IMEETMCon

2014

International Conference of Mechanical Engineering, Energy Technology and Management

3rd – 5th September, 2014

Conference Centre, University of Ibadan, Ibadan, Nigeria

NUMERICAL ANALYSIS OF SHELL AND TUBE HEAT EXCHANGER WITH CONVEX-CUT BAFFLES

M.O. Petinrin[1,2] and A.A.Dare[1]

[1]Department of Mechanical Engineering, University of Ibadan, Ibadan, Oyo State

[2]Prototype Engineering Development Institute, Ilesa, Osun State

E-mail: layopet01@yahoo.com, ademola.dare@mail.ui.edu.ng

Abstract

In this work, 3D numerical simulation for shell and tube heat exchanger is carried out with a commercial CFD code, COMSOL Multiphysics when utilizes the continuity, momentum and energy equations. It was discovered that much of the heat transfer was due to the cross flow at the tube bundles. The pressure drop increases rapidly while trying to increase the heat transfer rate by increasing the flow rate and this was in conformity with facts from literature. Performance of two shell and tube heat exchanger models were predicted, one with standard single-segmental baffles, STHE_SS (25% standard cut) and other with convex cut baffle, STHE_CS (30% convex cut). The STHE_SS has higher heat transfer rate and higher pressure drop than the STHE_CS for range of Reynolds number studied, a better performance ratio from Re ≈ 4600 over STHE_SS was recorded for STHE_CS

Keywords: Shell, Tube heat exchanger, heat transfer, Convex cut battles and Numerical Analysis

1.0 Introduction

One of the important processes in engineering is the heat exchange between flowing fluids, and many types of heat exchangers are employed in various types of installations, as process industries, petrochemical plants, nuclear power stations, pressurized water reactor power plants, building heating, ventilating, and air-conditioning and refrigeration systems (Raj and Ganne, 2012).

The most commonly used type of heat exchanger is the shell-and-tube heat exchanger (STHE). STHE provide relatively large ratio of heat transfer area to volume and weight and they can be easily cleaned. Shell and tube heat exchanger offer great flexibility to meet almost any service requirement. Although they are not specially compact, their robustness and shape make them well suited for high pressure operations (Leong and Toh, 1998).

The shell-and-tube heat exchangers are still the most common type in use. They have larger heat transfer surface area-to-volume ratios than the most of common types of heat exchangers, and they are manufactured easily for a large variety of sizes and flow configurations. It can be designed for high pressure relative to the environment and high pressure difference between the fluid streams (Patel and Mavani, 2012). The shell-and-tube heat exchangers consist of a bundle of tubes enclosed within a cylindrical shell. One fluid flows through the tubes and a second fluid flows within the space between the tubes and the shell. For this particular study E-shell is considered, which is generally a one pass shell. E shell is the most commonly used due to its low cost and simplicity, and has the highest log-mean temperature-difference (LMTD) correction factor (Raj and Ganne, 2012).

Baffles are used to support the tubes for structural rigidity, preventing tube vibration and sagging and to divert the flow across the bundle to obtain a higher heat transfer coefficient (Patel and Mavani, 2012). Baffles are used to support tubes, enable a desirable velocity to be maintained for the shell-side fluid, and prevent failure of tubes due to flow-induced vibration. Baffle can be categorized as TEMA baffle: single-segmental, double-segmental, or triple-segmental; and Non-TEMA baffles: Helical, Disc-and-Donut, Grid and Rod Type (Bouhairie, 2012)

An effective design for a heat exchanger is the one which maximizes the heat transfer while reducing the power expended (Nithiarasu, 2005). The configuration of baffles affects the performance and efficiency having selected the right shell, and tube layout and the best baffle spacing for STHE design. Performance and efficiency of STHEs are measured through the amount of heat transfer using least area of heat transfer and pressure drop (Roll, 2013).

In this study, 3D simulation of STHE with new set of baffles named convex-cut baffles (STHE_CS) and the standard single-segmental baffles (STHE_SS) will be carried out. These baffles are alterations in shape of the single-segmental baffles. The STHE_SS has attribute of high heat transfer rate but with high pressure drop, which requires higher pumping power to offset (Wang *et al*, 2008), therefore, it is of great importance to carry out performance analyses on both STHEs.

