Numerical study on the heat transfer characteristics of oscillating flow in Cryogenic regenerators

Swadesh Rath

(1) Introduction

In this project, the flow and heat transfer characteristics have been studied for an **oscillating flow regenerator**. An oscillating flow regenerator is a device (heat exchanger) which intermittently stores heat and then gives it back in the same periodic cycle. This can be seen extensively in the Stirling cycle, where the pressure variation can be modelled using a sinusoidal function.

The devices used today operate at frequencies ranging from 10 Hz to 120 Hz, but the analytical modelling of regenerators are very limited in literature. The important parameters we are interested in, is the study of drag coefficient for a steady flow simulation, which is dependent on the Reynolds number and the porosity of the wire packing. The Reynolds number for such mediums has been calculated using order of magnitude analysis. Further, Nusselt number and temperature profile has to be studied for a fully developed flow through the regenerator.

For this project, I have used OpenFoam to simulate the flow and heat Transfer. The *buoyantPimpleFoam* solver has been used to study the transient profile of the velocity and temperature, while the density changes with temperature have been neglected. The simulation is carried on for a cryogenic fluid, Liquid helium. The flow is assumed to be laminar for the experiment and *swak4Foam* has been used to set the user-defined functions at the inlet (for inlet velocity) and at the outlet (for outlet pressure).

The purpose of this endeavor is to study the several of Nusselt numbers at higher frequencies and also the trade-off between heat transfer and frictional losses as we increase the porosity of the medium.

Also, the paper talks about the cycle averaged Nusselt number comparison for each of the 9 unit blocks in the Mesh. Note, it is assumed that the flow develops in these 9 unit blocks.

Reference – Cryogenic journal 2018 – pages 99-107
*Authors - **Shuangtao Chen, Qian Huang, Menglin Liang, Houlei Chen, Liang Chen, Yu Hou***

(2) Domain

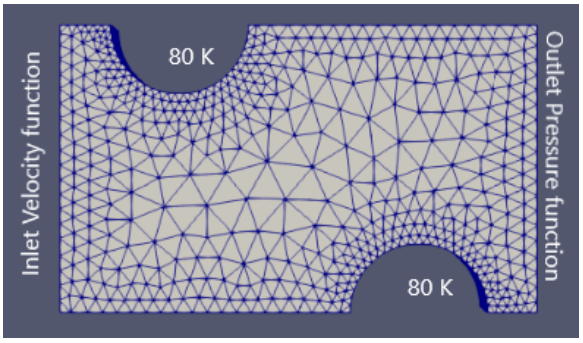


Fig 1 *– Unit block*

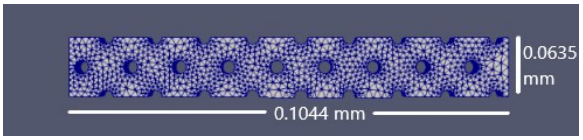


Fig 2 *– 9 such unit blocks*

The entire geometry has a dimension of 0.1044 mm × 0.0635 mm. For better results, the number of cells consider are 6214. Other meshes have also been taken into account, with the number of cells as 3407 and 1725 cells. Fluid enters with a user-defined function determining the velocity, which is sinusoidal with the simulation *runtime()*. Critical Reynolds number for the above porosity is around 107.

The major observation to take from here is the size of the dimension. For the courant number to be less than 1, moreover close to 0.7, the step time to be consider has to be very small. (**The inlet velocity has a maximum amplitude of 1.38 m/s**). The step time has to be small because of the small dx and dy values.

(3) Governing Equations

The equations we need to solve are –

Navier Stokes Equation

$$\frac{\partial u_j}{\partial x_j} = 0$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = \frac{-1}{\rho_f}\frac{dP}{dx_i} + \frac{\partial}{\partial x_j}\nu_f\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$$

Energy Conservation Equation

$$\rho_f C_{pf}\left(\frac{\partial}{\partial t} + \frac{\partial}{\partial x_j}u_j T\right) = \frac{\partial}{\partial x_j}\left(k_f\frac{\partial T}{\partial x_j}\right)$$

The PISO-SIMPLE algorithm has been used to solve the pressure-velocity linked NS equations. The initial conditions for the velocity field and pressure field –

(1) Inlet Velocity – User-Defined function as,

$$u = u_{max}\sin(2\pi f t)$$

Inlet pressure – *fixedFluxPressure* with a uniform 0 value.

