CFD analysis of fluid flow and heat transfer in a shell and tube heat exchange in OpenFOAM.

Aabhushan Regmi
FOSEE Team, IIT Bombay

Synopsis

In this study chtMutliRegionSimpleFoam is used for the numerical simulation of a shell and tube type heat exchanger. The structure of such type of heat exchanger consists of large pressure vessel inside which is a bundle of tubes carrying fluid, heat transfer takes place through the surfaces of these smaller tubes. This is a case of conjugate heat transfer in multi-region (between two fluids through a solid in between). Turbulent and steady state study is done. OpenFOAM-v2012 is used for this study. The profiles of velocity and temperature at the end of the flow is obtained from the simulation and validated against the value from a published paper.
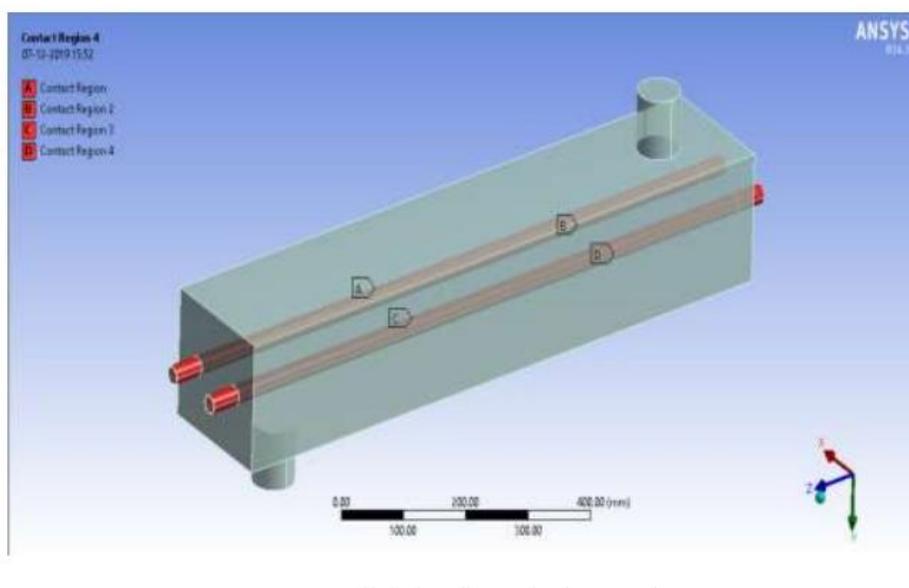
Figure 1: Geometry being Considered

References

CFD ANALYSIS OF HEAT EXCHANGER MODELS DESIGN USING ANSYS FLUENT, Ram Kishan, Devendra Singh, Ajay Kumar Sharma*

1 Introduction

A shell and tube heat exchanger consists of a series of tubes housed within a cylindrical container known as a 'shell'. All tubes within the shell are collectively termed a 'tube bundle' or 'tube nest'). One fluid runs through the tubes, and another fluid flows over the tubes (through the shell) to transfer heat between the two fluids. One of the tube sheets is fixed and one is free to move, this allows for thermal expansion as the heat exchanger is heated. They have a simple design, robust characteristics and relatively low purchase and maintenance costs. They also have a very high heat transfer rate although they require more space than a plate heat exchanger of similar thermal exchange capacity.

The case study demonstrates the use of OpenFOAMv2012 software to understand conjugate heat transfer in three-dimensional flow. In this simulation, finite volume method is used for incompressible, steady state and turbulent flow. Below are the steps that has been carried out during the study of this case study.

- Setup a problem case using PlaneWall2D case
- Creating a 3D mesh by using blockMesh, topoSet, createPatch and splitMesh
- Set up the properties of the solid and fluids;
- Set boundary/initial conditions (BC/IC)
- Set numerical schemes, solver parameters and control parameters
- Run the simulation using – chtMultiRegionSimpleFoam

The domain is a three-dimensional case with cubical shell that is 1m in length inside which flows hot water. In the domain of shell we have two cylindrical pipes made of copper where cold water flows. The width and height of the shell is 0.175m and inner and outer diameter of copper pipe is 22mm and 25mm respectively. The two fluid region are named cold and hot and since there are two pipe the different cold region is named as cold and cold2. Same is the case for copper pipe which is named as solid and solid2. So, all together there are five regions.

The details of the solid and fluid properties are shown in this table.

Table 1: Physical Properties

Solid (Copper)	• Density 8993 • Specific hear 385 • Thermal conductivity 401
Hot Fluid (water)	• Dynamic viscosity 959e-6 • Density 1000 • Specific hear 4181 • Prandtl number 6.62
Cold Fluid (water)	• Dynamic viscosity 959e-6 • Density 1000 • Specific hear 4181 • Prandtl number 6.62

2 Governing Equations and Models

For every region with fluids and solid, various equations are being solved for the fluids and the solid region which are listed below. Thermal boundary condition couples these different regions which connects them.

2.1 Equations for fluid

2.1.1 Equation of Mass conservation

The continuity equation for variable density is defined below whose implementation in OpenFOAM can be found in rhoEqn.H file.

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0$$

2.1.2 Equation of Momentum Conservation

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p_{rgh}}{\partial x_i} + \frac{\partial}{\partial x_j}(\tau_{ij} + \tau_{ji}) + \rho g_i$$

where, $p_{rgh} = p - \rho g h$ is the pressure without the hydrostatic pressure and is initialized from the pressure field in the p-file. τ_{ij} and $\tau_{t_{ij}}$ are the viscous and turbulent stresses. Its OpenFOAM implementation can be found in Ueqn.H.

