Available online at www.elixirpublishers.com (Elixir International Journal)

Thermal Engineering

Elixir Thermal Engg. 73 (2014) 26022-26026

ISSN: 2229-712X

Investigation of baffle spacing effect on shell side heat transfer characteristics in shell and tube heat exchanger using computational fluid dynamics

Kiran K[1,*], A M Mulla[2], Manoj[3] and Umesh C J[4]

[1]Department of Mechanical Engineering, East Point College of Engineering & Technology, Bangalore-560049.

[2]Department of Mechanical Engineering, SIET, Bijapur-586101.

[3]Department of Mechanical Engineering, Gurunanak Dev Engineering College,Bidar – 585403.

[4]Department of Thermal Power Engineering, Visvesvaraya Technological University PG centre, Gulbarga, Karnataka, India.

ARTICLE INFO

Article history:

Received: 14 June 2014;

Received in revised form: 20 July 2014;

Accepted: 31 July 2014;

Keywords

Baffle Spacing,

Numerical simulation,

Pressure drop,

Heat transfer.

ABSTRACT

Three-dimensional CFD simulations are carried out to evaluate the effect of baffle spacing on heat transfer and fluid flow characteristics of a single phase, single pass and 35% baffle cut, single segmental baffled Shell and Tube Heat Exchanger. Shell and tube heat exchanger is the most common type of heat exchanger and widely use in oil refinery and other large chemical processes because it is suitable for high pressure application. The processes in analyzing the simulation consist of modelling (SolidWorks 2012) and meshing (HyperMesh11). Then, the boundary condition will be set before been simulate in ANSYS Fluent version 6.3 based on the experimental parameters. The variation in Shell side Outlet temperature (K), Heat transfer coefficient (W/m^{2}K), Shell side Pressure drop (Pa) and Total transfer rate (W) are measured for different Baffle spacing Bc and for different mass flow rate Kg/s. The sensitivity of the shell side heat transfer characteristics are analysed and concluded.

Introduction

Shell-and-tube heat exchangers in various sizes are widely used in industrial operations and energy conversion systems. Tubular Exchanger Manufacturers Association (TEMA) regularly publishes standards and design recommendations (9th edition is published in 2007 [1]). STHE are successfully designed according to TEMA standards and using recommended correlation based analytical approaches. The analytical approaches have constantly improved since the early days due to accumulating industrial experience and operational data, and improving instrumentation. These correlation based approaches of designing can be used for sizing and can also be used iteratively to obtain general performance parameters (rating) of a heat exchanger. For a value of given iteration, the value of the performance for the considered design is calculated to be unacceptable, a better validating design can be obtained by changing the design parameters in the right direction. An experienced heat exchanger designer knows what to change in which direction. As the simplest example: if the tube side heat transfer coefficient comes out smaller than expected, one can guess that the velocities are low and try a configuration with a smaller cross-sectional flow area in the subsequent iteration. Even though it is relatively simple to adjust the tube side design parameters, it is very complex to get the right combination for the shell side. If possible, an ability to visualize the flow and temperature fields on the shell side can simplify the assessment of the weaknesses, thus directs the designer to the right direction. Computational Fluid Dynamics (CFD) can be very useful to gain that ability.

The shell side flow is very complicated in shell-and-tube heat exchangers due to many different leakage paths and bypass streams between different flow zones. For different parameters of shell designs and sizes, the importance of the leakage and bypass streams may vary. However in small heat exchangers, these bypass streams and leakages are either do not exist or are negligible compared to the main flow stream. The heat exchanger model used in this study is comparatively small sized; therefore compared to the main stream, all of the leakage and bypass streams can be deserted. Even for such designs, the shell side flow still has a complicated structure due to the existence of baffles. Baffles are used for directing the flow inside the shell from the inlet to the outlet while maintaining effective circulation of the shell side fluid hence providing in effect use of the area of heat transfer. In the present study Single segmental baffle is used, which is the most common baffle type. These baffles have a cut, which allows the fluid to pass through in parallel or counter flow direction. The baffle cut (Bc) is measured as a percent of the baffle diameter. A number of baffles (N_b) are placed along the shell in alternating orientations. This is done in order to create flow paths across the tube bundle (forming cross flow windows). The spacing between baffles (B) determines the arrangement of the stream. In the structure shown in Figure 1, equally spaced 4 baffles are used. The thermal properties like Fluid flow and heat transfer characteristics are very sensitive to baffle spacing.

