

Modeling and Simulation of Concentric Pipe Heat Exchanger with Varying Inlet Velocities

Research Scholar
Jasvinder Singh Saluja

Department of Mechanical Engineering,
LNCTS (RIT) Indore, MP, India
e-mail: jasvinder.saluja10@gmail.com

Assistant Professor
Rahul Vishwakarma

Department of Mechanical Engineering,
LNCTS (RIT) Indore, MP, India
e-mail: vishwakarmarahul39@gmail.com

Assistant Professor
Vipul Jain

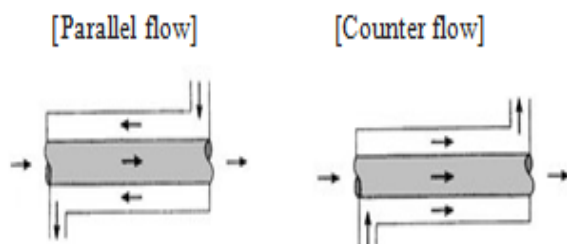
Department of Mechanical Engineering,
CDGI, Indore, MP, India
e-mail: vipul.jain@cdgi.edu.in

Abstract – Heat exchanger is a widely used for transfer of heat energy from one fluid to the other fluid. The heat is transferred in form of conduction and convection. Conduction occurs inside the material whereas convection occurs from materials to fluid and from fluid to fluid. The common application of the heat exchangers are like condenser, cooling tower, intercooler, refrigeration, and many other industrial applications. There are many types of heat exchangers wherein parallel and counter flow heat exchangers are extensively used in parallel flow heat exchanger hot and cold fluids are passes through the tubes in the same direction where as in counter flow heat exchanger both the fluids are passes in opposite direction to produce the desired effect. Baffles are sometimes used in heat exchanger to enhance heat transfer efficiency of heat exchangers. The concentric pipe heat exchanger is designed in present work in which different inlet velocities of fluid were taken and study is carried out for the temperature distribution, wall shear stress, and turbulent KE and pressure distribution for those velocities.

Keywords– Concentric pipe Heat Exchanger, Conduction, Convection, Parallel flow, Counter flow, Hot and cold inlets, pressure distribution, temperature distribution, Velocity inlet, wall shear stress, turbulent KE, ANSYS.

I. INTRODUCTION

A heat exchanger is a device that allows heat from a fluid (a liquid or a gas) to pass to a second fluid (another liquid or gas) without the two fluids having to mix together or come into direct contact. There are thus three heat transfer operations that need to be described: Convective heat transfer from fluid to the inner wall of the tube, Conductive heat transfer through the tube wall, and Convective heat transfer from the outer tube wall to the outside fluid. Figure 1.1 (a), the hot and cold fluids enter at the same end, flow in the same direction, and leave at the same end. In the counter-flow arrangement of Figure 1.1 (b), the fluids enter at opposite ends, flow in opposite directions, and leave at opposite ends.



Different types of heat exchanger:

1. Parallel-flow and counter-flow heat exchanger,
2. Finned and Un-finned tubular heat exchanger,

3. U-tube,
4. single pass straight and two pass straight heat exchanger,
5. Plate-and-frame heat exchanger,
6. Pate-fin heat exchanger,
7. Micro-channel heat exchanger.

Double Pipe Heat Exchanger Design with Counter flow or Parallel Flow:

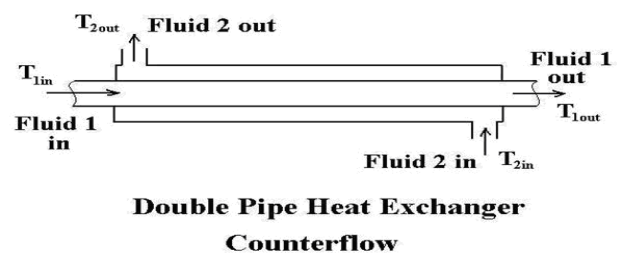


Fig.1. Concentric tube counter flow heat exchanger

II. LITERATURE REVIEW

Ibrahim, H. G. (2014) [1] Developed analytically the phenomenon of forced convection with turbulent flow of industrial processes. Pandey, P. K. [2] investigated the performance of the heat exchanger (tube bundle) using experimental observations and numerical techniques. Oclon P. [3] presents the numerical study on the influence of inner tube surface fouling on the thermal performance of a high temperature fin-and-tube heat exchanger.