2.1.3 Equation of Energy conservation

The energy equation or the change rate of the total energy being solved is:

$$\frac{\partial(\rho h)}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_i h) + \frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j k)$$
$$= \frac{\partial \rho}{\partial t} + \rho S + Rad + \rho g_j u_j - \frac{\partial}{\partial x_i}(q_j + q_{ti}) + \frac{\partial}{\partial x_j}(\tau_{ij} u_i)$$

The source code for this can be found in EEqn.H. Here, fluid elements total energy is the summation between internal energy (e) and kinetic energy, $q_i + q_{ti}$ represents the amount of diffusion and turbulence heat transferred to fluid element; heat source term S; the heat source term by radiation is Rad; the enthalpy(h), given by $h = e + \frac{p}{\rho}$; fluid density ρ .

2.2 Equations for solid

For the solid regions, only the energy equation needs to be solved whose implementation can be found in solveSolid.H.

$$\frac{\partial(\rho h)}{\partial t} = \frac{\partial}{\partial x_j}(\alpha \frac{\partial h}{\partial x_j})$$

where, ρ, h and α denote the density, specific enthalpy and thermal diffusivity (κ/c_p) respectively.

2.3 Fluids and solid coupling

The temperature (T) is same at interface of both regions solid and fluid and heat flux across the coupled interface should be same neglecting the radiation, so:

$$T_f = T_s \text{ and } \kappa_f \frac{dT_f}{dn} = -\kappa_s \frac{dT_s}{dn}$$

In above equation, n is a vector normal to the wall. And κ_f and κ_s are the thermal conductivity in two different region which are coupled. Its OpenFOAM implementation can be found in turbulentTemperatureCoupledBaffleMixedFvPatchScalarField.C.

2.4 Turbulence Modeling

The velocity near the wall needs to be modelled using wall functions. In other words, the velocity of the flow at different positions away from the wall is given by wall functions. Usually non dimensionalised forms of the velocity (u+) and distance (y+) are used.

The Y^+ value was selected as 5 based on the paper from M. Sahu et.al where they have listed the following details on wall function value based on Reynolds number.

Table 2: Selection of Y+ Value

Reynolds Number Flow Regime Y plus/First Cell Height
Re = 2000 Laminar First Cell Height = Pipe Radius/38
2000<Re<15000 Turbulent, Enhanced Wall Treatment Y plus<5.0
Re = 15000 Turbulent ,Standard Wall Functions Y plus>30

Turbulence models uses wall functions to model near wall regions in a turbulent flow. In order to satisfy the physics near the wall, the first cell center y_p should be within the log-law region from the wall. The dimensionless wall distance y^+ is given by:

The dimensionless wall distance y^+ is given by

$$y^+ = \frac{y_p \sqrt{0.5 C_f U_\infty^2}}{\nu}$$

Where, y is the distance from the wall,

U_∞ is the free stream velocity, and

ν is the kinematic viscosity.

The skin friction coefficient C_f is given by the -1/4 power law,

$$C_f = 0.078 \, Re^{-\frac{1}{4}}$$

Where, Re is the Reynolds number.

Turbulent Kinetic Energy is calculated using $k = \frac{3}{2}(U_\infty I)^2$. Where the turbulent intensity 'I' can be predicted from experimental data. For fully developed pipe flow we can use this relation, given by $I = 0.16 \, \text{Re}^{-\frac{1}{8}}$.

Similarly, the turbulence dissipation rate is given by

$$\epsilon = \frac{C_\mu^{\frac{3}{4}}}{0.07} \times \frac{k^{1.5}}{L}$$

where, C_μ is empirical constant with value 0.09.

3 Simulation Procedure

In OpenFOAM it is easy to use a case that is already setup and is similar to our problem definition to start our own study. So, the first step in setting up of an OpenFOAM case is to use one of the cases from working directory. Then we will change all input parameters before starting the simulation. For this problem case already present in the tutorial directories which can be accessed by $FOAM_TUTORIALS will be used. In this study PlaneWall2D case is used for the initial setup. The study is done using steady state with laminar and turbulent condition.

3.1 Geometry and Mesh

- The geometry and mesh are generated by using the blockMesh utility of OpenFOAM. Figure 2 generated mesh using blockMesh.
- The generated mesh is divided into different regions using topoSetDict and splitMeshRegions command.
- In the topoSetDict along with different regions for fluid, solid regions faceToSet is used to define faces which later shall be converted into input, output patch and container wall for hot fluid region using createPatchDict.
- The inlet and outlet of shell where hot region flows were simplified to be a square inlet instead of circular and instead of extended protruding structure the inlet and outlet faces are given right at the surface of the shell wall.
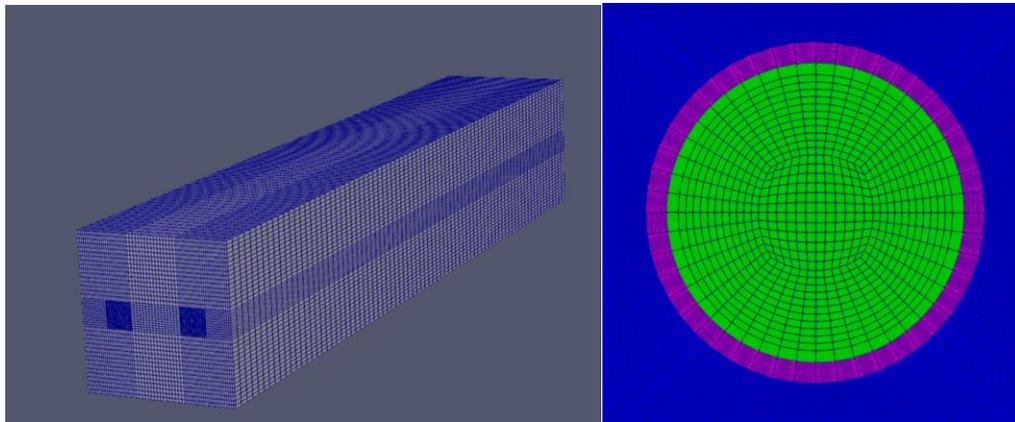

Figure 2: Mesh of the Domain.