The most commonly used correlation based approaches for designing the shell side are the Kern method [2] and the Bell-Delaware method [3]. The Kern method gives conservative results and is only suitable for the preliminary sizing. The Bell-Delaware method is a very detailed method and is usually very accurate in estimating the shell side heat transfer coefficient and the pressure drop for common shell side geometric arrangements. Generally, the Bell-Delaware method is used for rating, this method can indicate the existence of possible weaknesses in the shell side design, but it cannot explains where these weaknesses are.

To be able to understand the causes of the shell side design weaknesses, the flow phenomenon inside the shell must be well understood. For that purpose, numerous analytical, experimental and numerical studies have been performed. Most of these

Tele:

E-mail addresses: kirankavalli@gmail.com

studies were concentrated on the certain aspects of the shell-and-tube heat exchanger design. Among others, Gay et al. [4] worked on heat transfer, while Halle et al. [5], Pekdemir et al [6], Gaddis and Gnielinski [7] investigated pressure drop. Some of the researchers concentrated only on certain parts of the shell-and-tube heat exchanger. Li and Kottke [8], [9] and Karno and Ajib [10] investigated the effect of tube arrangement on the heat transfer. Sparrow and Reifschneider [11], Eryener [12], Karno and Ajib [13] studied the effects of baffle spacing on both the heat transfer and the pressure drop. In the present study, we made an approach to numerically simulate the Shell and tube heat exchanger (STHE) unit that been used in Department of PG Studies, VTU Gulbarga is modelled for detailed Computational Fluid Dynamics (CFD) simulations.

Fig. 1: An unit of STHE in VTU Gulbarga

Compared to the methods which are based on correlation, the application of CFD in heat exchanger design is limited to certain extent. CFD can be used both for the rating purposes as well as to iteratively in the sizing of heat exchangers. It can be particularly useful in the preliminary design steps, which reduces the number of testing of prototypes and providing a good insight in the transport phenomena occurring in the heat exchangers [14]. To be able to run a successful full CFD simulation for a detailed heat exchanger model, large amounts of computing power and computer memory as well as long computation times are required. Without any simplification, an industrial shell-and tube heat exchanger with 500 tubes and 10 baffles would require at least 150 million computational elements, to resolve the geometry [15]. It is not possible to model such geometry by using an ordinary computer. To overcome that difficulty, in the previous literatures, huge scale shell-and tube heat exchangers are modelled by using some interpretations. The commonly used interpretations are the porous medium model and the distributed resistance approach. Prithiviraj and Andrews [15], [16] modelled shell-and-tube heat exchangers using distributed resistance approach. From this method, a single computational cell may have multiple tubes; therefore, shell side of the heat exchanger was modelled by use of relatively coarse grid. Sha et al. [17] carried out a study by developing a multidimensional, thermal-hydraulic model. In this study shell side was modelled using surface permeability, volumetric porosity and distributed resistance approaches. He et al. [18] modelled three types of shell-and-tube heat exchangers using a distributed resistance approach with a modified porous medium model. Stevanovic et al. [19] carried out numerical analysis of three dimensional fluid flow and heat transfer in a shell-and-tube heat exchanger in which the baffles and the tube bundle were modelled using porous media. In all of these approaches, the results like shell side pressure drop and heat transfer rate results showed good agreement with experimental data.