Rutman, E [4] uses numerical modelling applications as a complement to the conventional experimental approach. De D [5] Constructed a coil heat exchanger with spiral deflector and the compared it with a right deflector with CFD analysis using ANSYS FLUENT software tools. Raj, R. T. K. [6] investigate the impacts of various baffle inclination angles on fluid flow and the heat transfer characteristics of a shell-and-tube heat exchanger for three different baffle inclination angles namely 0° , 10° , and 20° .

Ny, G. [7] provides a comprehensive overview of the use of Nano fluids to cool vehicle engines. Danook, S. [8] forced convection heat transfer through an elliptical inside a circular tube under turbulent flow by numerical simulation, with uniform heat flux boundary condition around circular tube has been studied.

III. GEOMETRIC MODELLING

In present work the geometry is created using CATIA V5R12 modelling software and the analysis is performed using ANSYS 14.0 analysis software. The figure 31. Shows the geometric model of heat exchanger, figure 3.2 represents the meshed model of the parts. Figure 3.3 represents the boundary hot and cold chamber inlet and exhaust zones. Figure 3.4 represents the contact region between two tubes. Figure 3.5 to figure 3.8 pressure, temperature, turbulence kinetic energy and wall shear maximum values for a boundary condition of 337 K-degree hot inlet temperature and 283 K-degree cold inlet temperature values. And considering average static pressure at the other end of the hot and cold chambers. The inlet velocity for both fluids are taken for comparing the results and these velocity values are 0.55 m/s, 0.6 m/s, 0.65 m/s and 0.7 m/s.