3.2 Initial and Boundary Conditions

The details of boundary condition are mentioned in the table below. There are five regions hot, solid, solid2, cold and cold2. Between inlet and outlet of both hot and cold/cold2 regions we have mass flow rate boundary condition for velocity. While no Slip boundary condition is applied to all the walls and interface between the regions. For temperature we have input temperature for the inlet face and insulated boundary condition for the walls and for interfaces as mentioned previously temperature and heat flux across the interfaces are equal.

Table 3: Boundary Condition

Boundary Name	Boundary Conditions		
	Solid/solid2	hot	cold/cold2
inlet_hot/inlet_cold	-	Mass flow rate	Mass flow rate
ext_wall	Wall	-	-
container_wall	-	Wall	-
hot_to_solid/hot_to_solid2	coupled	coupled	-
solid_to_hot/solid2_to_hot	coupled	coupled	-
cold_to_solid/cold_to_solid2	coupled	-	coupled
solid_to_cold/solid2_to_colid	coupled		coupled

Table 4: Details of Solid Boundary Condition

Boundary	Pressure	Temperature
ext_wall	calculated	zeroGradient
container_wall	calculated	zeroGradient
hot_to_solid/hot_to_solid2	calculated	compressible::turbulentTemperatureCoupledBaffleMixed
solid_to_hot/solid2_to_hot	calculated	compressible::turbulentTemperatureCoupledBaffleMixed
cold_to_solid/cold_to_solid2	calculated	compressible::turbulentTemperatureCoupledBaffleMixed
solid_to_cold/solid2_to_colid	calculated	compressible::turbulentTemperatureCoupledBaffleMixed

3.3 Solver

ChtMultiRegionFoam is a solver of steady or transient fluid flow and solid heat conduction, with conjugate heat transfer between regions. Buoyancy effects, turbulence, reactions between chemical species and radiation can also be considered as per required in this solver. For mesh generation and creation of patches './Allrun' command is executed in terminal. Then 'chtMultiRegionSimpleFoam >> log. Solved" command is used to run the simulation and save result in a log file. The log file is continuously updated while the solver is running during this

pyFoam is used to monitor the residuals and continuity using "pyFoamPlotWatcher.py --silent --progress log.solved" command.

The solver follows a segregated solution strategy. This means that the equations for each variable characterizing the system is solved sequentially and the solution of the preceding equations is inserted in the subsequent equation. The coupling between fluid and solid follows also the same strategy: First the equations for the fluid are solved using the temperature of the solid of the preceding iteration to define the boundary conditions for the temperature in the fluid. After that, the equation for the solid is solved using the temperature of the fluid of the preceding iteration to define the boundary condition for the solid temperature. This iteration procedure is executed until convergence. For each region defined as fluid, the according equation for the fluid is solved and the same is done for each solid region. The regions are coupled by a thermal boundary condition.

A general file Structure of the multi-region case looks like this:
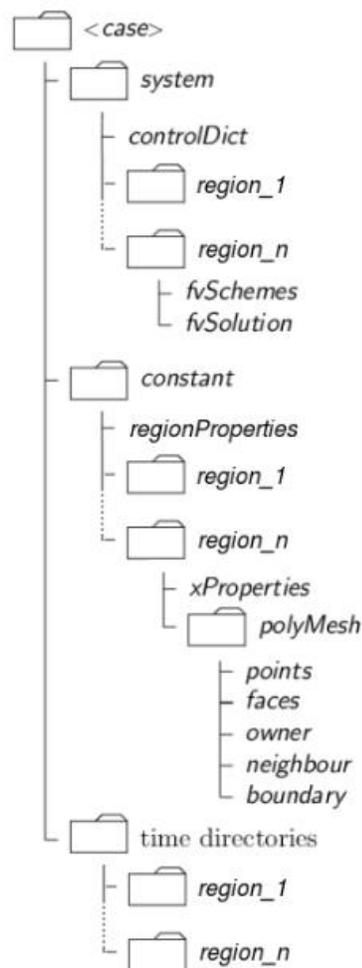
Figure 3: File Structure of multi-region problem in OpenFOAM