With these basic approaches, one can predict the parameters like shell side heat transfer coefficient and pressure drop successfully. But, the in detail visualization of the shell side flow and temperature fields, a full CFD model of the shell side is essential. With ever increasing computational capabilities, the number of elements or cells that can be used in a CFD model are also increasing. For a kind of double-pipe heat exchangers, there are two recent studies using full CFD models [20] [21], however to our knowledge there is a less no of analysis were made on shell-and-tube heat exchangers. Nowadays, there is a possibility to model a small shell-and-tube heat exchanger in detail with the available computers and software. By modelling the geometry as accurately to the mark, it is possible to get the flow structure and the temperature. This detailed data can be used for calculating global parameters such as heat transfer coefficient and pressure drop that can be compared with the correlation based ones. Furthermore, the data can also be used for visualizing the flow and temperature fields which can help to locate the weaknesses in the design such as recirculation zones. The objective of the present study is to explore the possibilities and limitations of full CFD modelling and analysis of the shell side by using current computer technology and a current commercial CFD packages to fill the gap in the literature.

In this study, a small shell-and-tube heat exchanger is modelled for CFD simulations. A commercial CFD package, ANSYS Fluent version 6.3 [22], is used together with HuperMesh11 as mesh generation software. After selecting a suitable mesh, a discretization scheme and a turbulence model, simulations are performed. The simulation results are used for calculating shell side heat transfer coefficient and pressure drop.

Model description

The number of tubes that can be placed within a shell depends on shell diameter, pitch size, number of passes, tube outer diameter and tube layout. These design parameters have been standardized and given as tabulated form that usually called ‘‘tube counts’’. Many tube count tables are available in open literature [4, 17, and 18].

Table 1: Specification of Heat Exchanger

Shell Size, D_s	200mm
Tube Outer Diameter, D_o	19mm
Tube Inner Diameter, D_i	16mm
Tube Bundle Geometry	Triangular
Pitch, P_t	30mm
Number Of Tubes, N_t	18
Heat Exchanger Length, L	800mm
Tube Length, L_t	825mm
Shell Side Inlet Temperature, T	300K
Baffle Cut, B_c	35%
Central Baffle Spacing, B	200, 120, 86, 67mm
Number of Baffles, N_b	4, 6, 8, 10

In this study, a small heat exchanger is selected in order to increase the model detail and to make solid observations about the fluid flow inside the shell. The design parameters and the predetermined geometric parameters are presented in ***Table 1.*** The geometric model with 4, 6, 8 and 10 baffles are represented in ***Figure 2, Figure 3, Figure 4 and Figure 5*** respectively. 35% baffle cut value is selected to place the cut just below or above the central row of tubes. The model is made by using SolidWorks 2012 tool.

The working fluid of the shell side is water. Since the properties of water are defined as constants in the fluent database, to improve the accuracy, they are redefined using piecewise-linear functions of temperature by using the "Thermo-Physical Properties of Saturated Water" tables available in the literature [23].

