

1 Theory

Computational Fluid Dynamics (CFD) is a powerful tool that allows flow problems which do not have a known analytical solution to be solved [1]. It is used to provide fast testing of new design concepts, gather detailed information of process conditions even for where measurements are difficult and to further our understanding of process performance [2]. However CFD is a dangerous tool as if used without a full understanding of the physical process which is occurring large errors can easily amount.

When using Solidworks® the following steps which must be performed when performing CFD computation:

- Specify fluid and thermodynamic properties
- Define computational domain
- Set mesh
- Apply Boundary conditions
- Specify goals

Within Solidworks® flow analysis the following conservation laws are solved:

- Conservation of momentum (Newton's 2nd law)
- Conservation of mass flow rate
- Conservation of energy

For a simulation to be performed and the above stated conservation laws to be solved, the simulation must be well poised, for this knowledge of the fluids viscosity μ , density ρ and thermal conductivity are required k .

When solving fluid flow problems, the area of interest - i.e. the area for which we want our study to be performed - is defined by the computational domain. This domain is split into many small 'sub-domains' - i.e. a mesh - within each of these subdomains the governing equations of the flow problem are discretised and solved. For fluid flow problems the finite volume numerical method is employed [3] and solves the problem for each of the many subdomains within the mesh, these solutions are then combined to provide a solution to the state of the fluid flow across the entire defined computational domain.

Solidworks® enables users to set properties of interest as 'goals', this then allows Solidworks® to ensure that the value of that property is of a suitable nature before the simulation is to be considered converged. Solidworks® also allows for users to manually monitor their simulations in real time, this is particularly useful when the simulations being performed are computationally taxing as it allows users to abort a simulation before potentially waiting long time periods to see that the goals have not progressed as expected and that the simulation is misbehaving.

2 Model I - "Pipe Flow"



Figure 1: Section view of fluid domain - all distance measurements in mm.

A 3D model was created within Solidworks® was created to simulate the flow of water at 353 K through a Steel pipe of surface roughness 0.045 mm, the pipe having an inlet pressure of 1 bar and an outlet pressure of 0.6 bar, as shown in figure 1

After the fluid had passed through the restrictor section of the pipe it undergoes an expansion, which leads to the occurrence of a recirculating region within the pipe **recirculating region** To determine the percentage

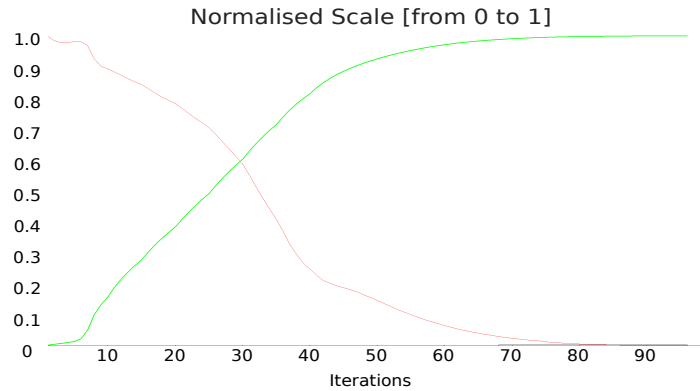


Figure 2: Normalised goals plot for Average Static Pressure (red), Average Velocity(X) (blue)

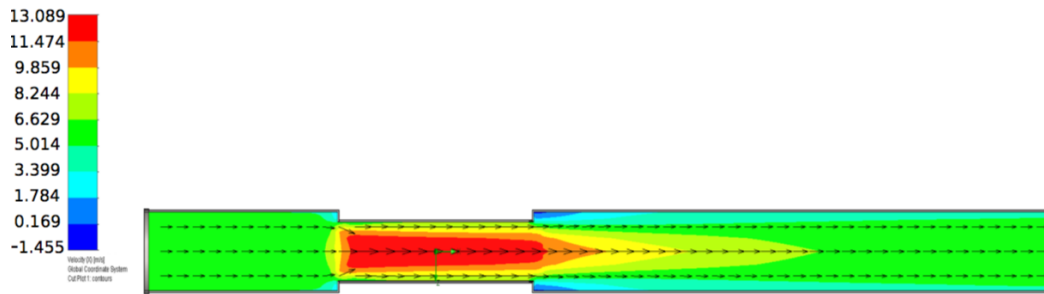


Figure 3: Cut plot showing contours of magnitude of the fluid velocity and vectors of fluid velocity

of the outlet pipe which the recirculating region occupies, an Isosurface plot was used to highlight the region of flow which had undergone a reversal in sign (direction). The length of this recirculating region was then measured and shown to approximately be 3.2 % of the outlet pipe.