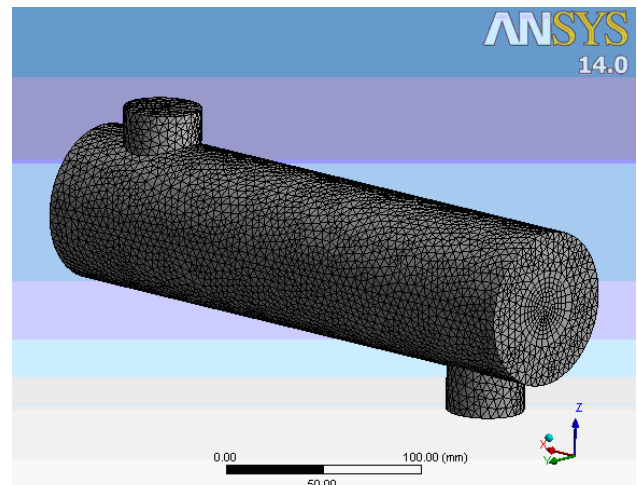


Fig.2. Meshed Model of Heat Exchanger

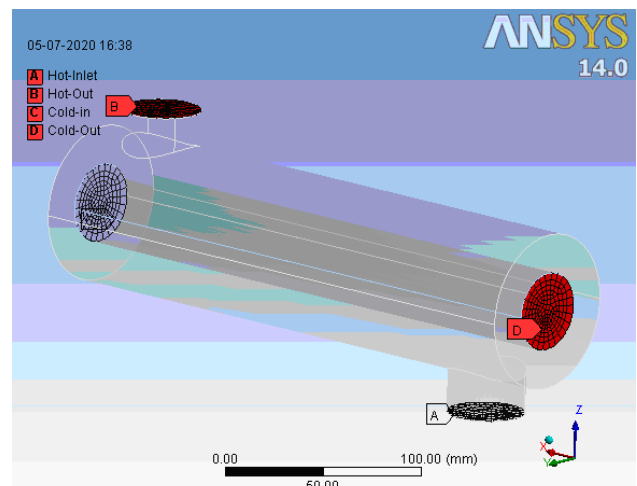


Fig.3. Named selection for providing the boundary conditions

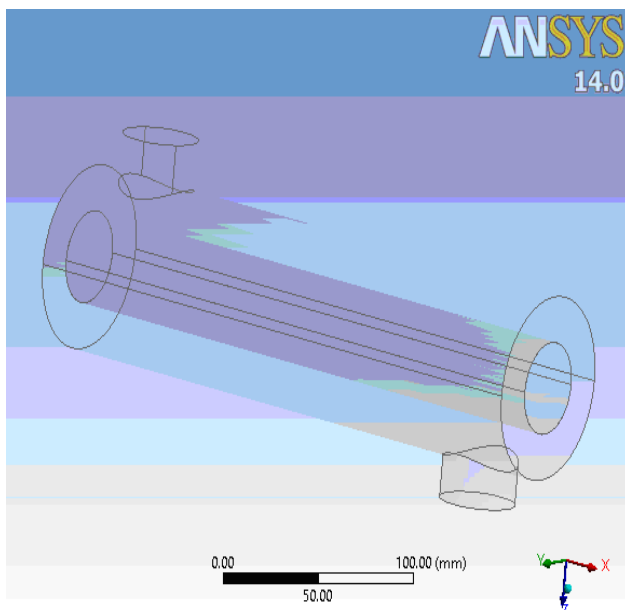


Fig.1. Geometry of Heat Exchanger

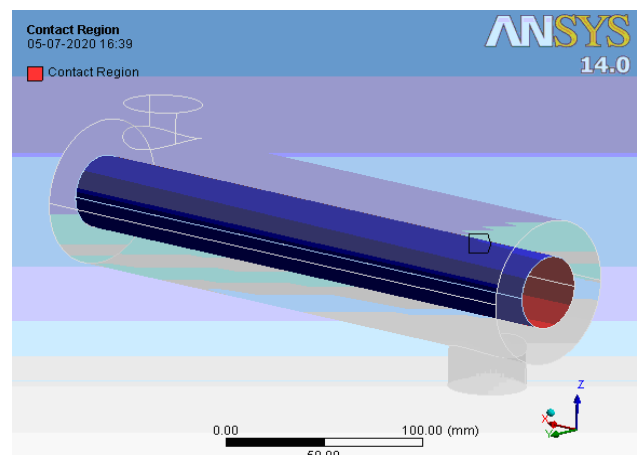


Fig.4. Contact region between hot and cold fluid chambers

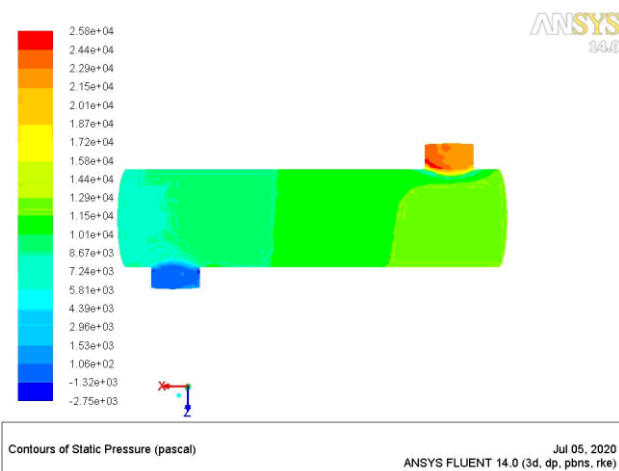


Fig.5. Pressure Distribution for inlet velocity flow rate of 0.7 m/s.