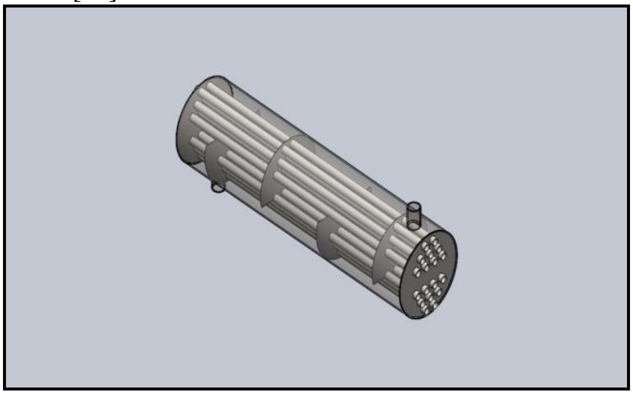

Fig. 2: Equally spaced 4 baffles

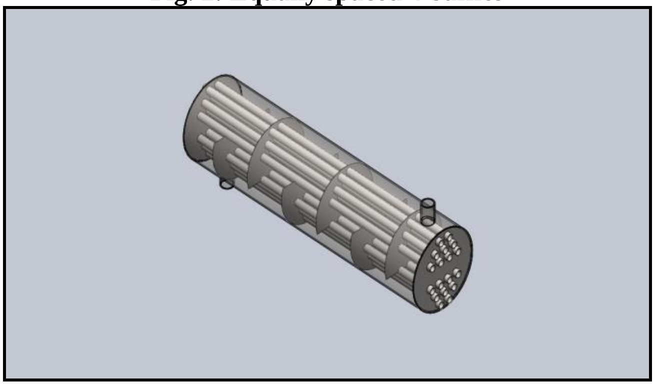

Fig. 3: Equally spaced 6 baffles

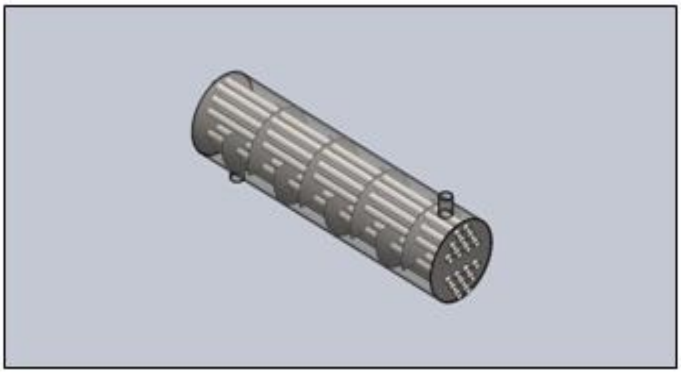

Fig. 4: Equally spaced 8 baffles

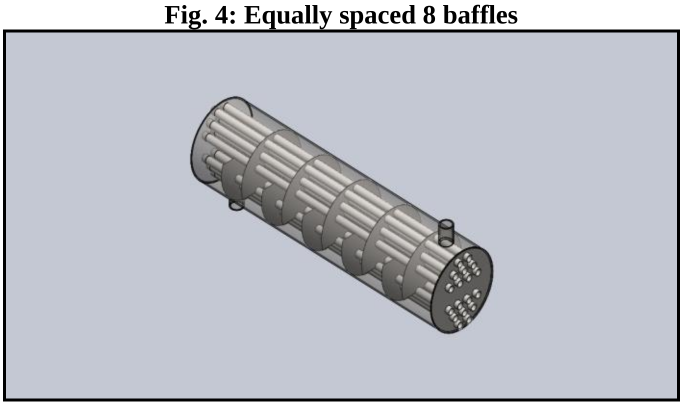

Fig. 5: Equally spaced 10 baffles

Mesh selection

A Tetrahedral mesh is generated using HyperMesh 11 pre-processor. First, the surface of the HE is meshed with R-TRI element. During the mesh generation, a special care has been taken to the zones close to the walls. The domain has been subdivided into growing boxes to make it easier to generate the grid. The tetrahedral mesh volume created consist of simple pyramids. Fine mesh has been created with element size of 5 with standard quality criteria. A typical 3D mesh is represented with different meshes that take part in the study are showed in the following detailed ***Figure 6.*** Proper deck is created prior to importing for Fluent 6.3.

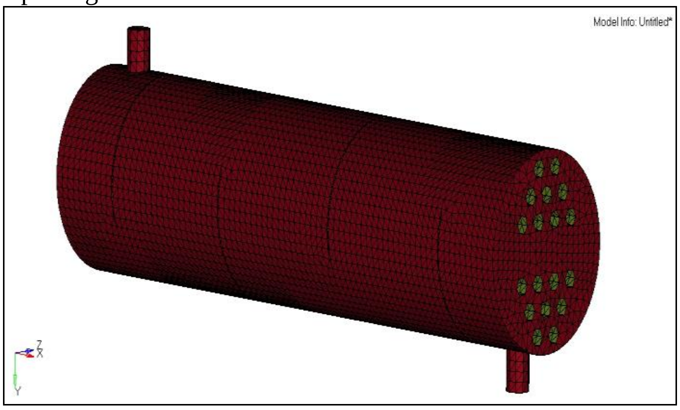

Fig. 6: Typical 3D Mesh

Boundary conditions

Since we are analyzing shell and tube heat exchanger, generally flow is Turbulent on shell side, Flow is steady, proper boundary conditions are needed for a successful computational work. The desired mass flow rate and standard temperature values are assigned to the inlet nozzle of the heat exchanger. The shell inlet temperature is set to 300K. Zero gauge pressure at the outlet nozzle is essential, in order to obtain the relative pressure drop between inlet and outlet.