(2) Outlet Velocity – *zeroGradient* assuming fully developed

Outlet Pressure – User-Defined function as,

$$p = p_{max}\sin(2\pi ft + 30^o)$$

The initial conditions for the Temperature field –

Inlet temperature is 100 K, and the wall Temperature is 80 K.

(4) Simulation approach

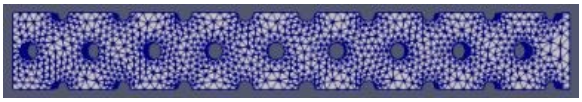
There are three folders in the case folders namely 0, constant and system. **0** folder consists the initial values U, p, p_rgh, T. In the **constant** folder the transport and thermophysical properties of the system have been provided, and finally we can decide the schemes and controlling parameters using the system folders.

4.1 Geometry and mesh

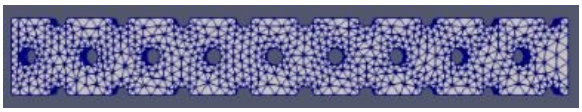
The geometry of the simulation has been discussed in the domain section of this paper. Again, we have to give emphasis to the small dimension of the domain. These leads to a very small cells in the domain.

The mesh has been created using SALOME 9.6.0. The geometry was specified in the software and then, the meshing has been done using Tetrahedralization Mesh tool.

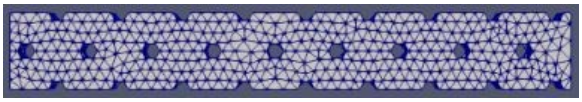
Mesh 1 - 6214 nodes



Mesh 2 - 3407

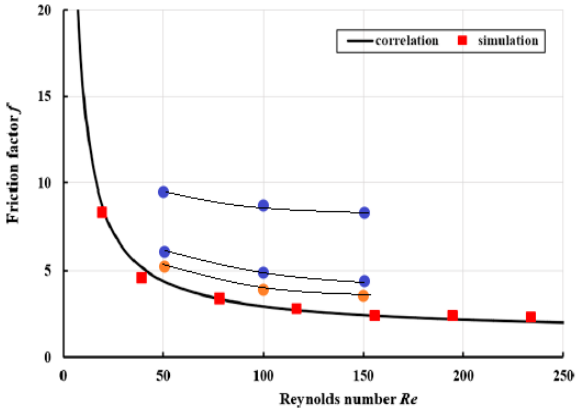


Mesh 3 - 1725



We can see that as we keep on decreasing the number of cells, the circular geometry is somewhat to maintain the tetrahedral profile. This is noticeable because the dimension of the geometry is too small for the software to mesh it properly. Each circular shape (representing a cylindrical wire's top view) has a diameter of 0.03 mm which is too small to mesh with smaller number of cells.

Grid Independence



Grid independence has been established by simulating a steady flow through the

geometry and comparing it with the steady flow correlations. Note, the orange dots represent the Mesh-1 and closest blue dots represent Mesh-2 and the farthest ones represent Mesh-3.

We can also observe that due to the distortion of the geometry of the circle in Mesh-3, the friction factor comes out to be relatively high.