For our problem below is the File Structure for system folder is shown here:

```
├── blockMeshDict          ├── controlDict          ├── solid
├── cold                   ├── createPatchDict      │   ├── fvSchemes
│   ├── fvSchemes          ├── fvSchemes            │   └── fvSolution
│   └── fvSolution         ├── fvSolution           ├── solid2
├── cold2                  ├── hot                  │   ├── fvSchemes
│   ├── fvSchemes          │   ├── fvSchemes        │   └── fvSolution
│   └── fvSolution         │   └── fvSolution       └── topoSetDict
```
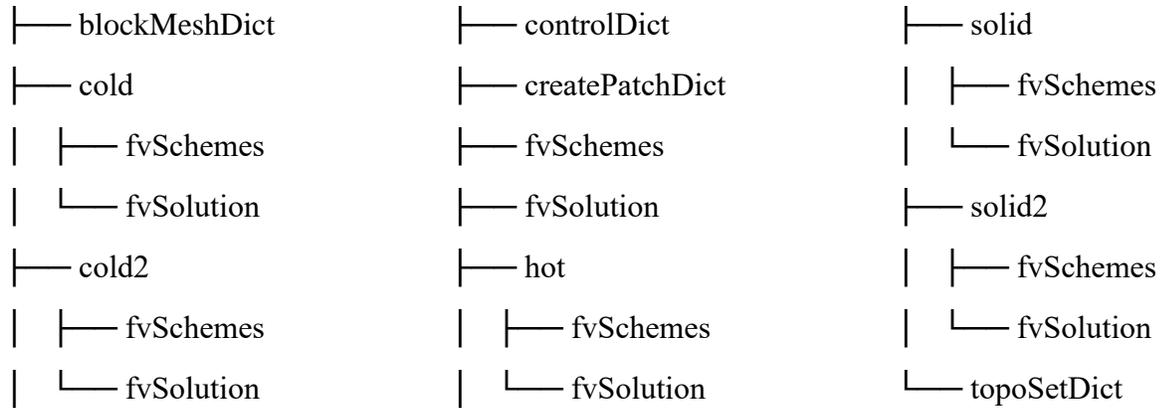
Similarly, the File Structure for '0' folder is show here:

```
├── cellToRegion          │   ├── alphat           │   ├── k
├── cold                  │   ├── cellToRegion     │   ├── nut
│   ├── T                 │   ├── epsilon          │   ├── p
│   ├── U                 │   ├── k                │   └── p_rgh
│   ├── alphat            │   ├── nut              ├── solid
│   ├── cellToRegion      │   ├── p                │   ├── T
│   ├── epsilon           │   └── p_rgh            │   ├── cellToRegion
│   ├── k                 ├── customRegexp         │   └── p
│   ├── nut               ├── hot                  └── solid2
│   ├── p                 │   ├── T                    ├── T
│   └── p_rgh             │   ├── U                    ├── cellToRegion
├── cold2                 │   ├── alphat               └── p
│   ├── T                 │   ├── cellToRegion
│   ├── U                 │   ├── epsilon
```
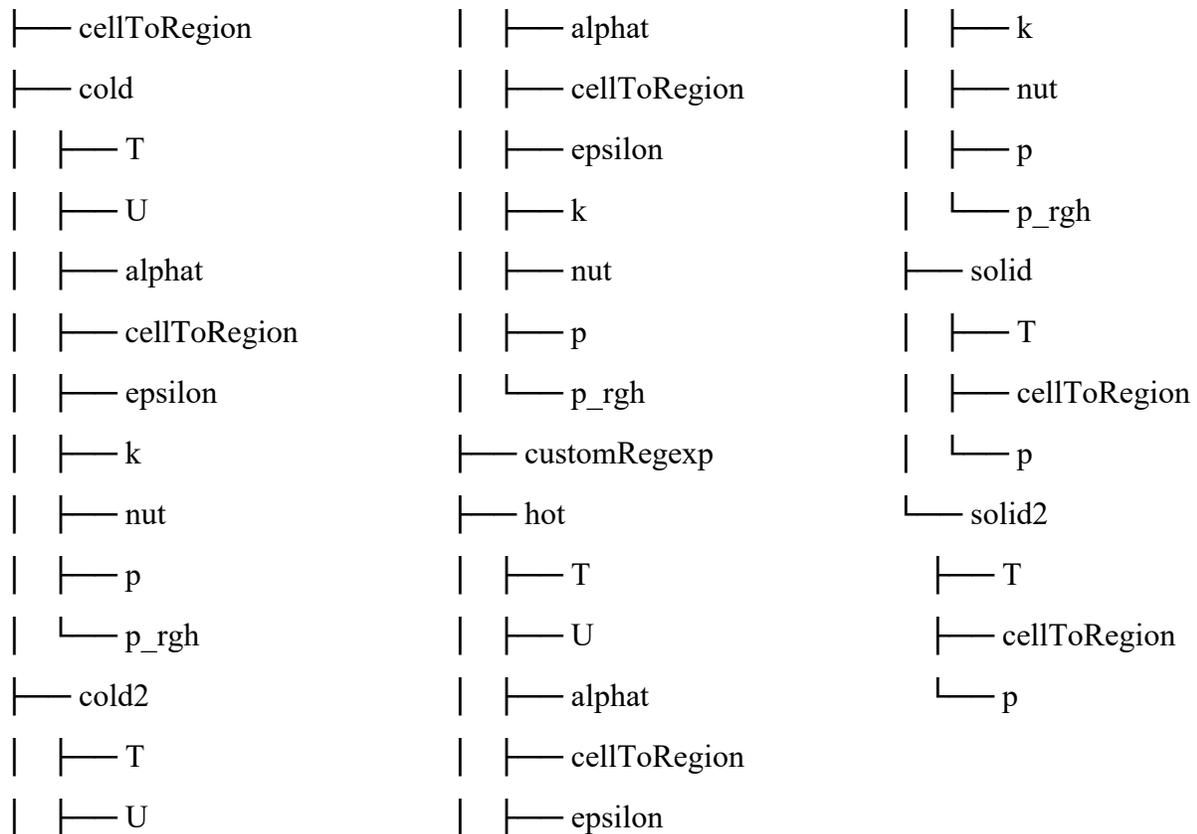
4 Results and Discussion

In this section, the numerical results obtained from the simulation is shown, the contours and plots have been obtained using ParaView and extracted data from it. The physical properties of the various regions are listed in Table 1. The input flow rate is 0.05kg/s and inlet temperature for cold water is 12^0C and inlet temperature for hot water is 90^0C.