Since water is using as fluid, Pressure based conditions are used due to incompressible property of the water. No slip condition is assigned to all surfaces. Assuming the shell is perfectly insulated outside, the heat flux is assigned to zero for the shell outer wall.

Since the tube side flow is easy to resolve (from the literatures), so the present study concentrated on the shell side flow. Tubes are modelled as solid cylinder. A constant wall temperature of 450K is assigned to the tube wall as the boundary condition.

In this study, K-ε turbulence model is tried: standard and realizable [22].

Results and discussion:

As the first step of the present study, for the present heat exchanger with turbulent flow, the sensitivity of the results to the turbulence model and to the Baffle spacing is investigated for three different shell side mass flow rate values. Then, with the selected turbulence model, suitable solver set up and discretization scheme, the variations in shell side heat transfer coefficient and pressure drop values with the different flow rate and with different baffle spacing are investigated.

Contours of Temperature for different baffle spacing are showed in below *Figure7, Figure8, Figure9 and Figure-10.*

Variation of *Shell side Outlet temperature (K), Heat transfer coefficient (W/m^{2}K), Shell side Pressure drop (Pa) and Total transfer rate (W)* for different Baffle spacing Bc are shown in *Table 2.*

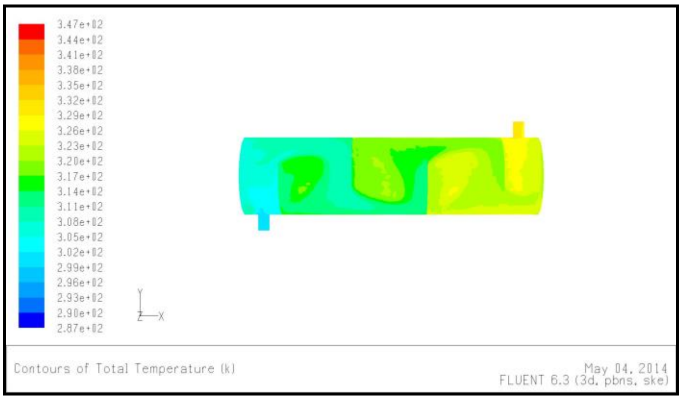

Fig. 7: Temperature Contour for Nb=4

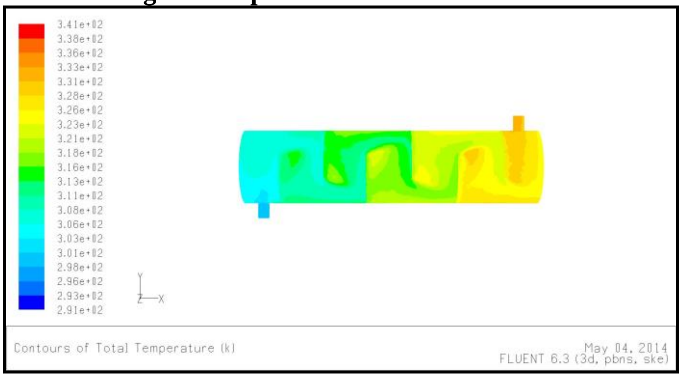

Fig. 8: Temperature Contour for Nb=6

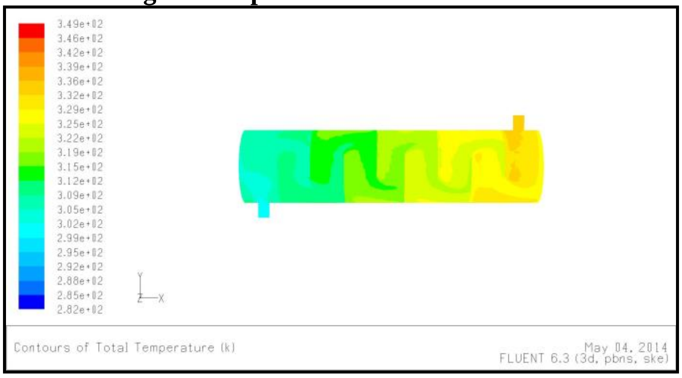

Fig. 9: Temperature Contour for Nb=8

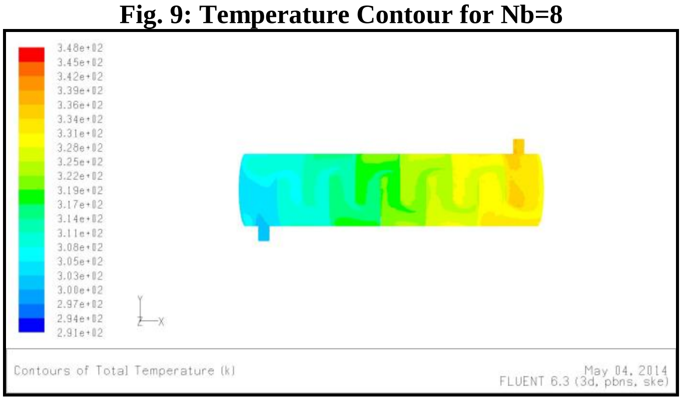

Fig. 10: Temperature Contour for Nb=10

Table 2:

N_b	*Mass Flow Rate Kg/s*	*Shell side Outlet temp. (K)*	*Heat transfer coeff. (W/m^2K)*	*Shell side Pressure drop (Pa)*	*Total transfer rate (W)*
4	*0.5*	357.04	32756	1483	72367.5
	1	346.47	35428	5640	122023.4
	1.5	344.7	37180	12434	173681
6	*0.5*	359.47	32795	1516	76213.71
	1	348.05	35566	5815	134898.7
	1.5	346.46	37184	12893	193769.7
8	*0.5*	361.43	32836	1551	80030.10
	1	350.36	35684	5991	144947.8
	1.5	348.89	37185	13329	206455.2
10	*0.5*	363.36	32964	1584	84752.66
	1	354.83	35812	6165	156143.7
	1.5	351.08	37190	13666.2	218651.2