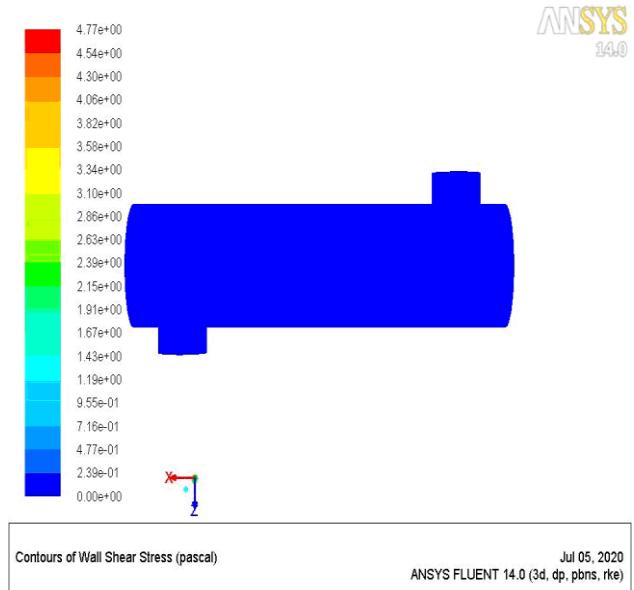


Fig.8. Wall Shear Stress Distribution for inlet velocity flow rate of 0.7 m/s

IV. RESULTS

Table –I: Pressure Distribution for inlet velocity flow rate

Sr. No	Inlet Velocity in m/s	Maximum Pressure in Pascal
1	0.7	2.58×10^4
2	0.65	2.63×10^4
3	0.6	2.66×10^4
4	0.55	2.95×10^4

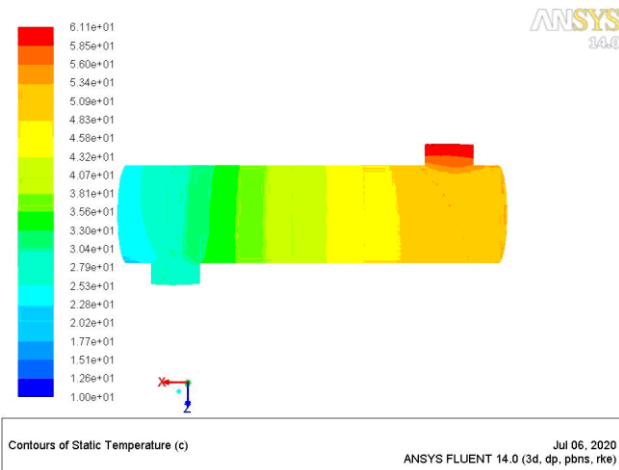


Fig.6. Temperature Distribution for inlet velocity flow rate of 0.75 m/s.

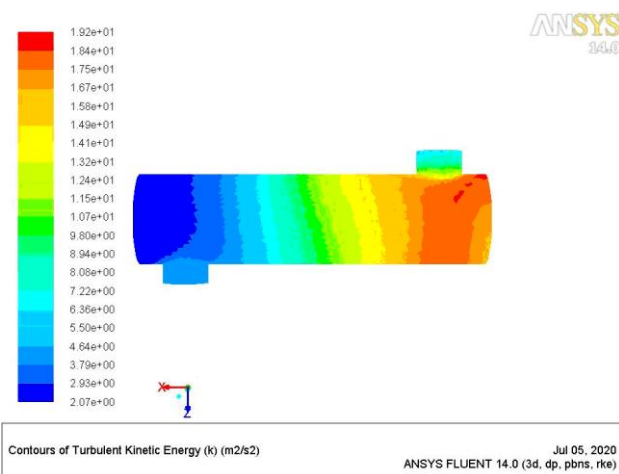


Fig.7. Turbulent K.E Distribution for inlet velocity flow rate of 0.7 m/s

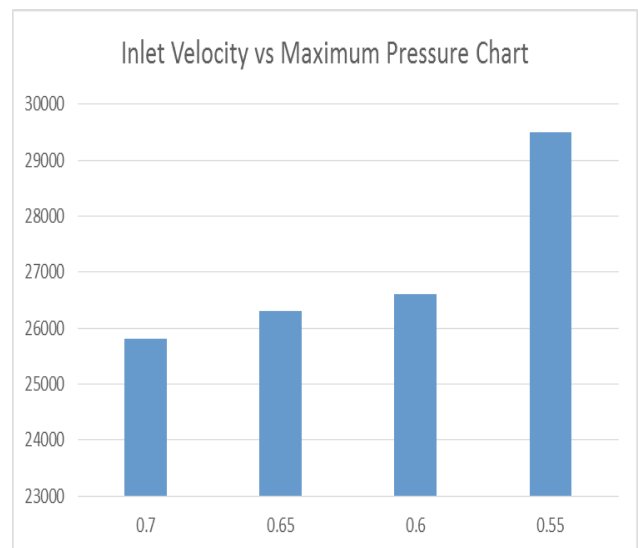


Fig.9. Temperature Distribution for inlet velocity flow rate.

