

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

Dr. Raj Kumar Saini

Ph.D, Indian Institute of Technology, Bombay (IIT Bombay)

M.Tech, Indian Institute of Technology, Madras (IIT Madras)

Email : raj.km.saini@gmail.com

Abstract

This case study demonstrates the simulation of a shell and tube heat exchanger. This type of heat exchanger (shell and tube) is most commonly used in oil refineries and other large chemical processes and where it suits for higher-pressure applications. It is an example of heat transfer between two fluids. The present case also describes the conjugate heat transfer in multi-region (solid and fluids). The flow is considered steady state, non-isothermal and turbulent. The simulations are performed using OpenFOAM-v7. The hydrodynamics of flow between tube and shell is investigated. The temperature and velocity profile is analyzed obtained from the simulation.

Keywords: Fluid flow, Heat transfer, Shell and tube heat exchanger, CFD, OpenFOAM

1 Introduction

The purpose of this case study is to learn OpenFOAM software [1] to the new users and to understand conjugate heat transfer in three dimensional flow in a shell and tube heat exchanger[2, 3]. In this simulation, a pressure-based finite volume method is used for incompressible, transient, non-isothermal and transient flow. The simulations are carried out using OpenFOAM-v7 [1]. In this case, conjugate heat transfer simulation are carried out for incompressible transient flow through a shell and tube heat exchanger with using **chtMultiRegionFoam** solver in OpenFOAM-v7 [1]. This case study demonstrates how to do the following:

- Set up a problem case;
- Creating a 3D mesh by using blockMesh & snappyHexMesh utility;
- Create the geometry and import the geometry in OpenFOAM;
- Set up the properties of the fluids;
- Initialize the flow;
- Consider the laminar model for laminar flow regime;
- Set boundary/initial conditions (BC/IC);
- Set numerical schemes, solver parameters and control parameters;
- Brief explanation of solver (chtMultiRegionFoam);
- Post processing the case for results.

2 Problem statement

This case considers the transient simulation of conjugate heat transfer in a shell and tube heat exchanger in OpenFOAM. Figure 1 shows the details of the geometry considered in the present study.

The domain is considered a three-dimensional case by assuming that the geometry has a height of heat exchanger in the y-direction. The geometric parameters of the domain such as length and diameter of shell are considered with 0.250 m and 0.08 m and the diameters (in/out) of tube are 0.012 and 0.020 m respectively. The solid and fluids regions are created using "createBaffles" in a 3D environment.

The two fluids are considered like cold and hot fluids. Table 1 shows the solid and fluids properties for the present study.

Initially, temperature is patched with value of 700 (600, 500) and 300 K. At time zero, the hot and cold fluids start entering and moving to leftwards in the heat exchanger. The mass flow rates of fluids at inlet are considered 0.05, & 0.05 kg/s respectively (+ve, y-direction).

Table 1: Details of solid and fluids properties

Solid property	
Parameters	Value
Density (ρ), kg/m ³	2700
Specific heat (C_p), J/(K.kg)	900
Thermal conductivity (κ), W/m.K	200
Shell (Hot, water) property	
Dynamic viscosity (μ), Pa.s	959e-6
Density (ρ), kg/m ³	1000
Specific heat (C_p), J/(K.kg)	4181
Thermal conductivity (k), W/m.K	0.6
Tube (Cool, water) property	
Dynamic viscosity (μ), Pa.s	959e-6
Density (ρ), kg/m ³	1000
Specific heat (C_p), J/(K.kg)	4181
Thermal conductivity (k), W/m.K	0.6