For the visualization of the simulations results from OpenFOAM, ParaView is used. In order to do this after the simulation is complete, we can run command "touch result.foam" this will create "result.foam" file in our work directory which can be opened using ParaView for post-processing and related stuff.

4.1 Residuals

The residuals obtained from the simulation is shown in the figure below. It can be seen that solution has converged well as the residuals are decreasing and in the order of 10^{-9}.
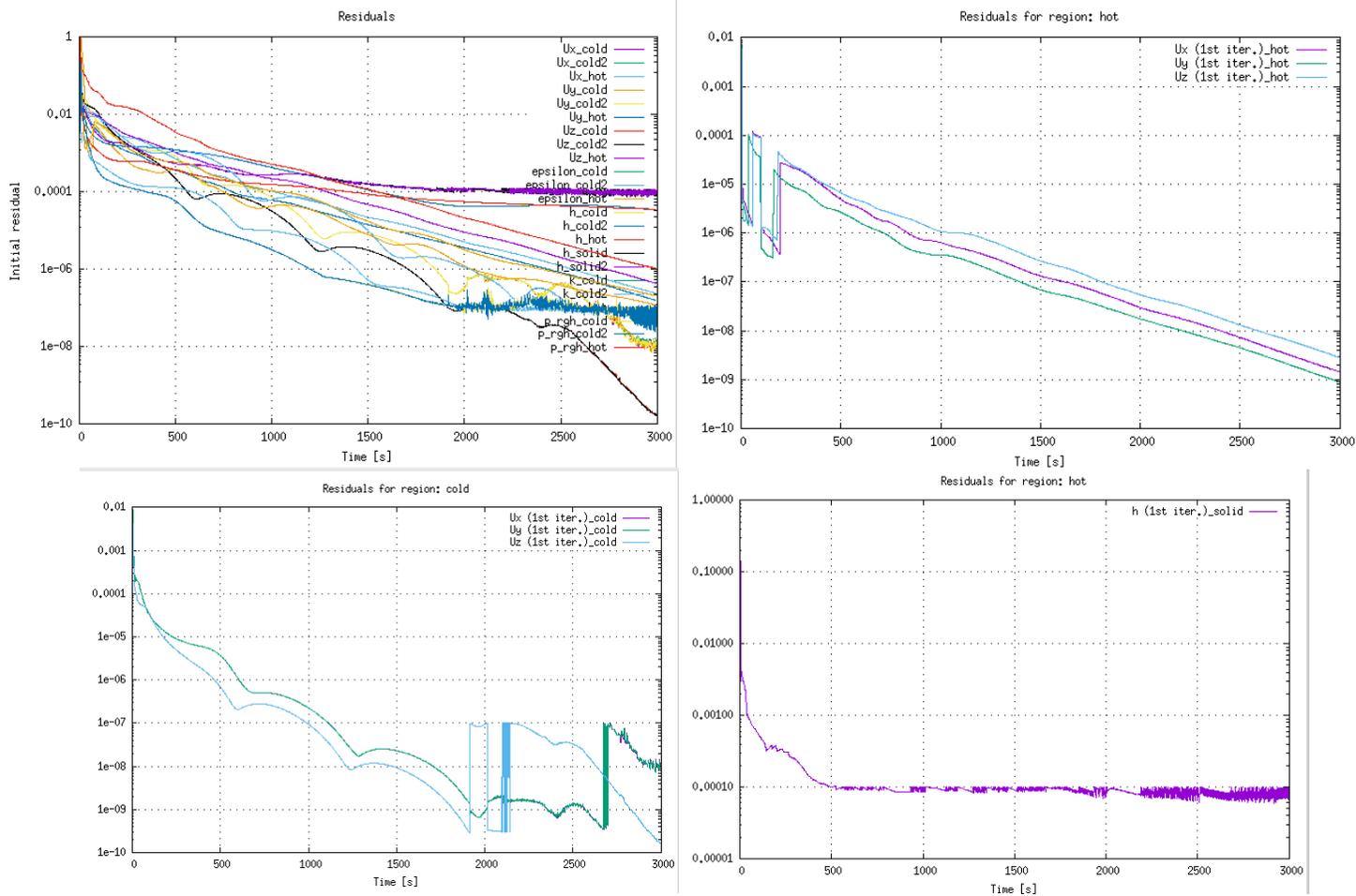

Figure 4: Residual Monitoring

4.2 Output Temperature Profile for hot and cold regions
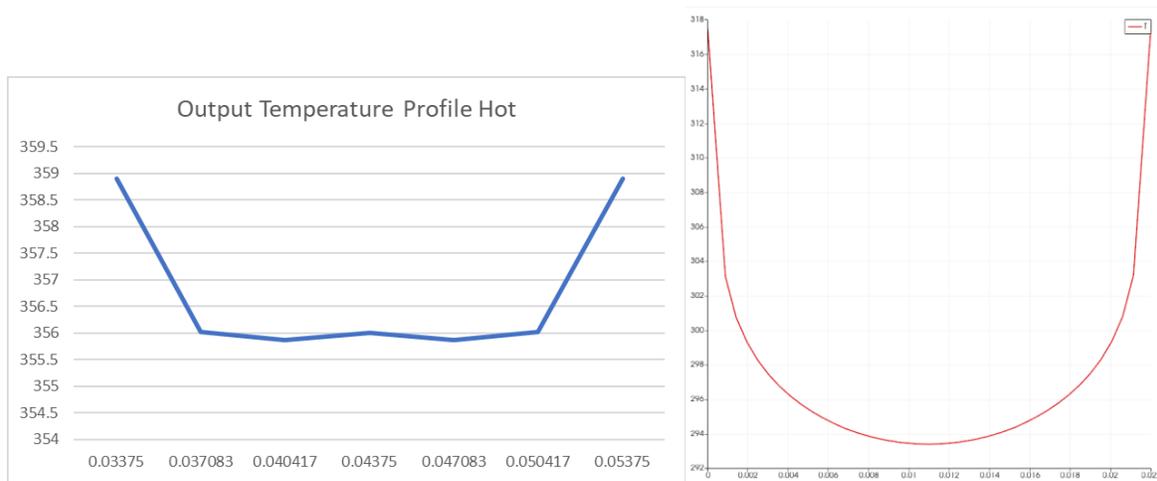
Figure 5: output temperature profile of hot region(left) and cold region(right)

