CFD ANALYSIS OF HEAT TRANSFER FOR TUBE-IN-TUBE HEAT EXCHANGER

Deepa Shrivastava\textsuperscript{1} Rahul Mishra\textsuperscript{2}
\textsuperscript{1}Department of Mechanical Engineering, Shri Shankaracharya Engineering College, Bhilai
\textsuperscript{2}Assistant Professor, Department of Mechanical Engineering, Shri Shankaracharya Engineering College, Bhilai

Abstract: Computational Fluid Dynamics has been successfully used for predicting flow and heat transfer characteristics in heat exchangers. In the present work a 3-dimensional grid is generated for tube-in-tube heat exchanger using ANSYS. The continuity equation, the momentum equation and the energy equations are solved using ANSYS. CFD model is validated by comparing the results from experimentation, for both the parallel and counter flow. The CFD result for heat transfer found to be in good agreement with the calculated results from experimentation.

Key words: Reynolds Number, Friction factor, Heat transfer, Nusselt number, Computational fluid dynamics.

I. INTRODUCTION
Heat exchanger is use for exchanging heat between two fluid. A preferred approach to the problem of increasing heat exchanger efficiency, while maintaining minimum heat exchanger size and operational cost, is to increase heat exchanger rate. Experimental analysis of the heat exchanger is a difficult task for the researchers, as it is costly and time consuming. Also, does not give complete idea about velocity, temperature and pressure variation with in the heat exchangers. To predict variation of fluid flow properties and effectiveness of heat exchangers, there is necessity of computational fluid dynamics (CFD) simulation.

Heat exchanger is process equipment designed for the effective transfer of heat energy between two fluids; a hot fluid and cooled fluid. The purpose may be either to remove heat from a fluid to add heat to the fluid. Notable example are
\begin{itemize}
  \item Boilers (evaporator), super heaters and condensers of a power plant
  \item Automobile radiators and oil coolers of heat engines
  \item Evaporator of an ice plant and milk-chiller of a pasteurizing plant
  \item Water and air heaters or coolers
\end{itemize}

The heat transferred in the exchanger may be in the form of latent heat (e.g. in boilers and condensers) or sensible heat (e.g. in heaters and coolers).

Figure represents the block diagram of a heat exchanger.

\textbf{Block diagram of heat exchanger}
II. LITERATURE REVIEW

I. Van Der Vyver, Dirker and Meyer (2006), reported the “Validation of a CFD model of a three dimensional tube-in-tube heat exchanger” shows investigation on a tube in tube heat exchanger with hot water (single phase) flowing in the inner tube and cold water in tube annulus. The heat exchanger was numerically modelled in 3-Dimensional computational fluid dynamics (CFD). The heat transfer coefficient and friction factors were determined with CFD and compared to established correlations. The second part of the study investigated the CFD’s ability to model a prototype configuration of a tube-in-tube exchanger. The numerical data was compared with analytical predictions and experimental results.

II. Zavala-Rio, and Santiesteban-Cos (2006), studied the “Reliable compartmental models for double-pipe heat exchangers: An analytic study’. In this work analytical properties of the heat exchanger second order lumped parameter dynamic model using the logarithmic mean temperature difference (LMTD) as driving were derived. Second order lumped parameter models with LMTD driving force are reliable dynamic representations for heat exchangers. This is especially important in case where it is not the quantitative solutions but the qualitative behaviour that is important, like modelling and simulation of heat exchanger networks and complex industrial processes where heat exchangers are involved.

III. GOVERNING EQUATION

Heat transfer equation
\[ q = mC_p(t_{h2} - t_{h1}) \]

Reynolds Number
\[ Re = \frac{\rho V D}{\mu} \]

Nusselt Number
\[ Nu = \frac{h_d}{h_k} \quad Nu = 0.023 \left( Re^{0.8} Pr \right)^n \]

Friction factor
\[ f = \frac{2d_k \Delta p}{\rho L v^2}, \quad f = (1.82 \log_{10} Re - 1.64)^{-2} \]

IV. PROBLEM DESCRIPTION

The solution of fluid flow governing equation is solved by numerical methods like finite volume, finite difference etc. which require to divide the flow domain into discrete element. The procedure to divide the flow domain into discrete element is called as grid or mesh generation. Mesh or grid generation consist of creating a set of grid points along the boundaries and throughout the domain of interest. The grids commonly generated are structured and unstructured. The unstructured grids provides easy way of handling complex geometries. Local mesh refinement, either adoptive or fixed, is another advantage of unstructured methods. Algorithms for managing data structure are vital part of unstructured grid analysis because they lack structured-ness, meaning that there is no definite order or numbering possible of neighboring nodes. Therefore solution of governing equation using unstructured grid is time consuming and difficult.

