Numerical Investigation of Shell and Tube Heat Exchanger for Heat Transfer Optimization

Sahil Suman, Vivek Kumar Chaudhary, Ayush Raj, Kuldeep Rawat.

Abstract- An un-baffled shell-and-tube heat exchanger design with respect to heat transfer coefficient and pressure drop is investigated by numerically modelling. The flow and temperature fields inside the shell and tubes are resolved using a commercial CFD package considering the plane symmetry. For single shell and tube bundle is compared experimentally with CFD simulation. The results are found to be sensitive to turbulence model and wall treatment method. There are certain regions of low Reynolds number in the core of shell of heat exchanger. Thus, k-ω SST model, with low Reynolds correction, provides better results as compared to other models. The temperature and velocity profiles are examined in detail. It is seen that the flow remains parallel to the tubes thus limiting the heat transfer. Approximately, $2/3^{rd}$ of the shell side fluid is bypassing the tubes and contributing little to the overall heat transfer. Significant fraction of total shell side pressure drop is found at inlet and outlet regions. Due to the parallel flow and low mass flux in the core of heat exchanger, the tubes are not uniformly heated. Outer tubes fluid tends to leave at a higher temperature compared to inner tubes fluid. Higher heat flux is observed at shell's inlet due to two reasons. Firstly due to the cross-flow and secondly due to higher temperature difference between tubes and shell side fluid.

Keywords- Heat transfer, Shell-and-Tube Heat exchanger, CFD.

Introduction

Heat exchangers are one of the mostly used equipment's in the process industries. Heat process streams. One can realize their usage that any process which involves cooling, heating, condensation, boiling or evaporation will require a heat exchanger for these purposes. Process fluids, generally are heated or cooled before the process their undergo a phase change. Different heat exchangers are named according to their applications. For example, heat exchangers being used to condense are known as condensers; similarly heat exchangers for boiling purposes are called boilers. Performance and efficiency of heat exchangers are measured through the amount of heat transferred using least area of heat transfer and pressure drop. A better presentation of its

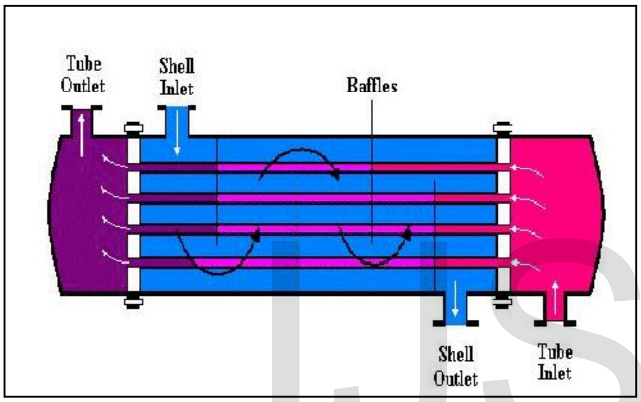
* *Author Sahil Suman currently pursuing B.tech degree program in department mechanical engineering in Shivalik College of Engineering, Dehradun, Uttrakhand, India. Ph- 9102934776, E-mail- sahilsuman20@gmail.com.*

* *Vivek Kumar Chaudhary currently pursuing B.tech degree program in department mechanical engineering in Shivalik College of Engineering, Dehradun, Uttrakhand, India. Ph- 7088932979.*

* *Ayush Raj currently pursuing B.tech degree program in department mechanical engineering in Shivalik College of Engineering, Dehradun, Uttrakhand, India. Ph- 976079603.*

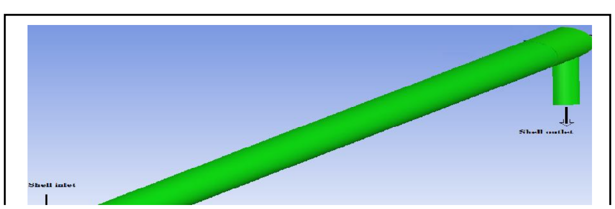
* *Kuldeep Rawat, Asst. Proff. at Shivalik College of Engineering Dehradun, Uttrakhand, India of mechanical engineering department. Ph- 9997974690*

efficiency is done by calculating over all heat transfer coefficient. Pressure drop and area required for a certain amount of heat transfer, provides an insight about the capital cost and power requirements (Running cost) of a heat exchanger. Generally, there is number of literature and theories to design a heat exchanger according to the demand. A good design is referred to a heat exchanger with least possible area and pressure drop to fulfil the heat transfer requirements.