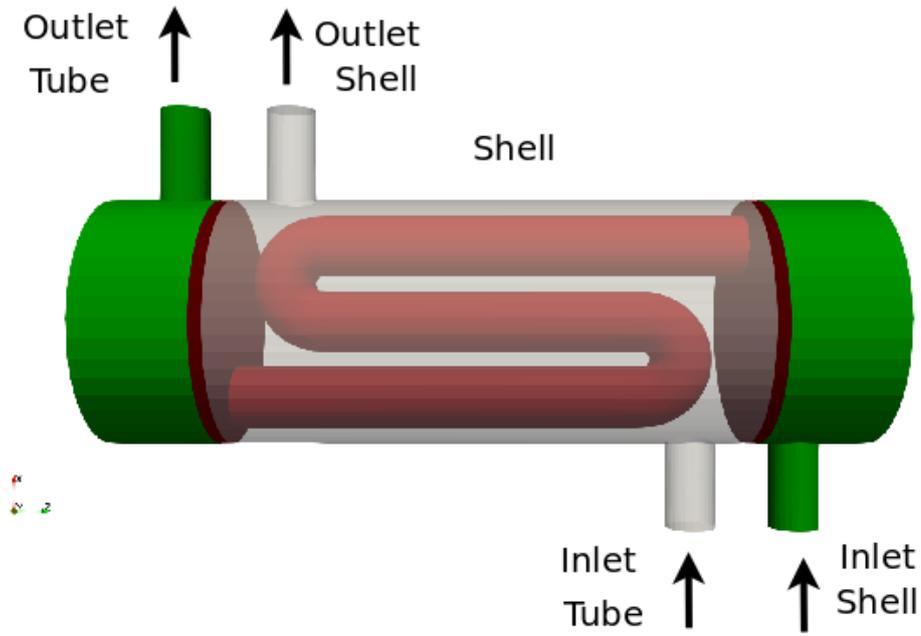


Figure 1: Schematic diagram of shell and tube heat exchanger

3 Mathematical modeling

For each region defined as fluids and solid, the equation for the fluids are solved as well as the solid region. The solid and fluids regions are coupled by a thermal boundary condition [4, 5, 6].

3.1 Equations for fluid

3.1.1 Continuity equation

The continuity equation for constant-density is defined as:

$$\frac{\partial u_j}{\partial x_j} = 0 \quad (1)$$

3.1.2 Momentum equation

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j u_i) = -\frac{\partial p_{rgh}}{\partial x_i} + \frac{\partial}{\partial x_j} (\tau_{ij} + \tau_{tj}) + \rho g_i \quad (2)$$

where, u represents the velocity, ρ density of the fluid, $p_{rgh} = p - \rho gh$, the pressure minus the hydrostatic pressure, g_i the gravity and τ_{ij} , and τ_{tj} are the turbulent, and viscose stresses respectively.

3.1.3 Energy conservation

The energy equation or the change rate of the total energy can be written as (Equation- 3):

$$\frac{\partial(\rho h)}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j h) + \frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j k) = -\frac{\partial(q_i + q_{ti})}{\partial x_i} + \rho S + Rad + \frac{\partial p}{\partial t} - \rho g_j u_j + \frac{\partial}{\partial x_j} (\tau_{ij} u_i) \quad (3)$$

where, kinetic energy ($k = 0.5 u_i u_i$), internal energy (e); heat transferred to fluid element by diffusion and turbulence $q_i + q_{ti}$; heat source term S ; the heat source by radiation Rad ; the enthalpy(h), $h = e + p/\rho$; density of fluid ρ .

3.2 Equations for solid

For the solid regions, the energy equation can be written as (Equation- 4):

$$\frac{\partial(\rho h)}{\partial t} = \frac{\partial}{\partial x_j} \left(\alpha \frac{\partial h}{\partial x_j} \right) \quad (4)$$

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

3.3 Coupling between fluids and solid

The temperature (T) is same at interface both phases .i.e solid (s) and fluid (f).

$$T_f = T_s \quad (5)$$

The heat flux entering one region at one side of the interface, it should be equal to the heat flux leaving the other region on the other side of the domain.

$$Q_f = -Q_s \quad (6)$$

If, radiation is neglected then the above expression can be written as:

$$\kappa_f \frac{dT_f}{dn} = -\kappa_s \frac{dT_s}{dn} \quad (7)$$

where, n represents the direction normal to the wall. κ_f and κ_s are the thermal conductivity of the fluid and solid respectively.

3.4 Boundary conditions

Details of boundary's name and corresponding boundary's conditions are presented in Table 2.

Table 2: Boundary conditions

Boundary Name	Boundary conditions		
	solid	water (shell)	water (tube)
lower		Mass flow rate	Mass flow rate
upper		Pressure	Pressure
wall		wall	wall
external	wall	wall	wall
shell_to_solid	coupled	coupled	coupled
solid_to_shell	coupled	coupled	coupled
tube_to_solid	coupled	coupled	coupled
solid_to_tube	coupled	coupled	coupled

4 Simulation procedure

This case deals with three-dimensional turbulent simulation of a shell and tube heat exchanger. First step in setting up of an OpenFOAM case is to present working directory. We need to set all required input parameters before starting the simulation. Mesh generation and implementation of boundary conditions are adopted from OpenFOAM/(username)-7/run/tutorials/heatTransfer/chtMultiRegionFoam/. This study is considered with steady state and turbulent case.

