International Journal of Mechanical Engineering (IJME)
ISSN(P): 2319-2240; ISSN(E): 2319-2259
Vol. 2, Issue 5, Nov 2013, 61-66

International Academy of Science, Engineering and Technology
Connecting Researchers; Nurturing Innovations

SIMULATION STUDIES ON WAVY AND CHEVRON-CORRUGATED PLATE HEAT EXCHANGERS USING FLUENT

B. SREEDHARA RAO[1] & R. C. SASTRY[2]

[1]Department of Chemical Engineering, CBIT, Hyderabad, Andhra Pradesh, India

[2]Department of Chemical Engineering, NIT, Warangal, Andhra Pradesh, India

ABSTRACT

Plate heat exchangers are commonly used for heat transfer enhancement at transitional and turbulent Reynolds Numbers. In the present study, numerical simulation is used to investigate the friction factor and Nusselt numbers in plate heat exchangers, for inferring the effect of channel geometry and flow conditions on the heat transfer of the exchangers. Two PHEs, one with wave geometry and another with a chevron geometry are considered with Fluent as a CFD tool. For both exchangers, the temperature of the wall is kept constant and water is used as the fluid and the mass flow rate varied to study the effect of Reynolds number.

KEYWORDS: Chevron Design, Fluent, Mathematical Model, Plate Heat Exchangers, Wave Design

INTRODUCTION

Plate heat exchangers are widely recognized today as the most economical and efficient type of heat exchangers, with its low cost, flexibility and high thermal efficiency. They are usually designed to achieve turbulence across the entire heat transfer area in order to get the highest possible heat transfer coefficient with the lowest possible pressure drop and the close temperature approach.

A wide range of corrugations are used in the plate design. Thermally, the corrugation would induce turbulence and thereby increases the heat transfer area from 1.5 to 25 times. The two most adopted forms can be described as ‘Wavy type’ and ‘Chevron type’. The generated geometries of these two types are shown in Figure 1 and Figure 2 respectively.

In the former case, there is a constant change in direction and cross-sectional area in the direction of flow and the turbulence is induced by continual variation of liquid velocity. In the latter case, the cross-sectional area in the direction of flow is constant, but the change in shape of the flow passage induces turbulence by continual change in direction.

MATHEMATICAL MODEL

Fluent, the CFD tool for the present work provides a number of choices, which include the standard RNG and realizable k-€ models, the Reynolds stress model etc., and researchers have used different turbulence models for their studies on plate heat exchangers.

The choice of the k-€ model is the most popular two-equation model and provides the best performance of all the model versions. The earliest development of k-€ model was due to [3] [5] [6]. The widespread use of the model began with the introduction of the [7] version. The model coefficients were retuned [8] to produce the standard k-€ model.

The standard k-€ model is given by [2]

Kinetic Eddy Viscosity

$$\nu_T = C_\mu \frac{k^2}{\varepsilon}$$

Turbulence Kinetic Energy

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial U_i}{\partial x_j} - \varepsilon + \frac{\partial}{\partial x_j}\left[\left(\nu + \frac{\nu_T}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]$$

Dissipation Rate

$$\frac{\partial \varepsilon}{\partial t} + U_j \frac{\partial \varepsilon}{\partial x_j} = C_{\varepsilon 1} \frac{\varepsilon}{k} \tau_{ij} \frac{\partial U_i}{\partial x_j} - C_{\varepsilon 2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j}\left[\left(\nu + \frac{\nu_T}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]$$

Closure Coefficients and Auxiliary Relations

$C_{\varepsilon 1}$	$C_{\varepsilon 2}$	C_μ	σ_k	σ_ε	ω	ℓ
1.44	1.92	0.09	1.0	1.3	$\varepsilon/(C_\mu k)$	$C_\mu k^{3/2}/\varepsilon$

METHODOLOGY

Geometry Generation and Meshing

The analysis is carried out using Gambit as the preprocessor for the geometry creation and mesh generation. The geometry is created with the ‘Tools’ operations, using the ‘display grid’ under the ‘coordinate system’. By specifying the minimum and maximum values on both x and y axes and the incremental values, the geometry is created.

Then the grid density on the boundary edges of the geometry and meshing are defined. At the inlet and outlet boundaries, a ratio value of 1.11 and spacing of 15 is considered. At the wall edges a ratio value of 1 and spacing of 60 is considered. In the final step the boundary types, which are mass flow rate (inlet) and out flow (outlet) are specified.