1 Geometry

Heat exchanger geometry is built in the ANSYS workbench design module. Geometry is simplified by considering the plane symmetry and is cut half vertically. It is a counter current heat exchanger, and the tube side is built with 11 separate inlets comprising of 8 complete tubes and 3 half tubes considering the symmetry. The shell outlet length is also increased to facilitate the modelling program to avoid the reverse flow condition. In the Figure 3.1, the original geometry



along with the simplified geometry can be seen.

2.1 Mesh

This mesh contains mixed cells (Tetra and Hexahedral cells) having both triangular and quadrilateral faces at the boundaries. Care is taken to use structured cells (Hexahedral) as much as possible, for this reason the geometry is divided into several parts for using automatic methods available in the ANSYS meshing client. It is worked to reduce numerical diffusion mostly

	Boundary condition type	Shell	Tube
Inlet	Velocity-inlet	1.2 m/s	1.8 m/s
Outlet	Pressure-outlet	0	0
Wall	No slip condition	No heat flux	Coupled
Turbulence	Turbulence Intensity	3.6%	4%
Temperature	Inlet temperature	317	298
Mass flow rate		20000 kg/hr	20000 kg/hr

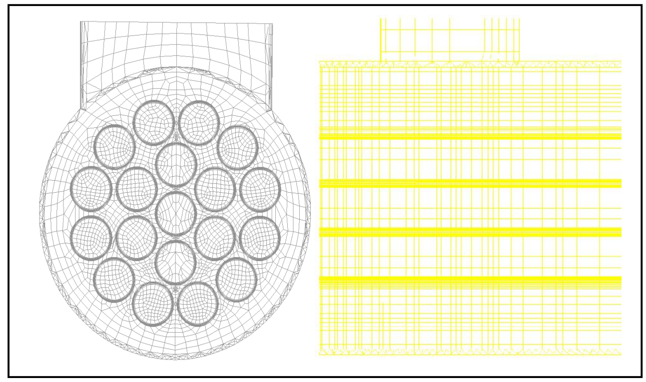
possible by structuring the mesh in a well manner, mainly near the wall region.

2.1.1 Y+ Values

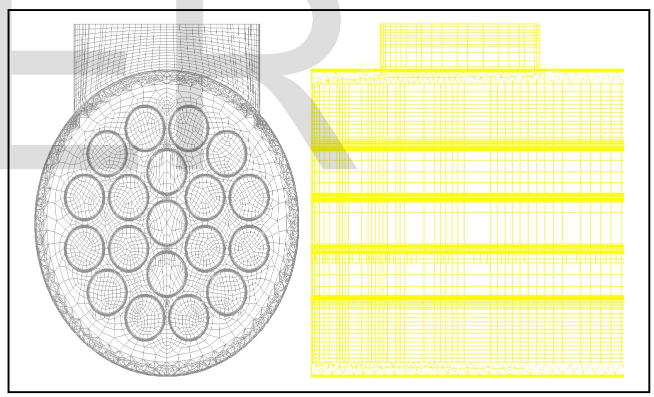
y+ values play an important role in turbulence modelling for the near wall treatment. For this reason, inflation on the walls is created to achieve the correct values of y+ values. Necessity for the y+ values for different wall treatments. It can be seen in the Table that y+ required for the standard and Non-equilibrium wall functions is high. The tubes inside the shell are very close to each other and thus have very little space in between. In order to resolve the boundary layer sufficiently, 10 to 15 cells are required between the adjacent tube walls. The inflation is thus kept very fine with first cell y+ < 5. This y+ condition puts a restriction thus limiting the use of standard and non-equilibrium wall functions. For all other walls of the heat exchanger, y+ values are set according to the wall treatment methods requirements. So when using Standard and Non-equilibrium wall functions y+ values are less than 5 at the tube walls and at all other walls are according to requirements mentioned in Table.