Sr. No	Inlet Velocity in m/s	Maximum Temperature in °C
1	0.7	62
2	0.65	61.1
3	0.6	60.4
4	0.55	59.8

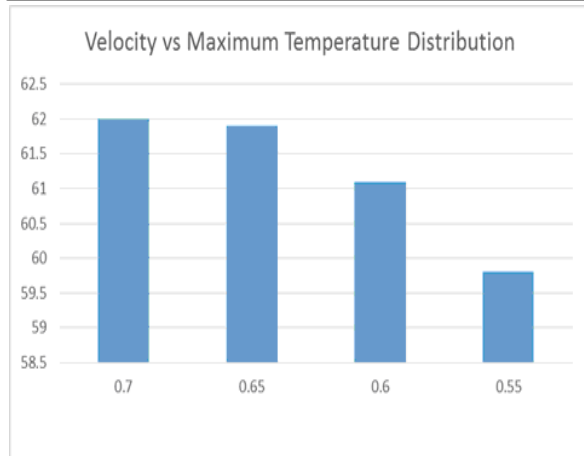


Fig.10. Turbulent K.E Distribution for inlet velocity flow rate of 0.7 m/s.

Sr. No	Inlet Velocity in m/s	Maximum Turbulent K.E in m^2/s^2
1	0.7	19.2
2	0.65	19.1
3	0.6	18.7
4	0.55	18.4

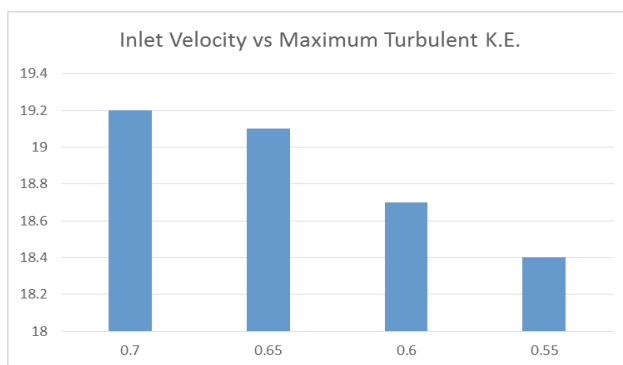
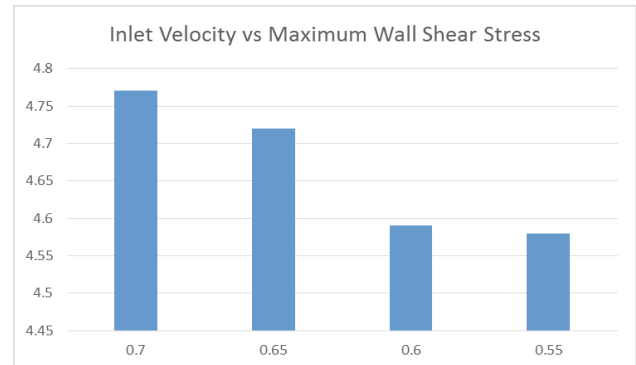


Table –II: Wall Shear Stress Distribution for inlet velocity flow rate of 0.7 m/s

Sr. No	Inlet Velocity in m/s	Maximum Wall Shear Stress in Pascal
1	0.7	4.77
2	0.65	4.72
3	0.6	4.59
4	0.55	4.58



IV. DISCUSSIONS

From the above research following discussions can be made:

1. With increase in inlet velocities of hot and cold fluid chambers the other parameters are also get affected and is reflected in our work.
2. Changing the inlet velocities drastically effects the temperature, pressure, and turbulent K.E and Wall shear stress values for a specific shift of inlet velocity so proper study is necessary before any variation of inlet velocity in actual applications.
3. It is found that the pressure drop is of the range between 30000 Pascal to 25800 Pascal for an increase of inlet velocity from 0.55 m/s to 0.7 m/s.
4. It is found that temperature rises from 59.8 to 62 degree for the increase in inlet velocity of 1.5 m/s.
5. It is observed that turbulent K.E of fluid increases from 18.4 to 19.2 m^2/s^2 for the increase in inlet velocity from 0.55 m/s to 0.7 m/s.
6. It is also observed that wall shear stress value increases from 4.58 Pascal to 4.77 Pascal for an increase in inlet velocity from 0.55 m/s to 0.7 m/s.