4.2 Boundary Conditions

Velocity (U)

Inlet: groovyBC

Outlet: zeroGradient

Wall: no Slip

fluidEdge: Symmetry

face: Empty

Hydrostatic pressure (p_rgh)

Inlet: fixedFluxPressure

Outlet: groovyBC

Wall: FixedFluxPressure

fluidEdge: Symmetry

face: Empty

Total pressure (p)

Inlet: Calculated (Value uniform 0)

Outlet: Calculated (Value uniform 0)

Wall: Calculated (Value uniform 0)

fluidEdge: Symmetry

face: Empty

Velocity (U)

Inlet: 100 K

Outlet: zeroGradient

Wall: 80 K

fluidEdge: Symmetry

face: Empty

4.3 Solver

The solver I am using here is the buoyantPimpleFoam solver. It is a transient compressible solver for heatTransfer occurring because of a fluid flowing. The problem I have is a case of forced convection and thus, buoyantPimpleFoam is an appropriate solver.

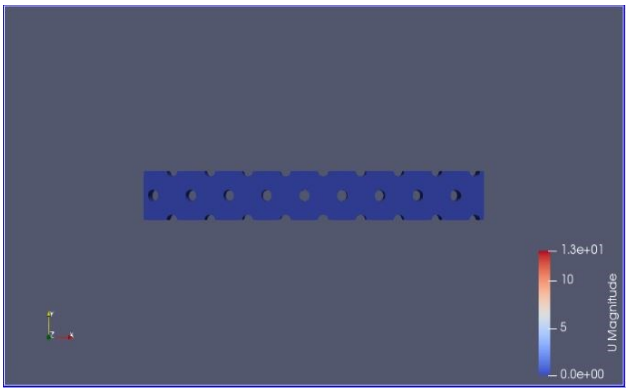
The pimpleFoam using the PISO-SIMPLE algorithm, which has a benefit of having larger time-steps to get the same accurate results.

The thermophysical properties have been modelled in a way such that the density of liquid helium remains constant. A *rhoConst* equation of state has been used for this simulation.

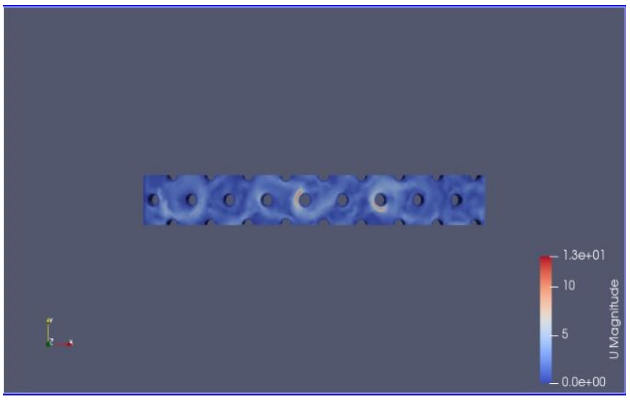
Note, swak4Foam's groovyBC has been used to model the user-defined functions for the inlet velocity and outlet pressure.

(5) Results and Discussion

Here we have the U-velocity –



(a) t = 0.0012 s



(b) t = 0.0024 s

(c) t = 0.0036 s

(d) t = 0.0048 s

Here we can see the U-velocity filling the medium, and also see the stagnation regions being formed at the back of the circular cylinders.

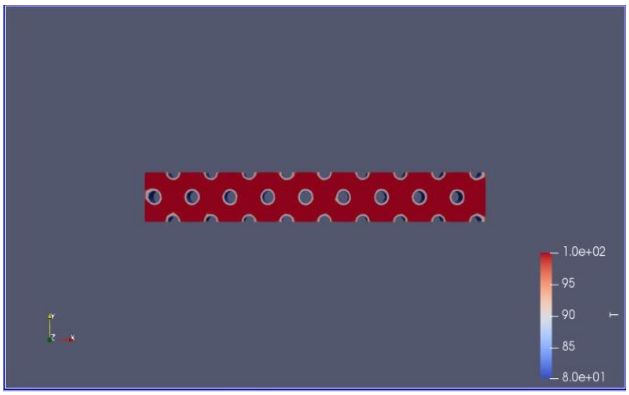
The maximum velocity was not reached by the inlet velocity, but the mass conservation leads to the maximum velocity of the flow to be around 13 m/s.

Since the time step is around 10^-8, the entire simulation could not be achieved and the total runtime of the simulation is around 0.005 s.