2.1.2 Grid independence

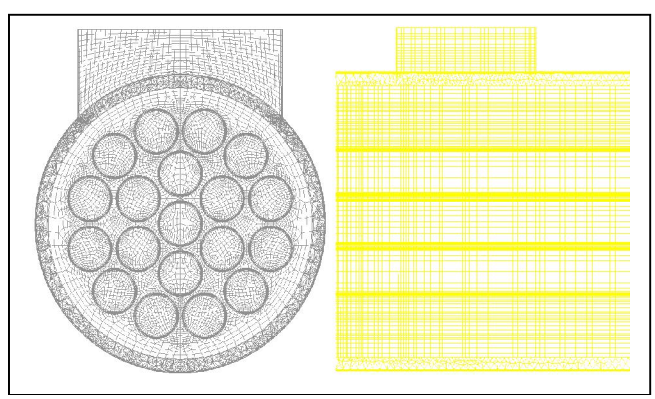
The contours from coarser mesh and fine mesh are analysed and it is noted that fine mesh resolves the region of high pressure and temperature gradients better as compared to coarse mesh. Thus taking care of these particular regions, coarse mesh is adapted to resolve these gradients. The criterions for adaption are temperature and pressure gradients. It is mainly refined in inlet and outlet regions to get the better estimations of pressure drop and heat transfer. Rapid mixing of hot and cold fluids is observed at the outlet, which led to refine the mesh further. Adoptions on the basis of temperature and pressure gradients are made to the mesh to get a fully grid independent model. Aspect ratio of the cells is kept same as coarse mesh because it is checked that the aspect ratio doesn’t affect much.



Coarse Mesh



Medium Mesh



3 Solution

3.1 Boundary conditions

Boundary conditions are used according to the need of the model. The inlet velocities and temperature are used similar to the experimental conditions in order to have a comparison. 11 tubes have 11 similar inlet and outlet boundary conditions. General correlations 3.1 and 3.2 are used to estimate the turbulence boundary conditions which are specified by estimating the turbulence intensity and length scale.

$I = 0.16Re^{-1/8}$

$l = 0.07L$

Later it is seen that the turbulence boundary conditions have a very little affect over the results and solution. The walls are separately specified with respective boundary conditions. 'No slip' condition is considered for each wall. Except the tube walls, each wall is set to zero heat flux condition. The tube walls are set to 'coupled' for transferring of heat between shell and tube side fluids. The details about all boundary conditions can be seen in the following table.

CFD comparison with experimental results

3.2 Discretization scheme

There are several discretization schemes to choose from. Initially every model is run with the first the second order upwind scheme. It is done to have better convergence but changed to higher order scheme to avoid the numerical diffusion. It

is seen that the flow is unidirectional in most of order upwind scheme and then later changed to