2.0 Shell and Tube Heat Exchanger Model

2.1 Physical Model

The model is designed according to Tubular Exchanger Manufacturers Association (TEMA) Standards (Kuppan, 2000; Shah and sekulic, 2003), and the geometric parameters of the STHE model is as listed in Table 1. The dimension details of the convex-cut baffles adapted from 25% baffle cut, which is most used baffle cut for STHE_SS (Kara and Guraras, 2004), is shown in Figure 1, H and h are 25% and 30% of the baffle diameter respectively. The radius, R and cut-out area, which is equal to that of STHE_SS baffle were computed from an iteration in Scilab Programming Language. The CAD model is shown in Figure 2.

It is assumed that the heat exchanger is well-insulated hence the heat loss to the environment is totally neglected, also the leak flows between tube and baffle and that between baffle and the shell are neglected while the thickness of the tube material is considered to be negligible.

Table 1: The geometric parameters of the Shell and Tube Heat Exchanger

Shell-side Parameter	Shell Diameter	108.06mm
	Inlet and Outlet Diameter	30mm
Tube Parameter	Tube Diameter, *d*	15.88mm
	Layout Pattern	Triangular (30^0)
	Pitch	1.25*d*
	Number of Tubes	19
Baffle Parameter	Number of Baffles	6
	Baffle Spacing	43.26mm

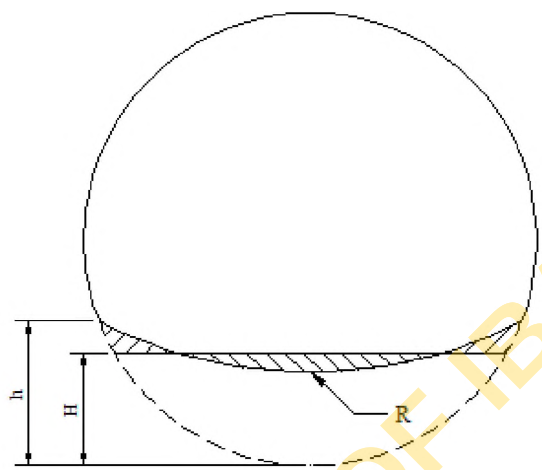

Figure 1: Convex-cut baffle for STHE_CS

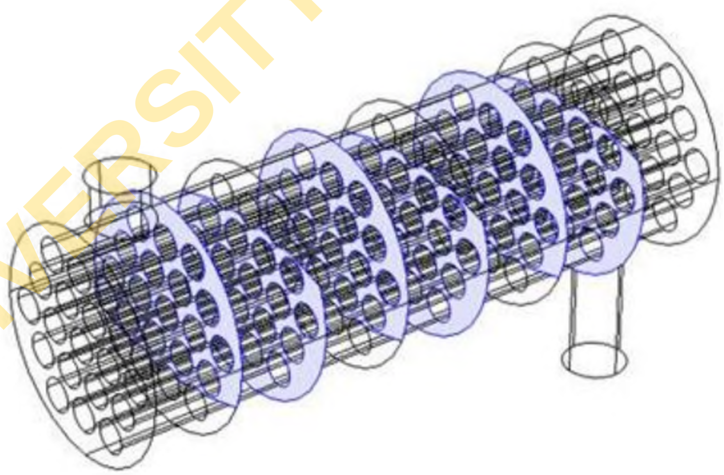

Figure 2: The STHE_CS CAD model showing the convex-cut baffles

2.2 Governing Equations and Numerical Methods

In the computational analysis of the flow, the three-dimensional RANS $k-\varepsilon$ turbulence model was used to describe the fluid flow and heat transfer in the STHE using the commercial CFD code, COMSOL Multiphysics. The governing equations are given in equations (1) to (5) (Ferziger, 2002) as:

Continuity Equation

$$\nabla.(\rho u) = 0 \tag{1}$$

Momentum Equation

$$\rho u.\nabla u = \nabla.\left[-pI + (\mu + \mu_T)\left(\nabla u + (\nabla u)^T\right) - \left(\frac{2}{3}\right)(\nabla.u)\boldsymbol{I}\right) - \left(\frac{2}{3}\right)\rho k\boldsymbol{I}\right] + F \tag{2}$$