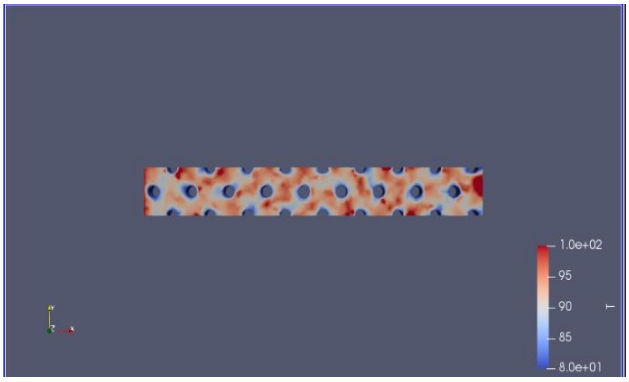
This is because the entire geometry is too small (less than 1 mm), while the velocity is around 1 m/s, these means the courant number condition restricts the step time to around 2.41 * 10^-8 s.

(5) Results and Discussion

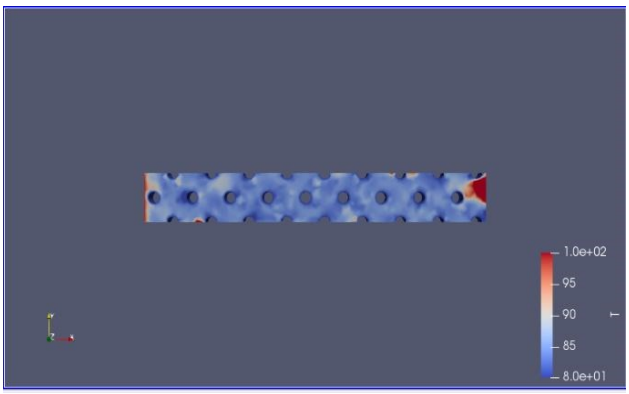
Here we have the Temperature –



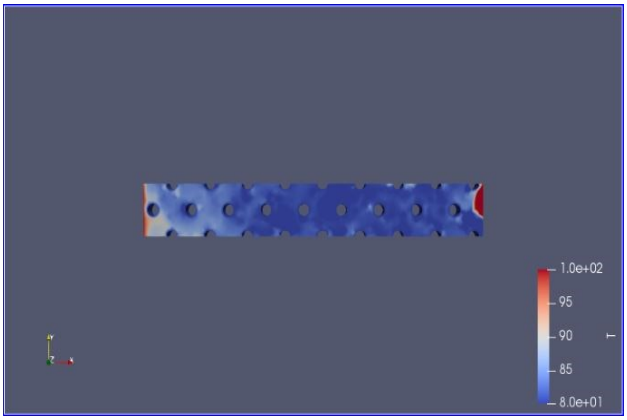
(a) t = 0.0012 s



(b) t = 0.0024 s



(c) t = 0.0036 s



(d) t = 0.0048 s

Here we can see that the initial temperature profile is 100 K, which cools down to about 80 K as the simulation starts. This is because the geometry is too small and the inlet mass flow rate has not yet become large enough to carry the heat towards the end of the pipe.

Again, the total simulation time was just 0.005 s, owing to the small geometry and thus, the courant number being less than 1, restricts the step time.

Note, it is necessary that the courant number remains below 1, or else the particles start skipping some of the cells due to their velocity.

To run the entire simulation, the computation expenses will increase substantially.

References –

(1) Numerical study on the heat transfer characteristics of oscillating flow in cryogenic regenerators
Shuangtao Chen, Qian Huang, Menglin Liang, et al, Cryogenics journal 2018

(2) Heat transfer characteristics of oscillating flow regenerators in cryogenic temperature range below 20 K

Yanyan Chen, Ercang Luo, et al, Cryogenics journal 2009

Reference – Cryogenic journal 2018 – pages 99-107
*Authors - **Shuangtao Chen, Qian Huang, Menglin Liang, Houlei Chen, Liang Chen, Yu Hou***