4.1 Creating geometry and mesh

- Geometry for the present problem is considered 3D dimensional domain. The geometry and mesh are generated by using the blockMesh & snappyHexMesh utility.
- Figure 2 shows the isometric view of the generated mesh using blockMesh & snappyHexMesh utility.
- Meshing a geometry with more than one region.
- All modifications for mesh with proper boundary conditions with defining inlet and outlet is to be done as velocity patch and its neighbor patch (`~/case/constant/polyMesh/boundary`).

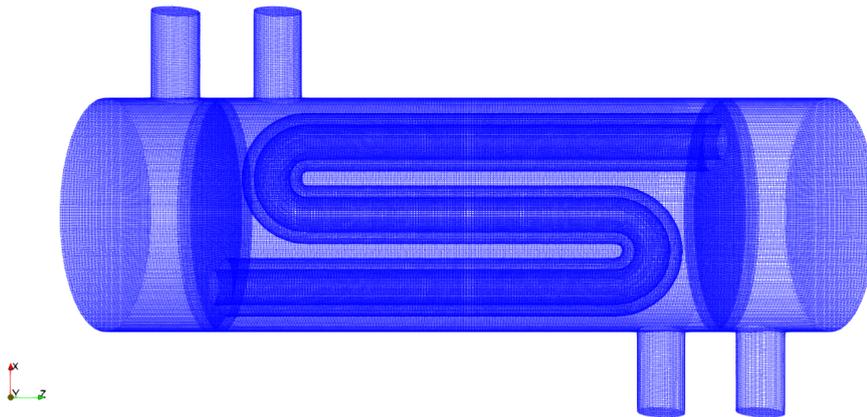


Figure 2: Computational geometry of heat exchanger

4.2 Setting boundary conditions (BC)/Initial conditions (IC)

All the boundary conditions for fields variables are mentioned in '0' file folder. Boundary conditions are discussed and that remain same during simulation. Files present in '0' folder (`~/case/0/shell/`, `~/case/0/solid/` and `~/case/0/tube/`) has been kept 'cellToRegion', 'p', 'p_rgh', 'T' and 'U' files for turbulent and steady flow. Boundaries are assigned and added seven boundaries of present case in three the files, i.e. 'cellToRegion', 'p', 'p_rgh', 'T', 'U', 'nut' and 'epsilon' respectively. Details of the boundary conditions are listed in Table 3, 4 & 5.

Table 3: Details of solid boundary conditions

Boundary	p	T
external	calculated	zeroGradient
shell_to_solid	calculated	compressible::turbulentTemperatureCoupledBaffleMixed
solid_to_shell	calculated	compressible::turbulentTemperatureCoupledBaffleMixed
tube_to_solid	calculated	compressible::turbulentTemperatureCoupledBaffleMixed
solid_to_tube	calculated	compressible::turbulentTemperatureCoupledBaffleMixed
walls	calculated	calculated

4.3 Solver details

In the present study, steady state and turbulent flow are considered. Turbulent flow model can be applied in the OpenFOAM by using ‘simulationType’ option in the ‘turbulenceProperties’ file in constant folder. In order to run study state simulations, controlDict, decomposeParDict (for parallel computation), fvSchemes, fvSolution, and setFieldsDict files are kept in the system directory folder. ‘./run’ command executes in terminal to run computations.

Here is the brief introduction of **OpenFOAM**, a toolbox of CFD simulation: "OpenFOAM" is an open source toolbox for CFD simulations. "chtMultiRegionFoam" is an open source CFD solver of OpenFOAM. Multi-region OpenFOAM case structure is slightly adapted from the standard case structure. Within each "fluids" and "solid" subdirector, there exists the standard contents ("polyMesh", "transportProperties", "fvSchemes", "fvSolution", etc.)