Turbulent Kinetic Energy Equation

$$\rho u.\nabla k = \nabla.\left[\left(\mu + \frac{\mu_T}{\sigma_k}\right)\nabla k\right] + \mu_T P(u) - \left(\frac{2\rho k}{3}\right)\nabla.u - \rho\varepsilon \tag{3}$$

Turbulent Energy Dissipation Equation

$$\rho u.\nabla\varepsilon = \nabla.\left[\left(\mu + \frac{\mu_T}{\sigma_\varepsilon}\right)\nabla\varepsilon\right] + \left(C_{\varepsilon 1}\frac{\varepsilon}{k}\right)\left[\mu_T P(u) - \left(\frac{2\rho k}{3}\right)\nabla.u\right] - C_{\varepsilon 2}\rho\frac{\varepsilon^2}{k} \tag{4}$$

Energy Equation

$$\rho c_p u.\nabla T = \nabla.(k\nabla T) + Q \tag{5}$$

where $P(u) = \nabla u{:}\,(\nabla u + (\nabla u)^T) - \left(\frac{2}{3}\right)(\nabla.u)^2$, and $\mu_T = \rho C_\mu \frac{k^2}{k}$

and the constants of the $k-\varepsilon$ turbulence model are given as:

$c_\mu = 0.09,\ c_{\varepsilon 1} = 1.44,\ c_{\varepsilon 2} = 1.92, \sigma_k = 1.0, \sigma_\varepsilon = 1.3$

For the boundary conditions applied, the velocity-inlet and pressure, no viscous stress boundary conditions are applied on the inlet and outlet respectively for the shell-side and tube-side. The standard wall function is applied on all the walls of the STHE within the computational domain. The shell wall of heat exchanger is well insulated against heat transfer. The thermal properties of the working fluids, water and engine oil in tube-side and shell-side respectively, vary with changes in temperature. The variations with temperature were found in Incropera and Dewitt (2005), and Keith and Bohn (1993). The tube-side fluid inlet temperature was fixed at 273K while the shell-side inlet temperature was at 373K.

The 3D computational domain was discretized with unstructured tetrahedral elements. The set of equations were solved with the finite element based commercial CFD code, COMSOL Multiphysics, using segregated solvers: two iterative solvers, GMRES with Incomplete LU as preconditioner for velocity and pressure, and temperature respectively; and one direct solver for the turbulent kinetic energy and turbulent energy dissipation (COMSOL AB, 2012).

3.0 Results and discussion

The velocity distribution for the tube-side of the STHE_CS with mass flow rate of 0.3kg/s (*Re* ≈ 30000) is as shown in the cut-plane section in Figure 3. The fluid velocity increases the inlet (right) to the outlet (left) of the tubes, this is due to the heating-up from the shell-side reducing the density while maintaining the mass flow rate to increase the velocity. The active zones are the portions with high velocity in each shell zones and this is caused by the cross-flow heating between the baffles from the shell-side.

The shell-side velocity streamlines are shown in Figure 4. It can be clearly observed that the fluid passes through the baffle windows to create counter-flow and over the tube bundles for cross-flow heat transfer. The fluid velocity reduces towards the shell outlet as indicated by the streamlines density and the legend shows, by colour, the temperature distribution.

In Figure 5, the cut-plane temperature distribution through the shell-side decreases as the fluid crosses over the tube bundles because its heat lost the tube-side fluid