Analysis and Solution

The governing equations are solved using the commercial CFD package FLUENT with the following assumptions [1]

- There is no maldistribution of flow
- The surface of heat transfer is free from fouling
- Periodicity exists in the x-direction

The following settings are used for the model solution

Table 1

Function	Specification
Solver	2D (space), implicit (formulation), absolute(velocity formulation),cell based (gradient orientation), superficial velocity (porous formulation)
Energy	Energy equation
Viscous	Laminar: laminar, Turbulent: k-€ , realizable, non-equilibrium wall function
Material	Water-liquid. By default the properties of water

Table 1: Contd.,

BCs Fluid	Water-liquid. Everything else id default
Inlet	Mass flow rate: 0.05 Kg/sec; Temperature: 315K; X-component of flow direction: 1; Y-component of flow direction: 0; Turbulence intensity: 5%; Hydraulic diameter:0.02
Outlet	Flow rate weighing: 1 by default
Wall	Thermal:Temperature(Thermal conditions); Temp: 350K; Wall thickness: 0.002m; Aluminium (material name) Momentum: Stationary wall; No slip; roughness height: 0; roughness constant:0.5

It is assumed that the flow is uniformly distributed at the entry of the heat transfer surface. The hydraulic diameter is specified as twice the mean flow channel gap. Most of the choices made are based on the previous studies [4], which state that these are the most appropriate settings for a k-€ model for FLUENT.

After the results are obtained, the mesh is redefined and the solution recalculated, until it became mesh independent. The mesh details are as follows:

Wave geometry:

19250 nodes, 18710 tetrahedral cells

Chevron design:

16500 nodes, 15900 tetrahedral cells

All numerical simulation is implemented on a personal computer with Intel Core 2 3 GHz CPU and 2GB RAM

RESULTS AND DISCUSSIONS

Friction Factor

Simulations are carried out with Reynolds numbers ranging from 100 to 25000. The post processing is carried out in FLUENT and during the process, 4 Reynolds numbers of 100,400,800 and 1600 corresponding to laminar flow and 4 Reynolds numbers of 3000,6000,12000 and 25000 corresponding to turbulent flow are used. The criteria for determination of laminar/turbulent flow are with reference to the straight pipe . But in the plate heat exchanger, the corrugated patterns induce turbulent flow at much lower Reynolds number compared to the straight pipe.

Therefore, a flow rate, which would fall in laminar flow in a straight pipe, might exhibit turbulent or transitional characteristics. From Figure 3, the effect of using a turbulent model for a laminar Reynolds number flow is very visible. For the laminar region, the wave design gives higher friction factor compared to chevron design. However from Figure 4, it is observed that, the chevron design gives higher friction factor values compared to wave design and these values are almost remaining constant with increase in the Reynolds number .This might also be the reason for the preference of the chevron design at very high values of Reynolds number.

Nusselt Number

The Nusselt number is an important dimensionless parameter that represents the temperature gradient at the surface, where heat transfer by convection is taking place. From Figure 5, the effect of using a turbulent model for a laminar Reynolds number can be observed by the sharp increase in the value of the Nusselt number, which is not seen in the case of a straight pipe.

From Figures 5 and 6, it can be observed that the chevron design offers a smoother transition, compared to wave design. Chevron design exhibits a constant and linear increase of Nusselt number with Reynolds number compared to wave design. This could be one of the contributing factors for its preferred usage in design. It can also observed that, up to a value of Re=16000, Both Figures 5 and 6 show a higher values of Nu for the wave design and for higher Reynolds numbers, the chevron design shows higher values of Nusselt numbers

CONCLUSIONS

The influence of the type of corrugations in a plate heat exchanger on friction factor and Nusselt number is studied and the following conclusion are made.

- The higher heat transfer rates in corrugated plates is primarily due to the induced turbulence created in the plate heat exchanger.
- The performance of a plate heat exchanger with reference to heat transfer and pressure drop is dependent on the plate geometry.
- The choice of a suitable plate geometry/corrugation depends on the specific application. For low Reynolds numbers, the wave design gives better convective properties (approximately 22% more convective heat transfer), while at higher Reynolds numbers, the chevron design gives better convective properties (approximately 23% more convective heat transfer)

