

For the submission please send following files to the course examiners (N. Samkhaniani or A. Stroh):

- the **source code** for your solver, boundary conditions and functions including the compilation script **Allwmake**,
- the working **simulation case** including the final solution (last time folder) and the run-script **Allrun**,
- the **plot** of bulk temperature over section A and the pressure loss as a function of time.
- your **presentation** in `.pdf`-format.

Please make sure that the submitted solution works out-of-the-box in OpenFOAM 8 and doesn't require any modifications from our side – especially avoid hardcoded paths in your code or configs. The examination date and time can be scheduled directly after submission.