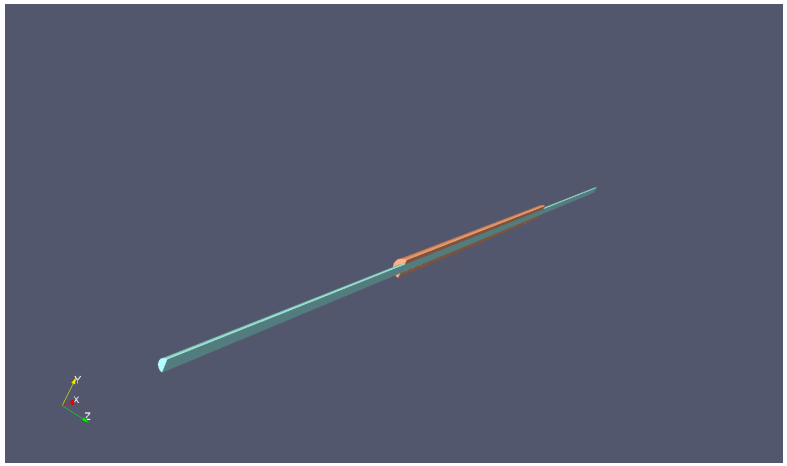
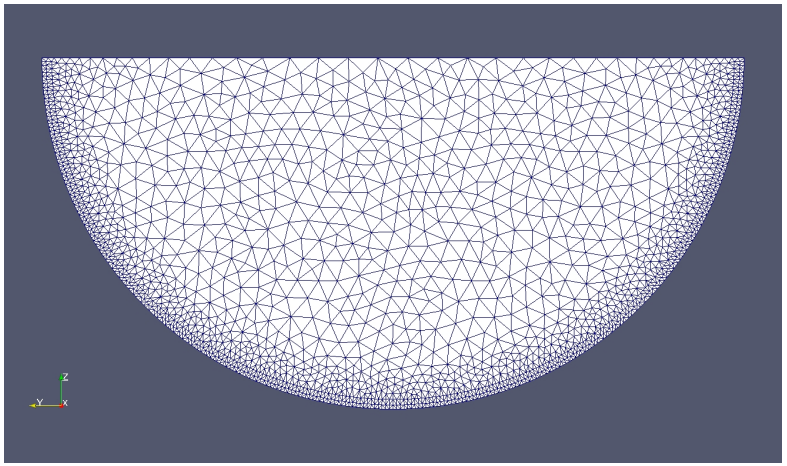
Introduction

A CFD simulation consisting of the fluid and solid region of a single tube cylindrical design has been carried out to understand the velocity, pressure and temperature profiles through the model.

The mesh consists of the fluid and solid regions of the model with a symmetry plane along the x-y plane. This allows for the mesh size to be halved and decreases the simulation time of the simulation. The mesh has 7 million cells generated by Pointwise mesh generation software.

The boundary conditions for this example are velocity inlet that corresponds to a Reynolds number of 200, an atmospheric pressure outlet and a fixed inlet temperature of 293K for the fluid region. The solid region has a boundary that is kept at a fixed temperature value of 375K.





Fluid/Solid properties and flow conditions

Region properties and flow conditions are given below. Temperature dependent functions of thermal conductivity/specific heat capacity can also be used if required in OpenFOAM versions 2.2.0 and above. No surface wall roughness is modelled.

	Fluid region	Solid region
Velocity inlet (m/s)	3.022	
Pressure outlet (N/m^2)	101325	
Fixed inlet temperature (K)	293	
Fixed boundary temperature (K)		375

	Fluid region	Solid region
Density (Kg/m^3)	Ideal gas law	8193.25
Viscocity (Ns)/m^2)	Sutherland law	
Thermal conductivity (W/mK)	Calculated	14
Specific heat capacity (J/KgK)	1006.43	435

In the model only a portion of the solid region is modelled since the remaining sections of the solid region are assumed to have no assumed heat transfer associated with them. This reduces the mesh size further.

Post-processing example

The plots below are of temperature and pressure along the model. These can be generated using a python script to take the average values over slices taken at 1.2mm intervals.