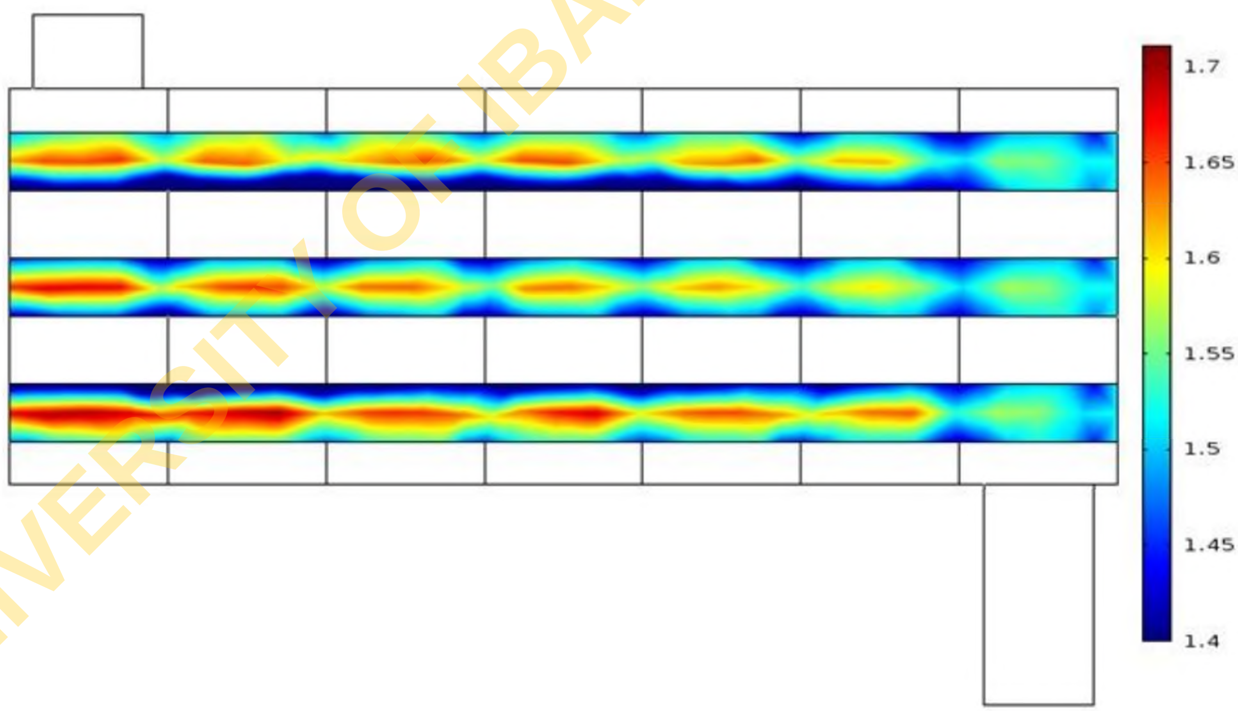

Figure 3: Velocity distribution in the tube-side through the cut-plane section at 0.3kg/s (Re≈30000)

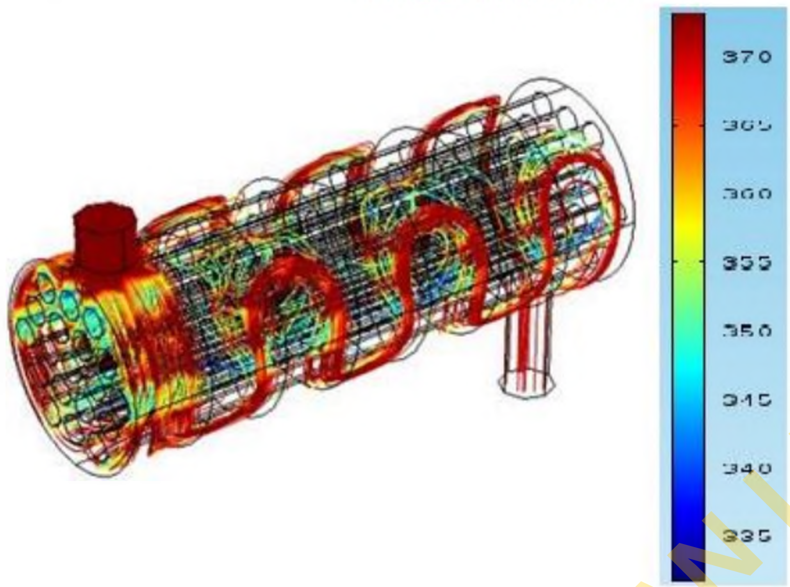

Figure 4: Shell-side velocity streamlines with temperature colouration in the STHE_CS