The average static pressure at the end of the restrictor section was then determined by the placement of a 'lid' at the end of the restrictor section, the lid was then suppressed, simulation ran again and subsequently unsuppressed so that the static pressure at that region could be read off as a surface parameter on the lid - it was shown to be 41.9 kPa. Then to determine the mass flow rate through the pipe the surface parameter was now employed for mass flow rate, and it was shown to be 26.3 kg s^{-1}

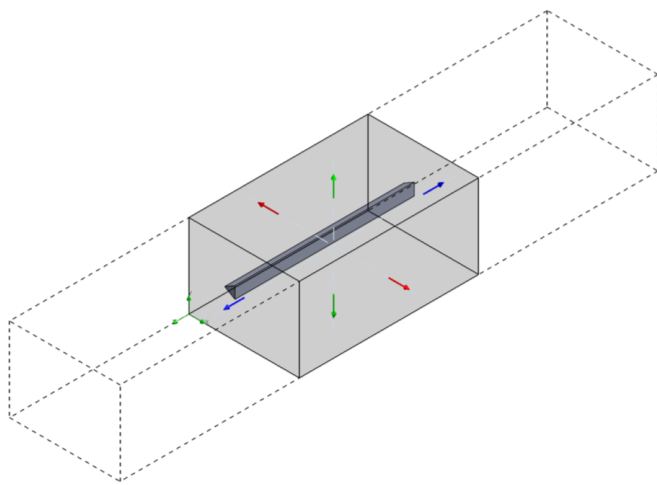
As is good practise when performing CFD, this obtained value from the simulation was to be verified through use of theoretical knowledge. Thus, the pipe was split into three different sections and the following equation used to define the head loss, h ,

$$\Delta h = \frac{\Delta P}{\rho g} = h_{Pipe1} + h_{Contraction1} + h_{Pipe2} + h_{Expansion3} + h_{Pipe3} \quad (1)$$

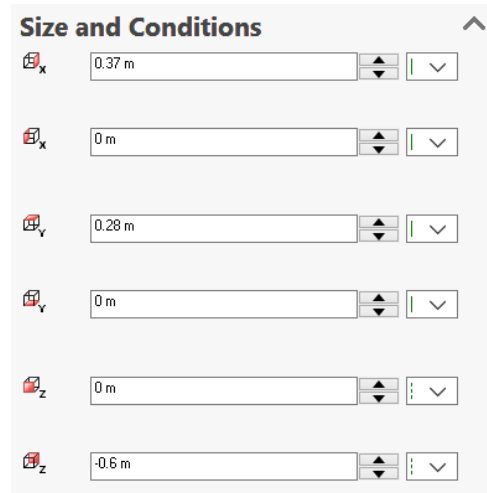
This head loss is then used to calculate the mass flow rate, via the volumetric flow rate. When using equation 1 the flow is assumed to be fully turbulent, thus the Moody diagram [6] is used to obtain the relevant friction factors for this problem, and then this assumption that the flow is fully turbulent is confirmed by calculating the Reynolds number, Re from the priorly obtained volumetric flow rate. The value of Re was shown to be 1800000, which is substantially greater than the threshold value for turbulent flow of 4000. Loss coefficients for $h_{Contraction1}$ and $h_{Expansion3}$ were found to be 0.125 and 0.191 respectively. The theoretical mass flow rate, \dot{m} , was found to be 33.9 kg s^{-1} , which is similar to value the value found through the simulation and thus implies that the simulation is trustworthy.

3 Model II - "Coolant Flow"

A further simulation was performed within Solidworks® this time simulating the external flow of real Carbon dioxide (CO_2) at 453 K around a Uranium equilateral prism of sides 40 mm and depth 600 mm at a temperature of 683 K, as can be seen housed within a specified computational domain in figure ?? on the next page. The CO_2



(a) Isometric view of the computational domain



(b) Computational domain dimension

Figure 4: Uranium Equilateral prism.

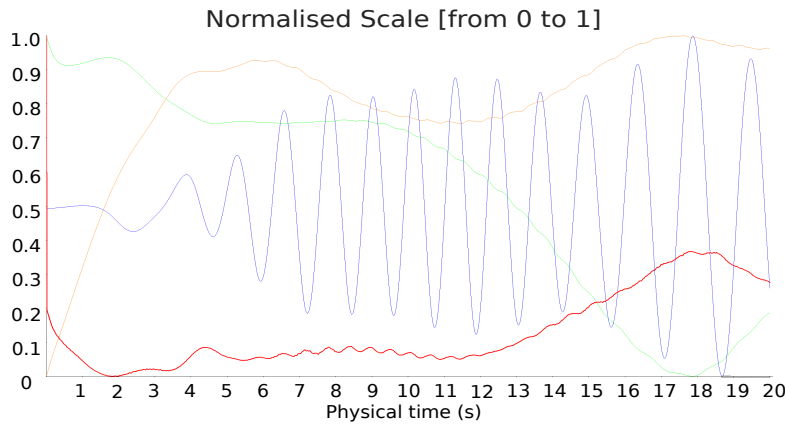


Figure 5: Normalised goals plot for Average Static Pressure (red), Average Velocity(X) (green), Average Velocity(Y) (blue) and Average Fluid Temperature (orange) with physical time