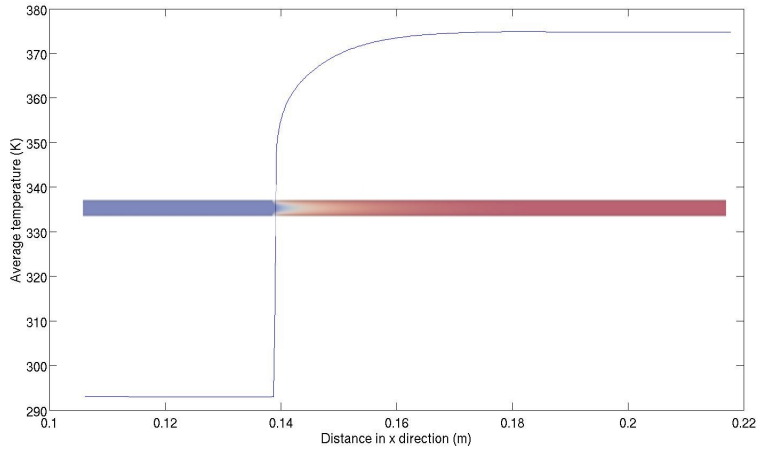


Figure 1: Plot of average temperature along x axis

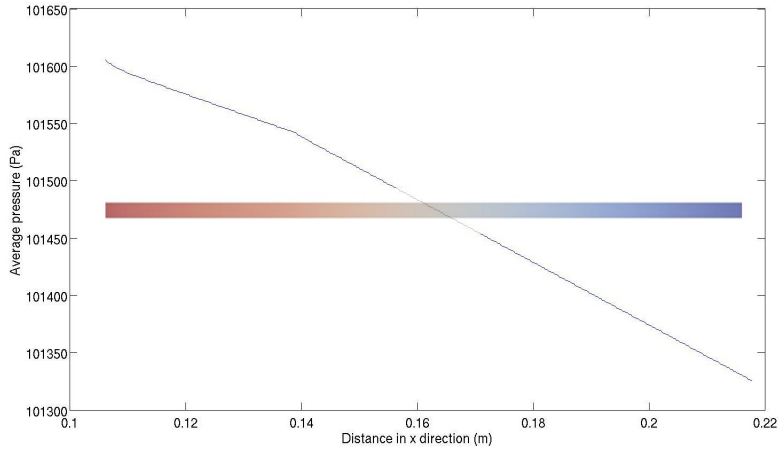


Figure 2: Plot of average pressure along x axis

For this initial run the behaviour of the model is as expected. At low Reynolds number the fluid is heated quickly and as it enters the heated section and the pressure drop is linear as expected.

Further work

Now we have a template CFD model we can apply this to the other single tube design models. Below are velocity profiles of two wavy design models subjected to the same mass flow rate inlet conditions. Once the simulation has converged we can run the same python script to obtain the same pressure drop and temperature data as before. Comparing the different model designs against each other with differing mass flow rates will give an interesting metric to performance.

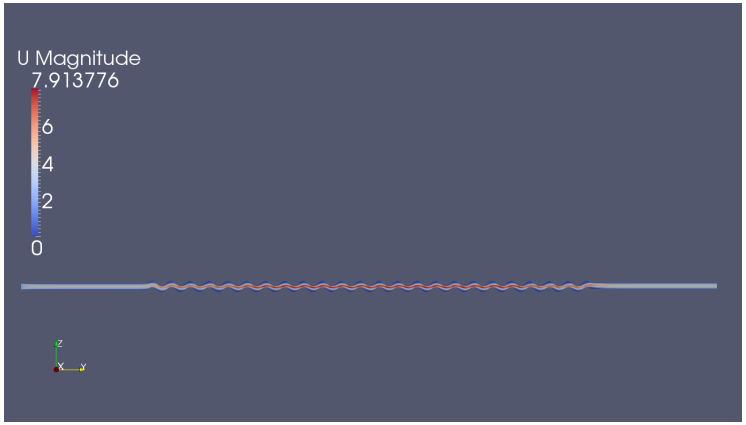


Figure 3: Velocity profile for wavy3 model.

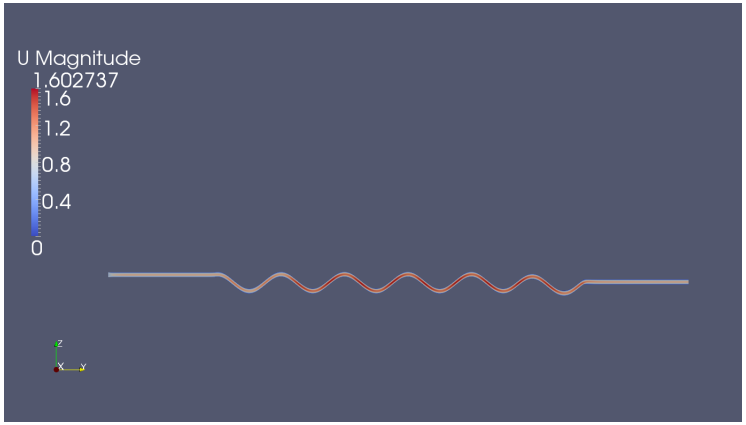


Figure 4: Velocity profile for wavy17 model.