The output temperature Profile for hot can cold region can be seen in the above figure respectively. Because of a smaller number of cells in the output of hot region the profile is not smooth. The mean temperature of the flow is 286.89K for cold fluid output and 356.80 for hot fluid output. Since the flow is symmetric the values and profiles are same in the other pipe that is region of cold2.

**4.3 Output Velocity Profile for hot and cold regions**
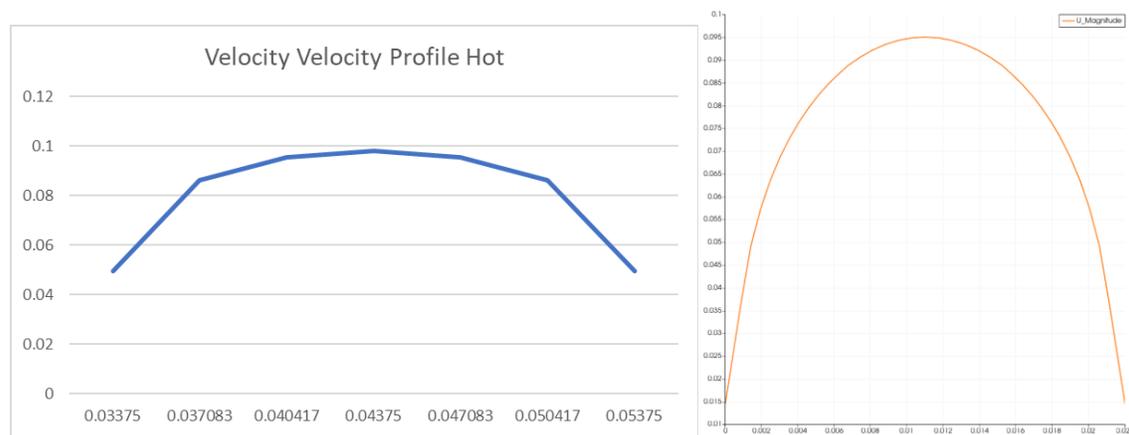
Figure 6: output velocity profile of hot region(left) and cold region(right)

The output temperature Profile for hot can cold region can be seen in the above figure respectively. Again, because of a smaller number of cells in the output of hot region the profile is not smooth. It can be seen that the flow is fully developed in the cold region.

4.4 Contour of Velocity and Temperature in Cold Region

The contour for velocity and Temperature for cold region can be seen in the figure below.
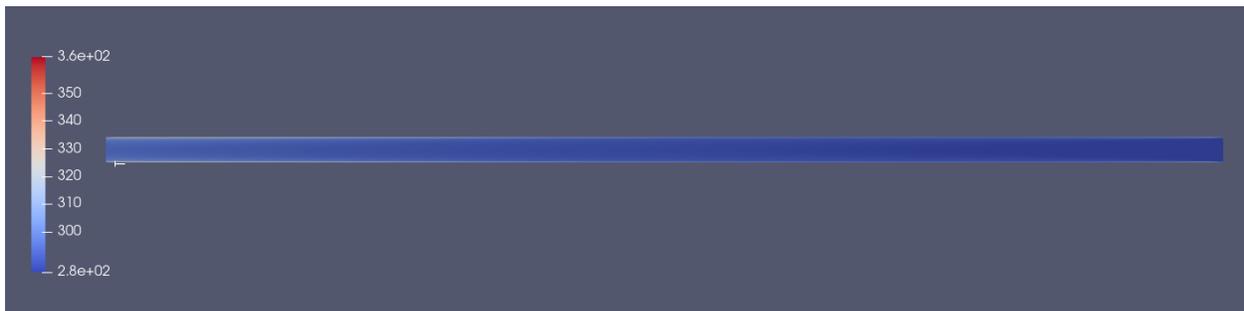
Figure 7: Temperature contour in cold region
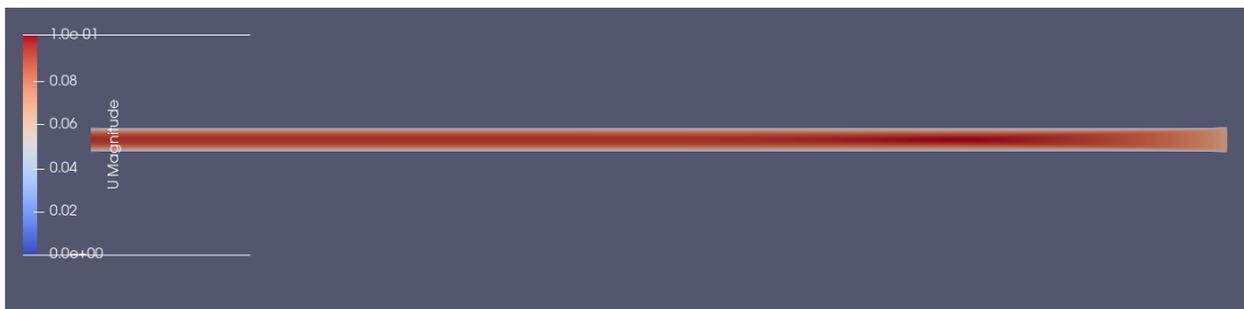
Figure 8: velocity contour in cold region