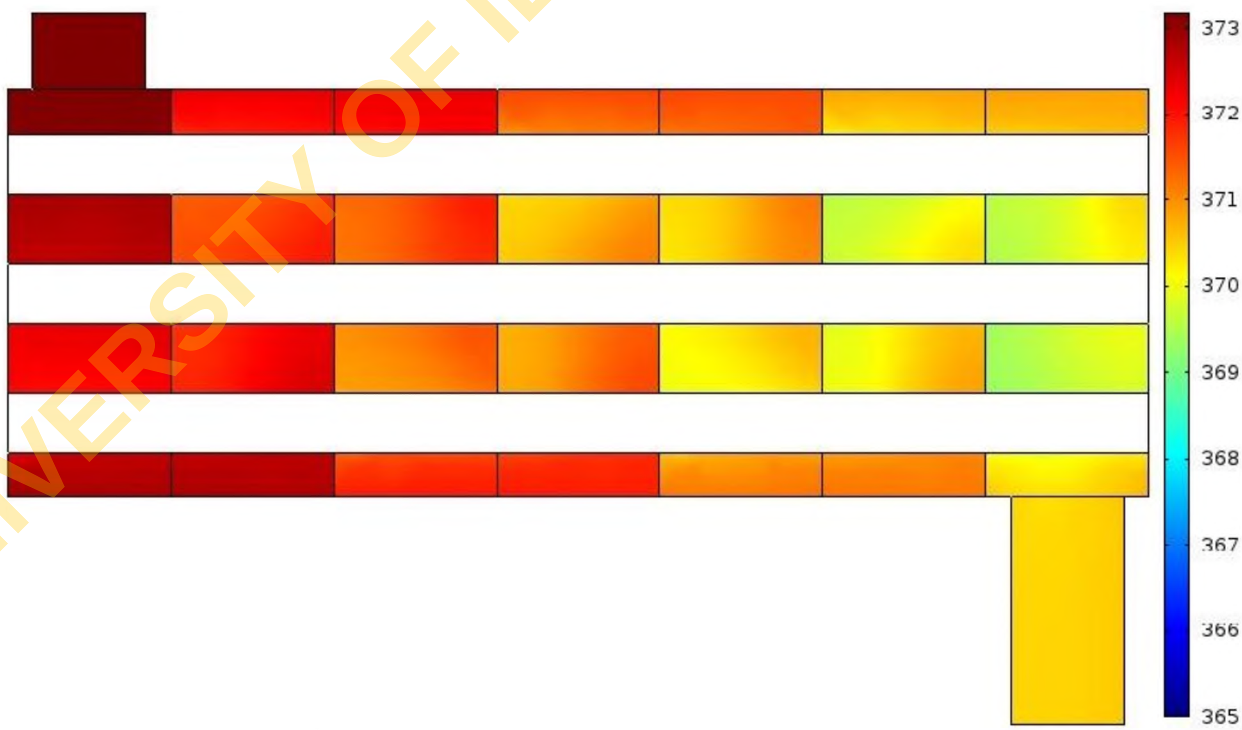

Figure 5: Temperature distribution in the shell-side through the cut-plane section

Figure 6 shows the shell-side heat transfer coefficient plot as a function of the Reynolds number with corresponding mass flow rate from 0.1kg/s to 3.1kg/s at a step of 0.5. As discussed by Bouhairie (2012), that STHE_SS (25% standard cut in this case) has high heat transfer rate, for varying Reynolds number the coefficient of heat transfer is higher than that of STHE_CS (30% convex cut). Also from Figure 7, the shell-side pressure drop through STHE_SS is higher than that of STHE_CS for the same range of Reynolds numbers. Again from Figures 6 and 7, as stated by Mukherjee (1998), there are indications that the pressure drop increases rapidly with Reynolds number as against less increase recorded for the heat transfer coefficient. This means gaining heat transfer as mass flow rate increases at the cost of higher pressure drop.