V. CONCLUSION

The following conclusions can be made from the present work:

1. Ansys 14.0 is an effective tool to analyse the heat transfer related problems and its fluent workbench is very helpful tool for fluid flow problems.
2. It can be stated that on increasing the inlet velocities of hot and cold fluid chambers pressure drops and the rest of the parameters are increases namely temperature, turbulence and wall shear stress values.
3. If the model is small then the velocity must be kept around 0.5 m/s or lower as we can see that on a small increase in velocity the heat transfer is not taking place effectively in heat exchanger applications. (In present case a length of 1m is taken for heat exchanger)
4. We can see on a small increase in inlet velocity from 0.6 m/s to 0.65 m/s adversely effects all the considered parameters and will be more significant where the length of heat exchanger is even larger.

5. With increase in temperature the turbulence K.E and wall shear stress vales also increases and is not a good sign for the heat exchanger.

VI. FUTURE SCOPE

1. The same analysis can be done with some different input parameter like changing the mass flow rate instead of velocity inlet.
2. In present analysis we have taken an average static pressure at outlet but in future one may also change the outlet pressure of the heat exchanger model.
3. In present work the inlet temperature of hot and cold fluid chambers are taken as 283 K and 335 K respectively and are small amount for simplicity purpose but in real applications these values may be more high and one can do that kind of analysis in future using even better configuration of computer.
4. Instead of using only concentric pipe heat exchanger one can also use several other types of heat exchanger too also one can use baffles inside the cold chamber to circulate the water for a longer duration for better heat exchange between the surfaces.
5. In present work cross flow heat exchanger is sued one can also use parallel flow heat exchanger by changing the direction of fluid flow inside the heat exchanger and then can see the effect of the same on output values.
6. Some other analysis software can also be used in future to compare and validate the results obtained here using ANSYS 14.0.

REFERENCES

- [1]. Florez-Orrego, D., Arias, W., Lopez, D., & Velasquez, H. (2012). Experimental and CFD study of a single phase cone-shaped helical coiled heat exchanger: an empirical correlation. Proceedings of ECOS, 375-394.
- [2]. Ibrahim, H. G. (2014). Experimental and CFD analysis of turbulent flow heat transfer in tubular exchanger. International Journal of Engineering, 5(07), 8269.
- [3]. Pandey, P. K., Lakhani, P. K., Kumar, K., Bohra, P., & Mishra, R. (2017). Heat Transfer Analysis of Shell and Tube Heat Exchanger using Al₂O₃/SiCNanofluid. European Journal of Advances in Enginee
- [4]. Oclon, P., kopata, S., Nowak, M., & Benim, A. C. (2015). Numerical study on the effect of inner tube fouling on the thermal performance of high-temperature fin-and-tube heat exchanger. Progress in Computational Fluid Dynamics, An International Journal, 15(5), 290-306.
- [5]. Rutman, E., Jurkowski, R., Bailly, A., Frankovic, B., & Vilicic, I. (2002, January). Industrial approach for CFD modelling applications for air-conditioning and heat exchanger systems. In Energija i okoliš 2002.

- [6]. De, D., Pal, T. K., & Bandyopadhyay, S. (2017). Helical baffle design in shell and tube type heat exchanger with CFD analysis. International Journal of Heat and Technology, 35(2), 378-383.
- [7]. Raj, R. T. K., & Ganne, S. (2012). Shell side numerical analysis of a shell and tube heat exchanger considering the effects of baffle inclination angle on fluid flow. Therm. Sci., 16(4), 1165-1174.
- [8]. Ny, G., Barom, N., Noraziman, S., & Yeow, S. (2016). Numerical study on turbulent-forced convective heat transfer of Ag/Heg water nanofluid in pipe. J. Adv. Res. Mater. Sci, 22(1), 11-27.



Jasvinder Singh Saluja
Research Scholar, Department of
Mechanical Engineering,
LNCTS (RIT) Indore, MP, India
E-mail:
jasvinder.saluja10@gmail.com