The structure of the multi-region **case** is like this:

```

case
|----0/
  |---- shell /
        |---- alphas
        |---- epsilon
        |---- k
        |---- nut
        |---- p
        |---- p_rgh
        |---- T
        |---- U
  |---- solid
        |---- p
        |---- T
  |---- tube /
        |---- alphas
        |---- epsilon
        |---- k
        |---- nut
        |---- p
        |---- p_rgh
        |---- T
        |---- U

```

Table 4: Details of fluid (shell) boundary conditions

Boundary	k	p_rgh	p	T	U	nut	epsilon
lower	inletOutlet	calculated	fixedValue	inletOutlet	pressureInletOutletVelocity	calculated	inletOutlet
upper	turbulentIntensity KineticEnergyInlet	calculated	fixedFluxPressure	fixedValue	flowRateInletVelocity	calculated	turbulentMixingLength
wall	kqRWallFunction	calculated	fixedFluxPressure	fixedValue	noSlip	nutkWallFunction	epsilonWallFunction
shell_to _solid	kqRWallFunction	calculated	fixedFluxPressure *	*	noSlip	nutkWallFunction	epsilonWallFunction
solid_to _shell	kqRWallFunction	calculated	fixedFluxPressure *	*	noSlip	nutkWallFunction	epsilonWallFunction

*compressible::turbulentTemperatureCoupledBaffleMixed

Table 5: Details of fluid (tube) boundary conditions

Boundary	k	p_rgh	p	T	U	nut	epsilon
lower	inletOutlet	calculated	fixedValue	inletOutlet	pressureInletOutletVelocity	calculated	inletOutlet
upper	turbulentIntensity KineticEnergyInlet	calculated	fixedFluxPressure	fixedValue	flowRateInletVelocity	calculated	turbulentMixingLength
wall	kqRWallFunction	calculated	fixedFluxPressure	fixedValue	noSlip	nutkWallFunction	epsilonWallFunction
tube_to _solid	kqRWallFunction	calculated	fixedFluxPressure *	*	noSlip	nutkWallFunction	epsilonWallFunction

*compressible::turbulentTemperatureCoupledBaffleMixed

```

|-----constant /
    |-----shell /
        |-----g
        |-----thermophysicalProperties
        |-----turbulenceProperties
    |-----solid /
        |-----thermophysicalProperties
    |-----triSurface /
        |-----stl files
        |-----stl files
        |-----stl files
    |-----tube /
        |-----g
        |-----thermophysicalProperties
        |-----turbulenceProperties
    |-----regionProperties

|-----system /
    |-----shell /
        |-----decomposeParDict
        |-----fvSchemes
        |-----fvSolution
    |-----solid
        |-----decomposeParDict
        |-----fvSchemes
        |-----fvSolution
    |-----tube /
        |-----decomposeParDict
        |-----fvSchemes
        |-----fvSolution
    |-----blockMeshDict
    |-----controlDict
    |-----createBafflesDict
    |-----decomposeParDict
    |-----meshQualityDict
    |-----fvSchemes
    |-----snappyHexMeshDict
    |-----residuals
|-----clean . sh
|-----run . sh

```

4.4 Post-processing

The paraFoam, it can be used to visualize the simulations results in OpenFOAM. This can be run by typing the following command line in the terminal **paraFoam -builtin** to open the ParaView software and upload the case.

5 Results and discussion

In this section, the numerical results are shown for conjugate heat transfer (conduction and convection) in a shell and tube heat exchanger. The physical properties of the solid and fluids (hot and cold) are given in Table 1.

Figure 3 shows the temperature distribution in the shell and tube heat exchanger, viewed along the z-axis. In Figure 3a, it is observed that the temperature distribution is more at high inlet water temperature in the shell side, $T=700$ K, and less temperature distribution at low inlet water temperature, $T=500$ K (Figure 3c).