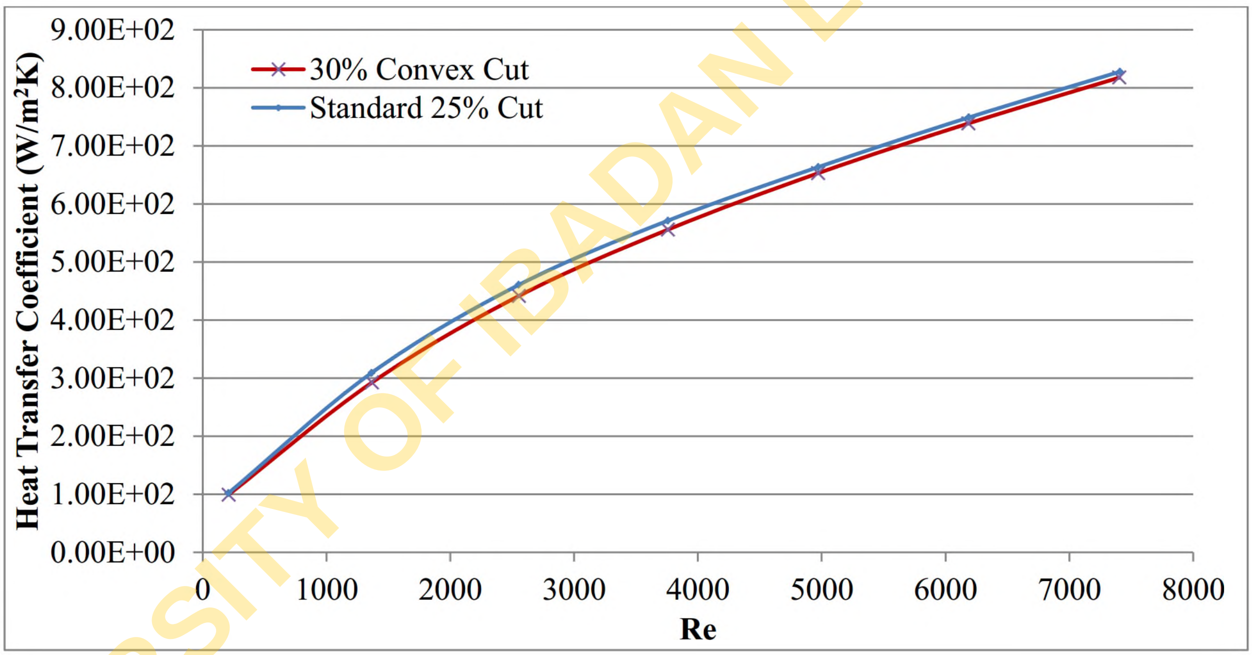

Figure 6: Heat transfer coefficient as a function of Reynolds number

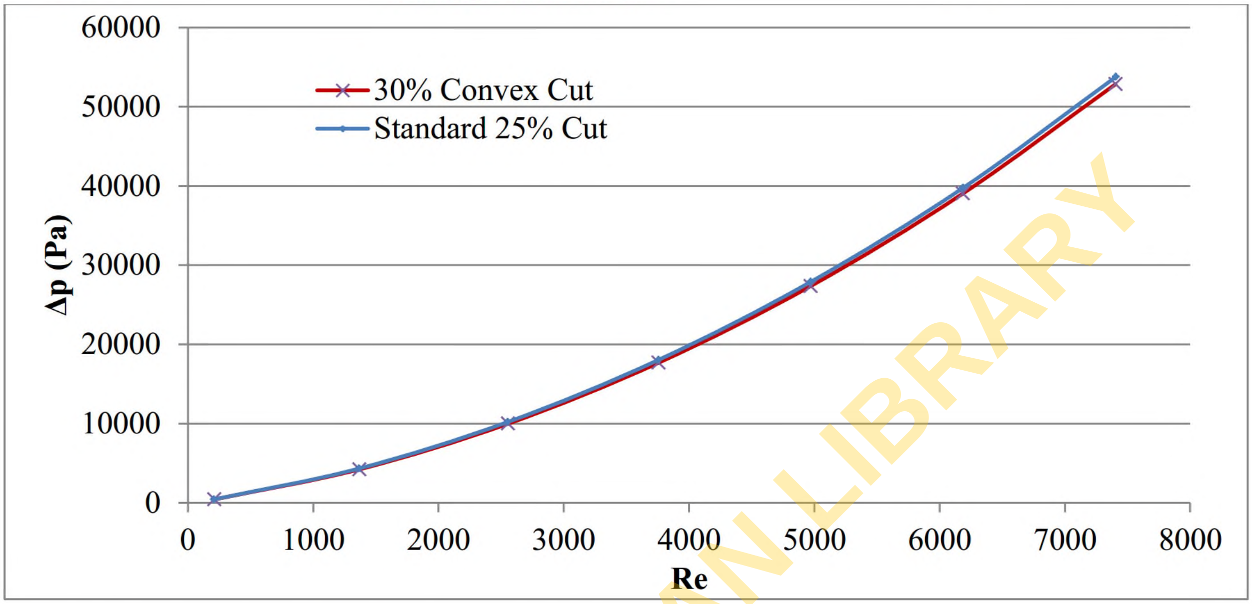

Figure 7: Pressure drop as a function of Reynolds number

To take assessment of the STHEs, a shell gain factor is introduced as follows (Mohammadi, 2011):

$$\Gamma = \frac{h}{\Delta p} \qquad (6)$$

where h is the coefficient of heat transfer and Δp is the pressure drop.

Comparison between the STHE_SS and STHE_CS gives a performance factor which is defined as:

$$\Phi = \frac{\Gamma_{CS}}{\Gamma_{SS}} \qquad (7)$$

The subscripts CS and SS in equation (7) are for the STHE_CS and STHE_SS respectively.

From Figure 8, a performance factor greater than one for indicates that STHE_CS (30% convex cut) is more desirable at that Reynolds number than STHE_SS (25% standard cut) while it is less desirable when the performance factor is less than one. Performance factor increases as the Reynolds number increases from $Re \approx 4600$ and this trend of performance factor seems approaching a constant with the increase in Reynolds number.