4.5 Streamlines of the Flow
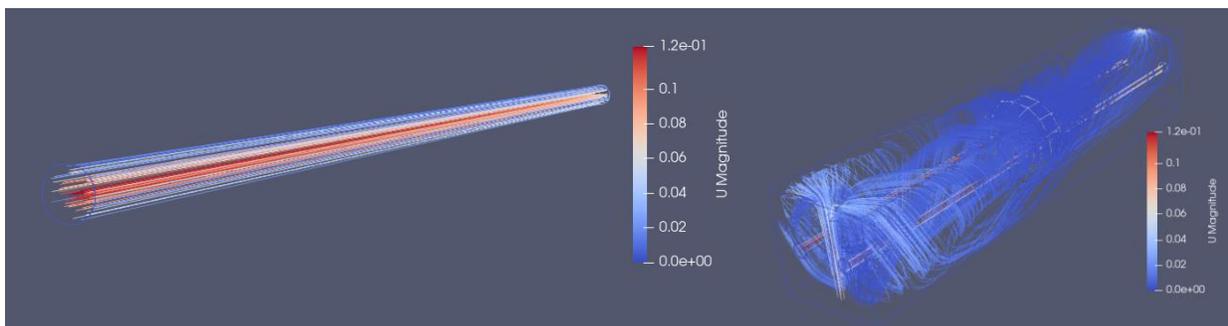
Figure 9: Streamlines in cold(left) and hot(region)

The Streamlines for the cold and hot region are shown in the figure above. It can be seen that for the cold region the streamlines are perfect straight lines. In the hot region, fluid is forming vortices around the cylindrical pipe inside which cold water flows. This is somewhat similar to flow past cylinder, one can argue that this was a design feature to increase mixing thus improving the heat transfer rate of heat exchanger.

4.6 Validation of Results
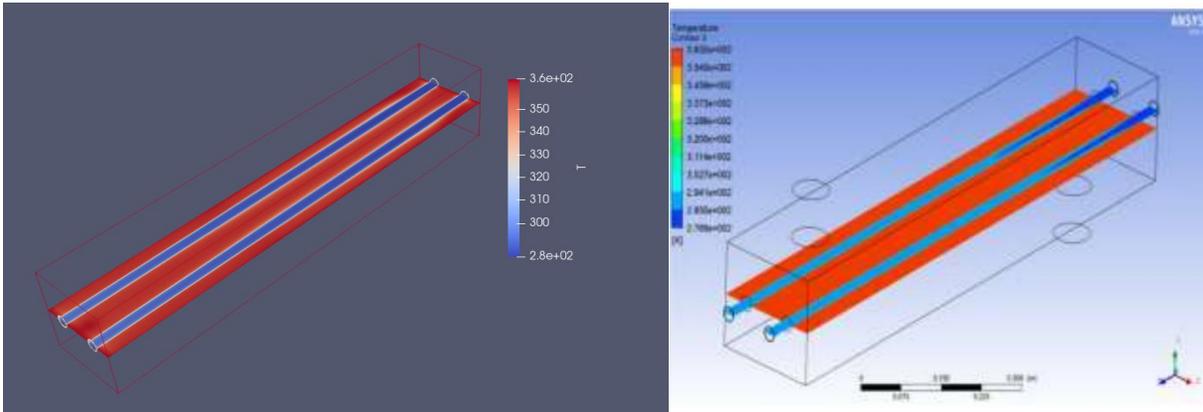
Figure 10: Temperature Contour along x-axis current study(left) previous published study(right)

The above figure compares the temperature contour obtained from present study and published paper being referred. The temperature contour looks similar. The paper being referred has provided data for input and output temperature of the study and can be seen in the table below. Our case is 'Parallel tubes' case and we will use this data to verify our result.

Table 5: Reference paper data

Pattern of Tubes	Cold inlet (°C)	Cold outlet (°C)
Parallel tubes	12	21
'S' Pattern tubes	12	35
Zigzag Pattern tubes	12	54

Pattern of Tubes	Hot inlet (°C)	Hot outlet (°C)
Parallel tubes	90	81
'S' pattern tubes	90	80
Zigzag Pattern tubes	90	70

4.6.1 Validation using Cold Water Temperature

From the Excel sheet extracted the Mean Temperature of output was found to be 296.89K i.e., $23.89C^0$. While in paper it is $21C^0$. An excerpt of the excel sheet is shown below.