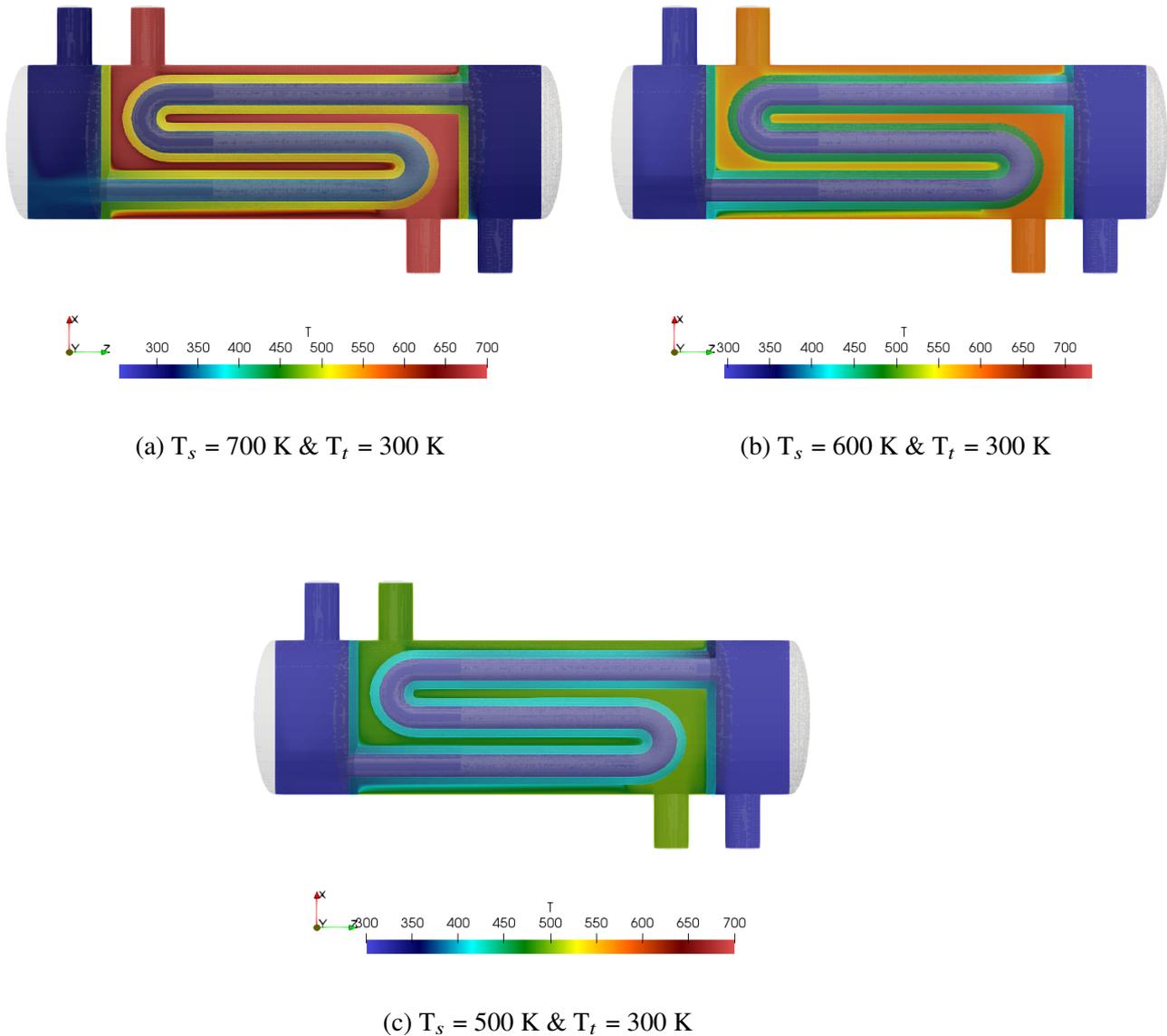


Figure 3: Temperature contour plots in shell and tube heat exchanger different inlet temperature of water at constant mass flow rate, $m = 0.05$ kg/s (steady-state achieved)

Figure 4 shows the velocity contour plots captured for three different inlet water temperatures

in the shell side, $T=700$ 600 & 500 K with constant mass flow rate, $m' = 0.05$ kg/s. It can be seen in Figure 4, there is no change in the velocity profile because the mass flow rate in the shell side and tube side is kept constant in all three cases. The velocity tends to zero near the wall so it is verified no-slip condition and maximum velocity at the center of the tube.

The simulation results are analyzed when the system is archived steady-state (more than 1000 iterations).