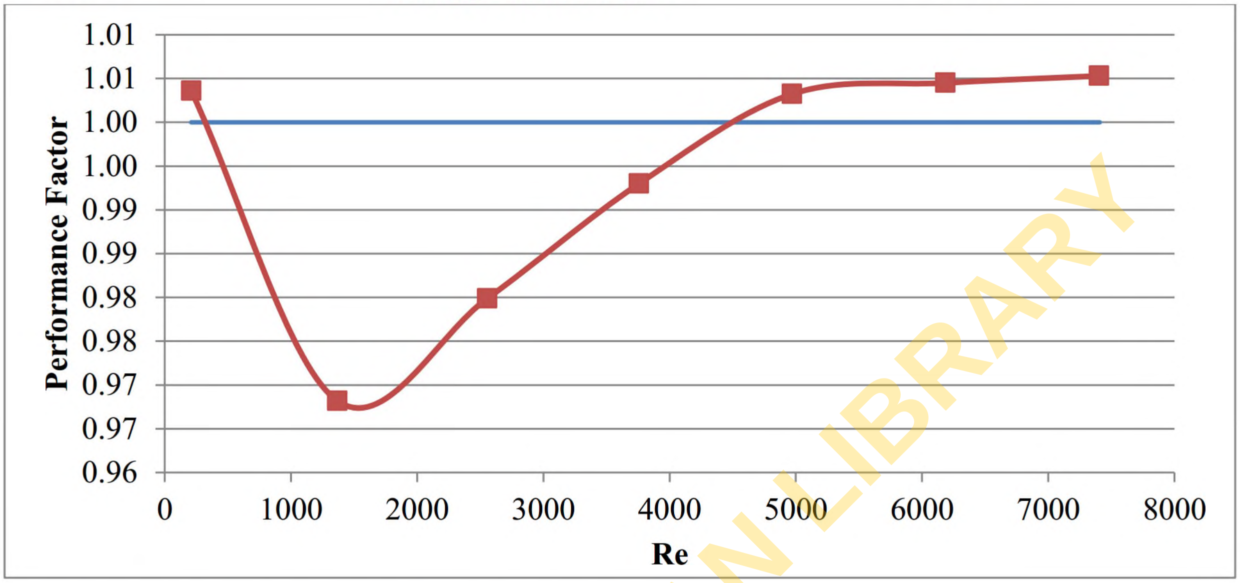

Figure 8: The performance factor as a function of Reynolds number

4.0 Conclusion

In this work, numerical model for shell and tube heat exchanger was developed and analysed using a commercial CFD code, COMSOL Multiphysics. The results from the numerically analyses showed that much of the heat exchange is carried out through the cross flow across the tube bundles. The pressure drop increases rapidly while trying to increase the heat transfer rate by increasing the flow rate, and this was in conformity with facts from literature. Two heat exchanger models were investigated, the STHE_SS (25% standard cut) has higher heat transfer rate and higher pressure drop than the STHE_CS (30% convex cut), and the STHE_CS only showed a better performance ratio from Re ≈ 4600.

References

Bouhairie, S. (2012), "Selecting Baffles for Shell and Tube Heat Exchangers", Heat Transfer Research, Inc., *American Institute of Chemical Engineers (AIChE)*

COMSOL AB (2012), COMSOL Multiphysics Documentation

Ferziger, J. H. and Peric, M. (2002), *Computational Methods for Fluid Dynamics*", 3rd rev. ed., Springer-Verlag, Berlin

Incropera, F. P. and Dewitt, D. P. (2005), *Fundamentals of Heat and Mass Tranfer*, 5th edition, John Wiley & Sons, New York

Kreith, F. and Bohn, M. S. (1993), *Principles of Heat Transfer*, 5th edition, West Publishing Company, St. Paul Kuppan, T. (2000), *Heat Exchanger Design Handbook*, New York, Marcel Dekker, Inc.

Leong, K. C. and Toh, K. C. (1998), "Shell and Tube Heat Exchanger Design Software for Educational Applications", vol. 14, No. 3, pp. 217-224, *Int. J. Engng Ed*.