The variation of Shell side Outlet temperature (K) with respect to baffle spacing (B) is increasing by small value, but the variation in Heat transfer coefficient (W/m2K), Shell side Pressure drop (Pa) and Total transfer rate (W) respect to baffle spacing has increased by small value.

The variation of Shell side Outlet temperature (K) with respect to mass flow rate (Kg/s) is decreasing by a significant value, but the variation in Heat transfer coefficient (W/m2K), Shell side Pressure drop (Pa) and Total transfer rate (W) respect to baffle spacing has increased by significant value.

Conclusion

The Shell side CFD analysis of a small shell-and-tube heat exchanger is modelled with sufficient detail to resolve the flow and temperature parameters. The resulted values of shell side heat transfer coefficient, pressure drop and heat transfer values are obtained from the CFD simulation of fixed tube wall and shell inlet temperatures. K-ε standard turbulence model with first order discretization and fine mesh is selected for simulation approach.

By varying the baffle spacing between shell 4 and 10 for 0.5, 1 and 1.5 kg/sec shell side flow rates, the simulation results are found.

From the above results we can conclude that the sensitivity of ***Shell side Outlet temperature (K)*** is less w.r.t. baffle spacing (B) where as it has a significat change w.r.t. mass flow rate (Kg/s)**.**

Heat transfer coefficient (W/m^2K), Shell side Pressure drop (Pa) and Total transfer rate (W) increases w. r. t. both *baffle spacing Bc and mass flow rate Kg/s.*

But for better performance of STHE one should design with considerable change in Shell side Pressure drop (Pa). Depending upon the need and heat transfer characteristics one can predict and decide from the above baffle spacing (B) and mass flow rate (Kg/s) as the best operating parameters for the STHE.

References

TEMA Inc.; 2007.

Kern DQ. Process heat transfer. New York, N.Y.: McGraw-Hill; 1950.

Bell KJ. Delaware method for shell side design, in heat exchangers: thermal hydraulic fundamentals and design. (Eds) Kakaç S, Bergles AE, Mayinger F. New York: Hemisphere; 1981; 581-618.

Gay B, Mackley NV, Jenkins JD. Shell-side heat transfer in baffled cylindrical shell- and-tube exchangers - an electrochemical mass transfer modelling technique. Int J Heat Mass Tran 1976; 19:995-1002.