The grid generation using structured grid for simple geometries is quite easy and solving of governing equations is also simple. The structured grid solvers lead to the simplest algorithms with the fewest operations and smallest memory per grid point, but there are serious difficulties in their application to complex curved surfaces. While the grid exhibits much of the same regularity that makes structured grids attractive, the grid is not tied to the structure itself; this can be an advantage when building the grid, but few features that are caused by structural features may not be well resolved.

V. METHODOLOGY

The heat exchanger model is created in ANSYS software package which is designed to help analysis and designers to build and mesh models for computational fluid dynamics (CFD) and other scientific
applications. A structured mesh is applied to the computational domain with a refined mesh density near the wall boundary; the mesh is composed primarily of hexahedral elements but includes wedges elements where appropriate.

Volume grid of tube-in-tube heat exchanger

VI. RESULT

The results obtained from the CFD modelling shows the effect of varying independently the flow rates of cold and hot fluid streams on the heat transfer through the value of Nusselt number, Reynolds number, Friction factor and outlet temperature of both the streams for different case. The effect is also analysed by using the Numerical model and by the experimentations. The results obtained from the numerical model and the experimentations show a good agreement with that obtained from CFD modelling, thus validating the results of CFD modelling. Show the agreement between the results obtained (Nusselt Number Vs Reynolds Number; Friction factor Vs Reynolds Number) in fig 1,2,3 and 4.

Results For Counter Flow Condition (CFD Vs Experimentation)

Figure 1 Nu Vs Re
Figure 2 $f$ Vs $Re$

Results For Parallel Flow Condition (CFD Vs Experimentation)

Figure 3 $Nu$ Vs $Re$
VII. CONCLUSION

The CFD based simulation carried out for Tube-in-Tube heat exchanger using ANSYS software. The CFD results for heat transfer characteristics (Nusselt number) are compared with result from the experimentation. The results are presented in graphical form and they are found to be in good agreement with each other.

For flow characteristics, the pressure drop and the friction factor from the CFD simulation is compared with friction factor. The results are presented in graphical format and they are found to be in good agreement with each other.

VIII. FUTURE SCOPE

In the present work the heat transfer characteristics i.e. the Nusselt number and the flow characteristic i.e. the friction factor from the CFD results is compared with the CFD results is compared with the results from the co-relations for Nu and f for both inner tube and annulus, for various values of the Reynold’s number. The inlet temperatures of hot and cold fluids are kept constant.

In future the inlet temperatures $T_{hi}$ and $T_{ci}$ can be varied and their effect on the heat transfer and flow characteristic can be studied.

ACKNOWLEDGEMENT

We feel highly elevated in expressing our immense sense of gratitude towards our project guide for his most esteemed guidance, prompt suggestion and the interest towards the completion of the project entitled “CFD Analysis of Heat Transfer for Tube-in-Tube Heat Exchanger”. It was especially valuable asset for us to have him for the guidance of the project as he always filled us with ideas and suggestions, also responding in the best manner whenever we were in distress.

REFERENCES

I. Hilder Van Der Vyer, Jaco Dirker and Josua P. Meyer(2006), “Validation of a CFD model of a three dimensioninal tube-in-tube heat exchanger”

II. Sadik Kakac Hong tan Liu(1997), “Heat exchangers Selection, Rating and Thermal Design”

III. L.Redjem-saad, M.Ould-Rouiss, G.Lauriat (April 2007), “Direct Numerical Simulation of Turbulent heat Transfer in pipe flow: Effect of Parantle number”
IV. M.T. Madhav and M.R. Malin (1997), “The Numerical Simulation of Fully Developed Duct Flow.”

V. Zhengguo Zhang, Dabin Ma, Xiaoming Fang, Xuenong Gao, “Experimental and Numerical Heat Transfer in a Helically Baffled Heat Exchanger Combined with One Three-Dimensional Finned Tube”.

VI. Mourad Yataghene, Jeremy Pruvost, Francine Fayolle, Jack Legrand (July 2007) “CFD Analysis of the flow pattern And local Shear Rate in a Scraped Surface Heat Exchanger”

VII. Yunus Cengel, “Heat Transfer”.

VIII. D.S. Kumar “Heat and Mass Transfer”.