Mohammadi, K. (2011), "Investigation of the Effects of Baffle Orientation, Baffle Cut and Fluid Viscosity on Shell Side Pressure Drop and Heat Transfer Coefficient in an E-Type Shell and Tube Heat Exchanger", Unpublished D.EngSc. Thesis, Energy Technology, Process Engineering and Biological Engineering, Institute for Thermodynamics and Thermal Engineering, University of Stuttgart, Germany

Mukherjee, R. (1998), "Effectively Design Shell and Tube Heat Exchangers", *Chemical Engineering Progress, American Institute of Chemical Engineers*

Nithiarasu, P. (2005), "Role of Numerical Modeling in Compact Heat Exchanger Analysis", *Proceedings of Fifth International Conference on Enhanced, Compact and Ultra-Compact Heat Exchangers: Science, Engineering and Technology, Hoboken, NJ, USA*

Patel, S. K. and Mavani, A. M. (2012), "Shell and Tube Heat Exchanger Thermal Design with Optimization of Mass Flow Rate and Baffle Spacing", vol. 2, issue 1, pp. 130-135, *IJAERS*

Raj, R. T. K. and Ganne, S. (2012), "Shell Side Numerical Analysis of a Shell and Tube Heat Exchanger Considering The Effects of Baffle Inclination Angle on Fluid Flow", vol. 16, No. 4, pp. 1165-1174, *Thermal Science*

Roll, A. K. S. (2013), "Shell and Tube Heat Exchanger Design Using CFD Tools",

Unpublished B. Tech Thesis, Chemical Engineering, National Institute of Technology, Rourkela

Shah, R. K. and Sekulic, D. P. (2003), *Fundamentals of Heat Exchanger Design*, Wiley

Wang, Q; Chen, Q; Chen, G. and Zeng, M. (2008), "Numerical Investigation on Combined Multiple Shell-Pass Shell and Tube Heat Exchanger with Continuous Helical Baffles", vol. 52, pp. 1214–1222, *Int. J. of Heat and Mass Transfer, Elsevier Ltd*

Yusuf Ali Kara, Y. A. and Guraras, O. (2004), "A Computer Program for Designing of Shell and Tube Heat Exchangers", vol. 24, pp. 1797–1805, *Applied Thermal Engineering, Elsevier Ltd*