Table 6: Calculation of cold-water outlet temperature

Q2				f_x	=AVERAGE(A:A)-273												
	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q
1	T	U:0	U:1	U:2	alphat	epsilon	k	nut	p	p_rgh	rho	vtkValidPo	arc_length	Points:0	Points:1	Points:2	Average Temperature
2	317.5	2.27E-11	1.74E-09	0.014452	0.000682	0.002586	0.000198	6.82E-07	0.036685	0.036685	1000	1	0	0	-0.011	1	23.89447552
3	317.13	2.39E-11	1.84E-09	0.015011	0.000706	0.002567	0.000197	7.06E-07	0.035766	0.035766	1000	1	2.20E-05	0	-0.01098	1	
4	316.77	2.50E-11	1.93E-09	0.01557	0.00073	0.002547	0.000197	7.30E-07	0.034846	0.034846	1000	1	4.40E-05	0	-0.01096	1	
5	316.41	2.62E-11	2.03E-09	0.01613	0.000754	0.002528	0.000196	7.54E-07	0.033927	0.033927	1000	1	6.60E-05	0	-0.01093	1	
6	316.05	2.74E-11	2.13E-09	0.016689	0.000778	0.002508	0.000195	7.78E-07	0.033007	0.033007	1000	1	8.80E-05	0	-0.01091	1	
7	315.69	2.85E-11	2.23E-09	0.017248	0.000802	0.002488	0.000195	8.02E-07	0.032088	0.032088	1000	1	0.00011	0	-0.01089	1	

$$Error = \frac{296.89 - 294}{294} = 9.82 \times 10^{-3} = 0.98\%$$

4.6.2 Validation using Hot Water Temperature

From extracted Excel Sheet the mean temperature of output for hot region was found to be 356.80K which is 83.80C°. While in paper it is 81C°. An excerpt of the excel sheet is shown below.

Table 7: Calculation of hot water outlet temperature

	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q
	T	U:0	U:1	U:2	alphat	epsilon	k	nut	p	p_rgh	rho	vtkValidPo	arc_length	Points:0	Points:1	Points:2	Taverage
1																	
2	358.9	0.026348	0.041788	-0.0026	0.001862	4.84E-05	3.63E-05	1.86E-06	2.5104	2.5104	1000	1	0	0.03375	0.0875	0.025	83.79571
3	356.02	0.034368	0.0789	-0.00448	0.003637	8.88E-05	5.99E-05	3.64E-06	0	0	1000	1	0.003333	0.037083	0.0875	0.025	
4	355.86	0.017292	0.09356	-0.00478	0.004703	0.000124	8.05E-05	4.70E-06	0	0	1000	1	0.006667	0.040417	0.0875	0.025	
5	356.01	3.64E-07	0.097819	-0.00487	0.005229	0.000143	9.12E-05	5.23E-06	0	0	1000	1	0.01	0.04375	0.0875	0.025	

$$Error = \frac{356.80 - 354}{354} = 7.909 \times 10^{-3} = 0.79\%$$

4.7 Miscellaneous Results

The simulation was carried out without turbulence i.e., laminar simulation and it was found that on doing so the heat transfer rate decreases as the output temperature of the cold-water decreases compared to the data obtained using turbulent simulation. This, could be attributed to the fact that turbulence causes mixing and thus increase in heat transfer rate. Also, on increasing the cold-water mass flowrate, the heat transfer rate decreases, this probably is due to the fact that now the cold water flowing has less time to gain heat from the system. The summary of the data obtained from these simulations is shown in the bar chart below.
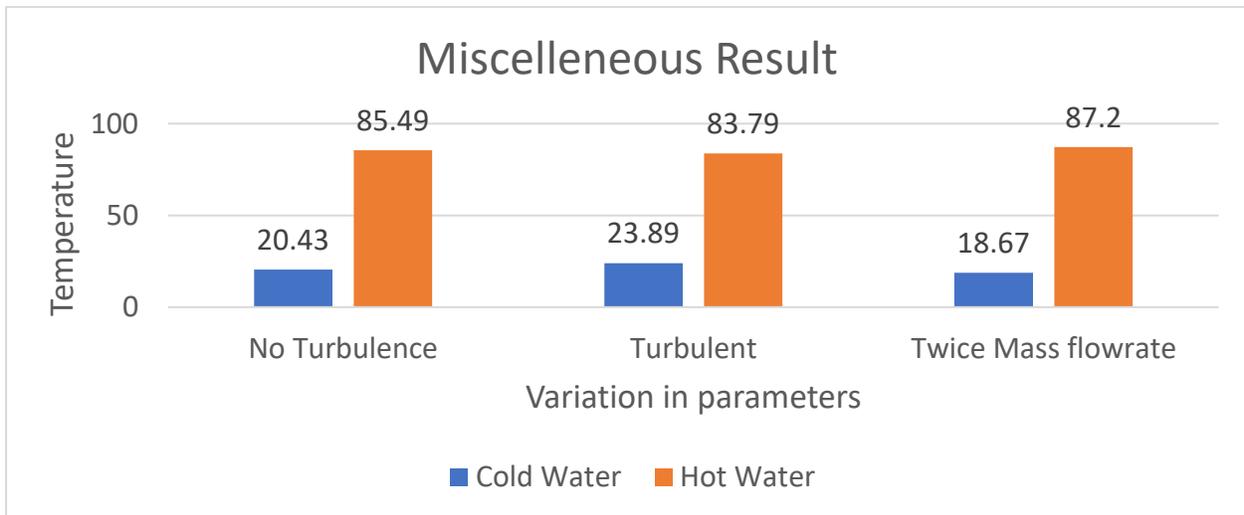
References

[1] DEVELOPED LAMINAR FLOW IN PIPE USING COMPUTATIONAL FLUID DYNAMICS, M. Sahu1 , Kishanjit Kumar Khatua2 and Kanhu Charan Patra3 , T. Naik4

[2] CFD ANALYSIS OF HEAT EXCHANGER MODELS DESIGN USING ANSYS FLUENT, Ram Kishan, Devendra Singh, Ajay Kumar Sharma*

[3] https://openfoamwiki.net/index.php/ChtMultiRegionFoam#:~:text=Solver%20for%20steady%20or%20transient,turbulence%2C%20reactions%20and%20radiation%20modelling

[4] https://foamingtime2.wordpress.com/starting-with-openfoam/mr-semicylinder-generation/