Halle H, Chenoweth JM, Wabsganss MW. Shell side water flow pressure drop distribution measurements in an industrial-sized test heat exchanger. J Heat Trans 1988; 110:60-7.

Pekdemir T, Davies TW, Haseler LE, Diaper AD. Pressure drop measurements on the shell side of a cylindrical shell-and-tube heat exchanger. Heat Transfer Eng 1994; 15:42-56.

Gaddis ES, Gnielinski V. Pressure drop on the shell side of shell-and-tube heat exchangers with segmental baffles. Chem Eng Process 1997; 36:149-59.

Li HD, Kottke V. Visualization and determination of local heat transfer coefficients in shell-and-tube heat exchangers for staggered tube arrangement by mass transfer measurements. Exp Therm Fluid Sci 1998; 17:210-6.

Li HD, Kottke V. Visualization and determination of local heat transfer coefficients in shell-and-tube heat exchangers for in-line tube arrangement by mass transfer measurements. Heat Mass Transfer 1998; 33:371-6.

Karno A, Ajib S. Effect of tube pitch on heat transfer in shell-and-tube heat exchangers - new simulation software. Heat Mass Transfer 2006; 42:263-70.

Sparrow EM, Reifschneider LG. Effect of interbaffle spacing on heat transfer and pressure drop in a shell-and-tube heat exchanger. Int J Heat Mass Tran1986; 29:1617-28.

Eryener D. Thermoeconomic optimization of baffle spacing for shell and tube heat exchangers. Energ Convers Manage 2006; 47:1478–89.

Karno A, Ajib S. Effects of baffle cut and number of baffles on pressure drop and heat transfer in shell-and-tube heat exchangers - numerical simulation. Int J Heat Exchangers 2006;7:299-322.for Bioenergy Elephants in stone (Hoysala period, c. 1260 A. D) bv Manjula Wijesundara Gajah, vol,22 (2003).

Sunden B. Computational heat transfer in heat exchangers. Heat Transfer Eng 2007; 28:895-7.

Prithiviraj M, Andrews MJ. Three dimensional numerical simulation of shelland- tube heat exchangers. Part 1: foundation and fluid mechanics. Numer Heat Transfer. Part A, Applications 1998; 33:799-816.

Prithiviraj M, Andrews MJ. Three dimensional numerical simulation of shell and- tube heat exchangers. Part 2: heat transfer. Numer Heat Transfer. Part A,Applications 1998; 33:817-28.

Sha WT, Yang CI, Kao TT, Cho SM. Multidimensional numerical modeling of heat exchangers. J Heat Trans 1982; 104:417-25.

He YL, Tao WQ, Deng B, Li X, Wu Y. Numerical simulation and experimental study of flow and heat transfer characteristics of shell side fluid in shell-and-tube heat exchangers. Proc Fifth Int Conf on Enhanced, Compact and Ultra-Compact Heat Exchangers: Science, Engineering and Technology. Hoboken, NJ. Sep. 2005:29-42.

Stevanovi$ Z, Ili$ G, Radojkovi$ N, Vuki$ M, Stefanovi$ V, Vu%kovi$ G. Design of shell and tube heat exchangers by using CFD technique – Part one: thermo hydraulic calculation. Facta Universitatis Series: Mechanical Engineering 2001; 8:1091-105.

Moawed MA, Ibrahim E, Gomaa A. Thermal performance of a pipe in pipe heat exchanger with sinusoidal inner pipe. Energ Convers Manage 2008; 49:678–86.

Chen W-L, Dung W-C. Numerical study on heat transfer characteristics of double tube heat exchangers with alternating horizontal or vertical oval cross-section pipes as inner tubes. Energ Convers Manage 2008; 49:1574–83.

ANSYS Fluent version 6.3. Fluent 6.3 User's Guide. ANSYS Inc.

Incropera FP, Dewitt DP. Fundamentals of heat and mass transfer. 4th Ed. New York: J. Wiley; 1996.

Sunden B. Computational fluid dynamics in research and design of heat exchangers. Heat Transfer Eng 2007; 28:898-910.

Spalart PR, Allmaras SR. A One-Equation turbulence model for aerodynamic flows. AIAA Paper 92-0439.