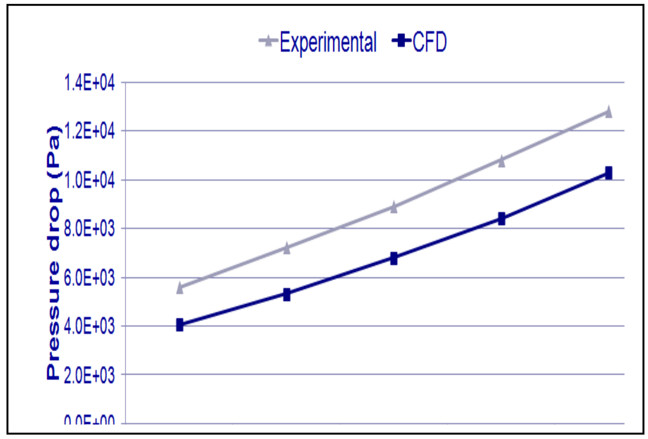


Figure: Comparison of Shell Side Pressure Drop

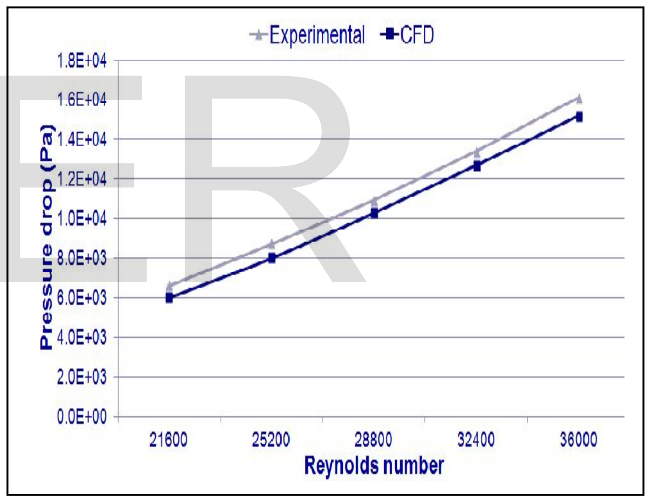


Figure: Comparison of Tube Side Pressure Drop

the second order upwind scheme. It is done to have better convergence but changed to higher order scheme to avoid the numerical diffusion. It is seen that the flow is unidirectional in most of the domain. So, it is recommended to use second order schemes for strong convection. The pressure drop in the shell is under-predicted through SST k-ω model which is almost 20-27%.This could be due to the several reasons including complicated geometry of the shell side and numerical diffusion. Whereas, the pressure drop in tube side (straight tubes) is predicted with an average error between 5-9%. It can be due to small baffles in the tubes used in the experimental setup. Overall heat transfer coefficient comparison with experiments can also be seen in the Figure. It is also been under-predicted by this model but still better than other models with an average error of 19-20%. The good thing about these results is the constant difference from experimental results and consistency with the real systems, i.e. with higher pressure drop, higher heat transfer is achieved.

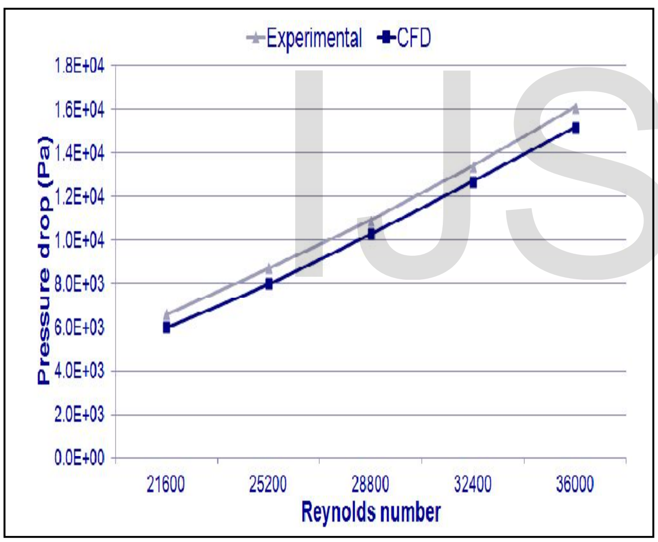


Figure: Comparison of Overall Heat Transfer Coefficient

4 Conclusion

The heat transfer and flow distribution is discussed in detail and proposed model is compared with the experimental results as well. The model predicts the heat transfer and pressure drop with an average error of 20%. Thus the model still can be improved. The prediction of inlet regions where the rapid mixing and change in the flow direction takes place. There is improvement is expected if complete geometry is modelled. sometimes, SST k-ω model has provided the reliable results given the y+ limitations, but this model over predicts the turbulence in regions with large normal strain (i.e. stagnation region at inlet of the shell). Thus the modelling can also be improved by using Reynolds Stress Models, but with higher computational costs. The enhanced wall functions are not used in this project due to convergence issues, but they can be very useful with k-ω models.

The heat transfer is found to be poor because the most of the shell side fluid by-passes the tube bundle without interaction. Thus the design can be modified in order to achieve the better heat transfer in two ways. Either, the shell diameter is reduced to keep the outer fluid mass flux lower or tube spacing can be increased to enhance the inner fluid mass flux. Just doing this might not be enough, because it is seen that the shell side fluid after 3m doesn't transfer heat efficiently.

It is because the heat transfer area is not utilized efficiently. Thus the design can further be improved by creating cross-flow regions in such a way that flow doesn't remain parallel to the tubes. It will allow the outer shell fluid to mix with the inner shell fluid and will automatically increase the heat transfer.

REFERENCES:

[1] "A heat exchanger applications general." http://www.engineersedge.com/

Heat exchanger/heat exchanger application.htm, 2011.