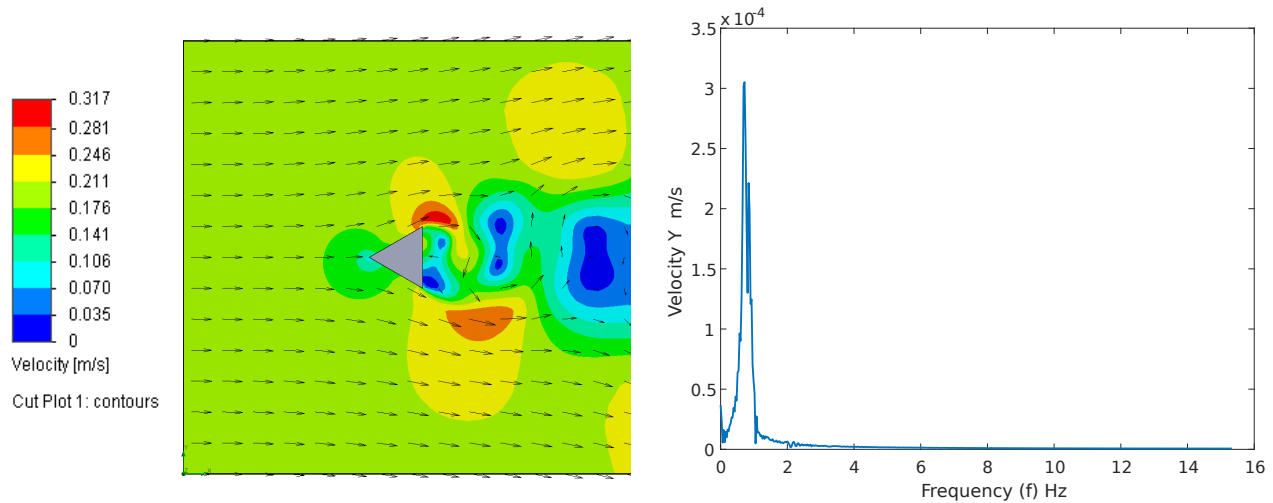
is at a pressure of 25 bar and a velocity in the x-direction of 0.2 ms^{-1} . For this simulation goals of Average Static Pressure, Average velocity (X), Average Velocity (Y) and Average Fluid Temperature were set, the calculation was set to run for 20 seconds physical time and the progression of these goals across the time period can be seen in figure 5.

Furthermore, the drag force acting on the uranium equilateral prism was determined use of surface parameters and shown to be 0.023 N . This value was then substituted into the following equation,

$$C_D = \frac{F_D}{\frac{1}{2}\rho U^2 DL} \quad (2)$$

In which F_D is the drag force which has been determined, U is the velocity of the fluid, D is the effective diameter of the uranium equilateral prism (0.04m), L is the length of the uranium equilateral prism and ρ which is the fluid density of CO_2 (29.25kgm^{-3}). From equation 2, the drag coefficient, C_D was found to be approximately 1.5, which is consistent with reference values for other triangular prisms [4], this means that the simulation may be trusted in its calculated value of drag force, F_D . Again, by using the surface parameters feature, the heat removal rate from the equilateral prism is found to be approximately 1.0kW .

As the Uranium equilateral prism is not streamlined, as the flow passes by it, it causes an oscillating flow to occur (as can be seen by the Velocity (Y) in figure ?? on page ??), this is referred to as 'Vortex Shedding' [5]. To determine the period of vortex shedding a fast Fourier transform (fft) was performed in MATLAB®, as can be seen in figure ?? on page ??.



(a) Cut plot showing contours of magnitude of fluid velocity and vectors over two-dimensional domain

(b) Fast Fourier transform (fft) of Velocity (Y)

Figure 6: Model II - "Coolant Flow"

The peak value occurs at a frequency of 0.7 Hz , thus meaning that the period of or vortex shedding is $\frac{1}{0.7} = 1.4 \text{ s}$.

To verify the obtained vortex shedding period the Strouhal number may be used. This is a dimensionless number which is a function of both the vortex shedding frequency, f , and the Reynolds number, Re .

$$S = 0.198 \left(q - \frac{19.7}{Re} \right) = \frac{fD}{U} \quad (3)$$

Through use of the above relationship the the theoretical and experimental value of the Strouhal number may be compared. Calculating the Reynolds number and substituting it's value into equation 3 yields a value of 0.20 for the Strouhal number - this is validated by [4]. Now, calculating the Strouhal number through use of the experimentally obtained vortex shedding frequency gives a value of 0.14, this is 30 smaller than the theoretical value. If more accurate results were to be obtained from this simulation, using a better - finer - mesh may be considered, or a more powerful compute which is able to run the simulation with a smaller time step and therefore fit perform more iterations within the 20 s time period.

4 References

- 1 - http://www.vttresearch.com/Documents/VTT_CFD_General.pdf
- 2 - <https://www.team-consulting.com/insights/%error-cfd-does-not-compute/>
- 3 - Solidworks Flow Simulation: Instructor guide, p5.
- 4 - Comparison of flow characteristics around an equilateral triangular cylinder via PIV and Large Eddy Simulation methods
- 5 - Sunden, B., 2011. Vortex shedding. Thermopedia.
- 6 - https://www.engineeringtoolbox.com/moody-diagram-d_618.html