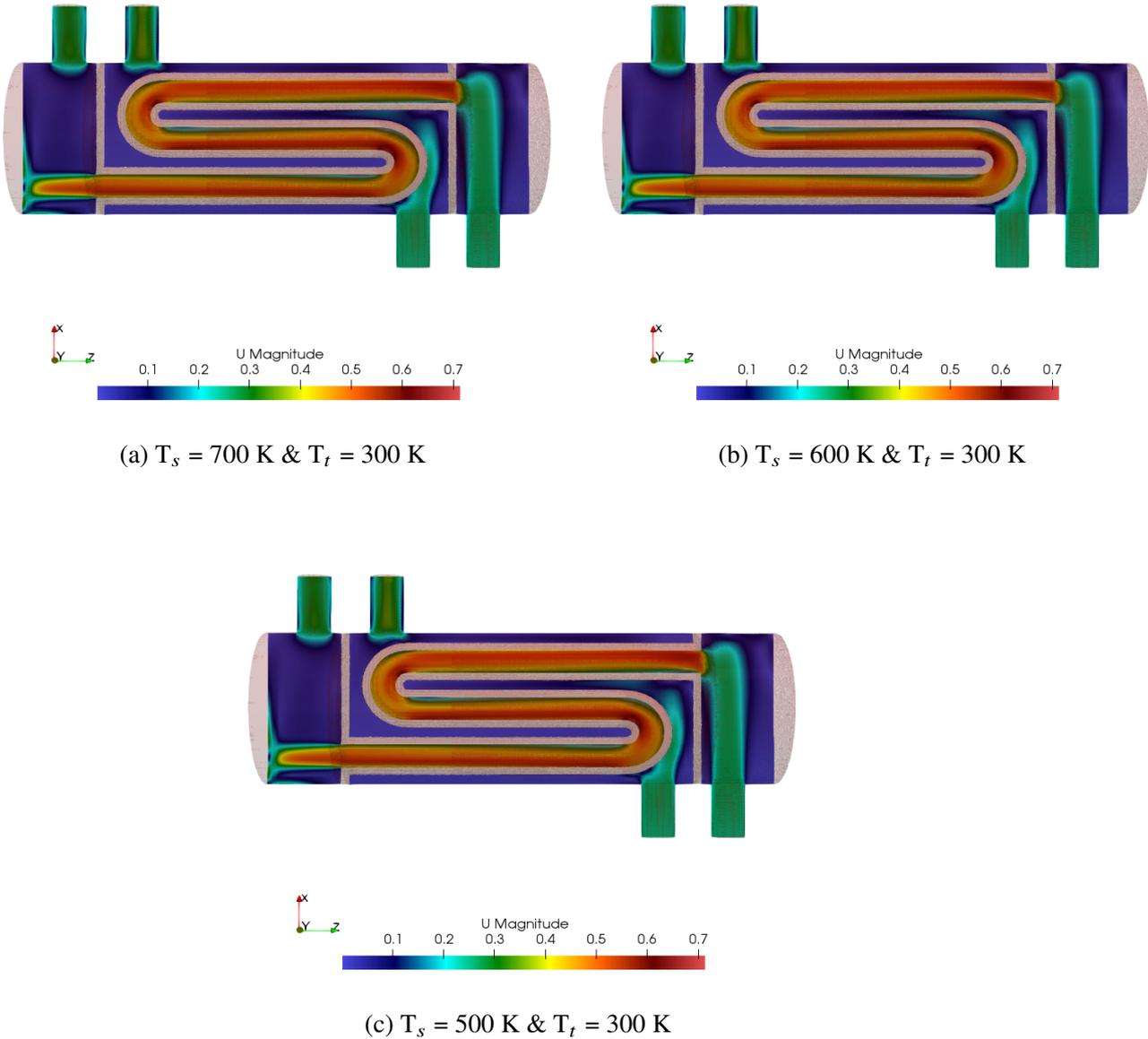


Figure 4: Velocity contour plots in shell and tube heat exchanger different inlet temperature of water at constant mass flow rate, $m' = 0.05$ kg/s (steady-state achieved)

References

[1] C. J. Greenshields, *OpenFOAM: The OpenFOAM Foundation. User Guide Version 7.* CFD Direct Limited, July. 2019.

- [2] J. B. B. Rao and V. R. Raju, "Numerical and heat transfer analysis of shell and tube heat exchanger with circular and elliptical tubes," *International Journal of Mechanical and Materials Engineering*, vol. 11, no. 1, p. 6, 2016.
- [3] N. Afsar and M. I. Inam, "Cfd analysis of shell and tube heat exchanger with different baffle orientation and baffle cut," *AIP Conference Proceedings*, vol. 1980, no. 1, p. 050006, 2018.
- [4] L. M. Moukalled, F. and M. Darwish, *The finite volume method in computational fluid dynamics. An Advanced Introduction with OpenFOAM and Matlab*, 2016.
- [5] M. Darwish, F. Moukalled, *A unified formulation of the segregated class of algorithms for fluid flow at all speeds. Numerical Heat Transfer: Part B: Fundamentals*, 2000.
- [6] P. J. P. Robert W. Fox, Alan T. McDonald, *Introduction to Fluid Mechanics (8th ed)*, 8th ed., 2011.