[2] Kakac and Liu, 1998. "Heat exchangers: Selection, rating, and thermal performance,"

[3] http://www.wcr-regasketing.com, "Heat exchanger applications." http://

www.wcr-regasketing.com/heat-exchanger-applications.htm, 2010.

[4] Kern,1950 Process Heat Transfer. McGraw-Hill,.

[5] Serth, Process Heat Transfer, Principles and Applications. Elsevier Science and Technology Books,.

[6] Ozden and Tari, "Shell side CFD analysis of a small shell-and-tube heat exchanger," Energy Conversion and Management, vol. 51, no. 5, pp. 1004 – 1014,.2014

[7] Gay, and Mackley,"Shell-side heat transfer in baffled cylindrical shell and tube exchangers- an electrochemical mass transfer modelling technique," Heat Mass Transfer, vol. 19, pp. 995–1002,.1976

[8] Gaddis, "Pressure drop on the shell side of shell-and-tube heat exchangers with segmental baffles," ChemEng Process, vol. 36, pp. 149–59,.1997

[9] Idelchik, I.E., Handbook of hydraulic resistance. Hemisphere Publishing, New York, NY, second edition.1986

[10] Van Der VyverandDirker ,"Validation of a cfd model of a three dimensional tube in- tube heat exchanger.,"2003.

[11] S. B., "Computational heat transfer in heat exchangers," Heat Transfer Eng, pp. 24:895–7,.2007

[12] Prithiviraj,"Shell and tube heat exchangers. part 1: foundation and fluid mechanics, "Heat Transfer, p. 33:799–816.,.1998

[13] Sha and, Yang Cl, "Multidimensional numerical modeling of heat exchangers., "Heat Transfer, vol. 104, pp. 417–25,.1982

[14] Bhutta, and Hayat, "CFD applications in various heat exchangers design: Areview," Applied Thermal Engineering, vol. 32, pp. 1–12,2011

[15] Jae-Young and Afshin, "Comparison of near-wall treatment methods for high Reynolds number backward-facing step flow," Int. J Computational Fluid Dynamics, vol. 19.2005

[1] "A heat exchanger applications general." http://www.engineersedge.com/

Heat exchanger/heat exchanger application.htm, 2011.

[2] Kakac and Liu, 1998. "Heat exchangers: Selection, rating, and thermal performance,"

[3] http://www.wcr-regasketing.com, "Heat exchanger applications." http://

www.wcr-regasketing.com/heat-exchanger-applications.htm, 2010.

[4] Kern,1950 Process Heat Transfer. McGraw-Hill,.

[5] Serth, Process Heat Transfer, Principles and Applications. Elsevier Science and Technology Books,.

[6] Ozden and Tari, "Shell side CFD analysis of a small shell-and-tube heat exchanger," Energy Conversion and Management, vol. 51, no. 5, pp. 1004 – 1014,.2014

[7] Gay, and Mackley,"Shell-side heat transfer in baffled cylindrical shell and tube exchangers- an electrochemical mass transfer modelling technique," Heat Mass Transfer, vol. 19, pp. 995–1002,.1976

[8] Gaddis, "Pressure drop on the shell side of shell-and-tube heat exchangers with segmental baffles," ChemEng Process, vol. 36, pp. 149–59,.1997

[9] Idelchik, I.E., Handbook of hydraulic resistance. Hemisphere Publishing, New York, NY, second edition.1986

[10] Van Der VyverandDirker ,"Validation of a cfd model of a three dimensional tube in- tube heat exchanger.,"2003.

[11] S. B., "Computational heat transfer in heat exchangers," Heat Transfer Eng, pp. 24:895–7,.2007

[12] Prithiviraj,"Shell and tube heat exchangers. part 1: foundation and fluid mechanics, "Heat Transfer, p. 33:799–816.,.1998

[13] Sha and, Yang CI, "Multidimensional numerical modeling of heat exchangers., "Heat Transfer, vol. 104, pp. 417–25,.1982

[14] Bhutta, and Hayat, "CFD applications in various heat exchangers design: Areview," Applied Thermal Engineering, vol. 32, pp. 1–12,2011

[15] Jae-Young and Afshin, "Comparison of near-wall treatment methods for high Reynolds number backward-facing step flow," Int. J Computational Fluid Dynamics, vol. 19.2005