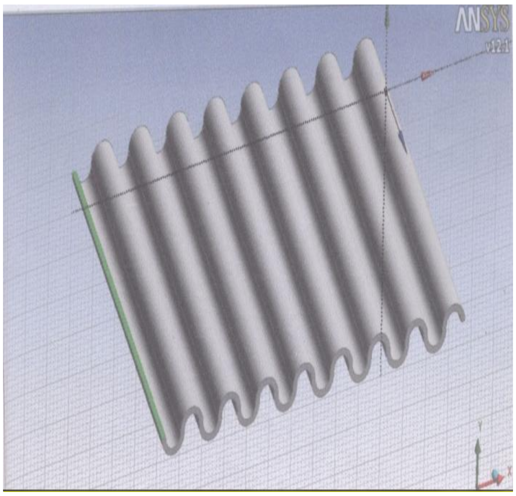

Figure 1: Wave Design

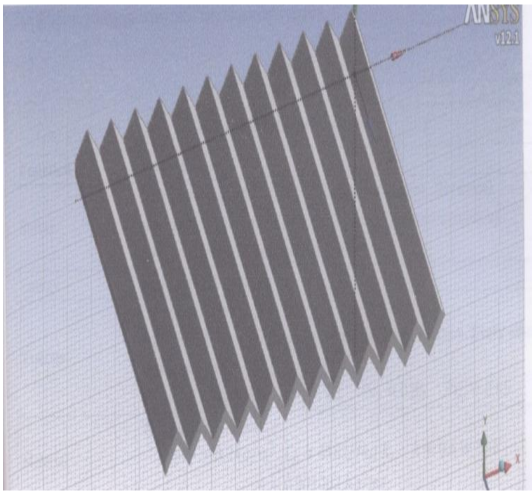

Figure 2: Chevron Design

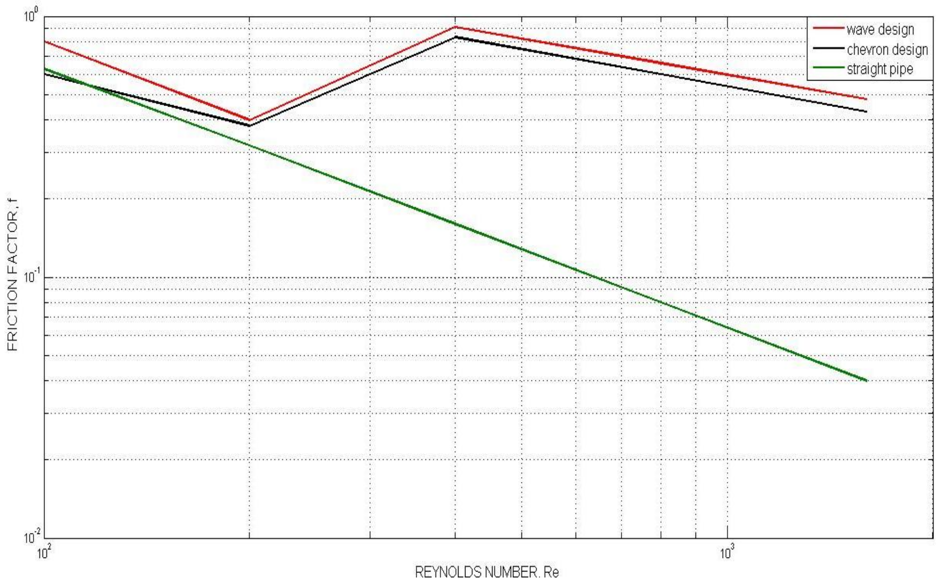

Figure 3: Friction Factor vs Reynolds Number

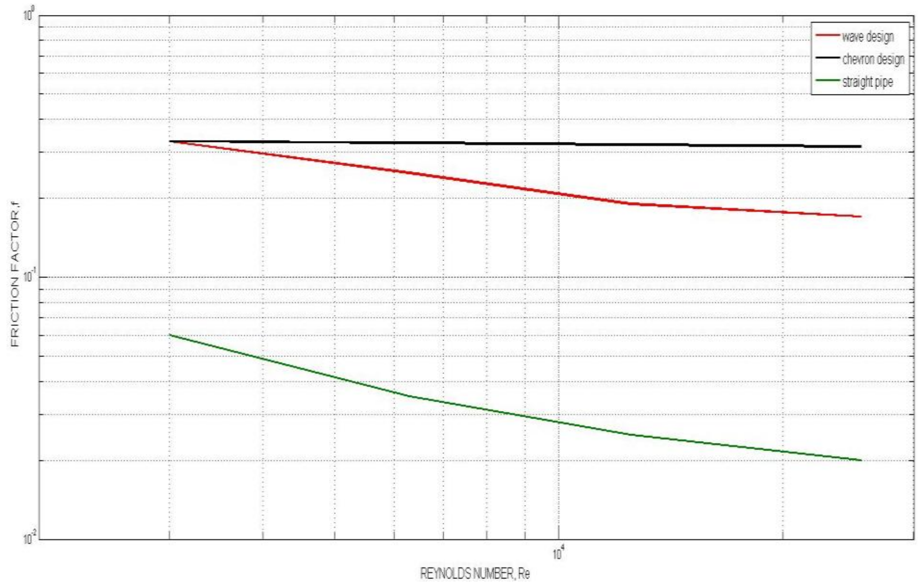

Figure 4: Friction Factor vs Reynolds Number

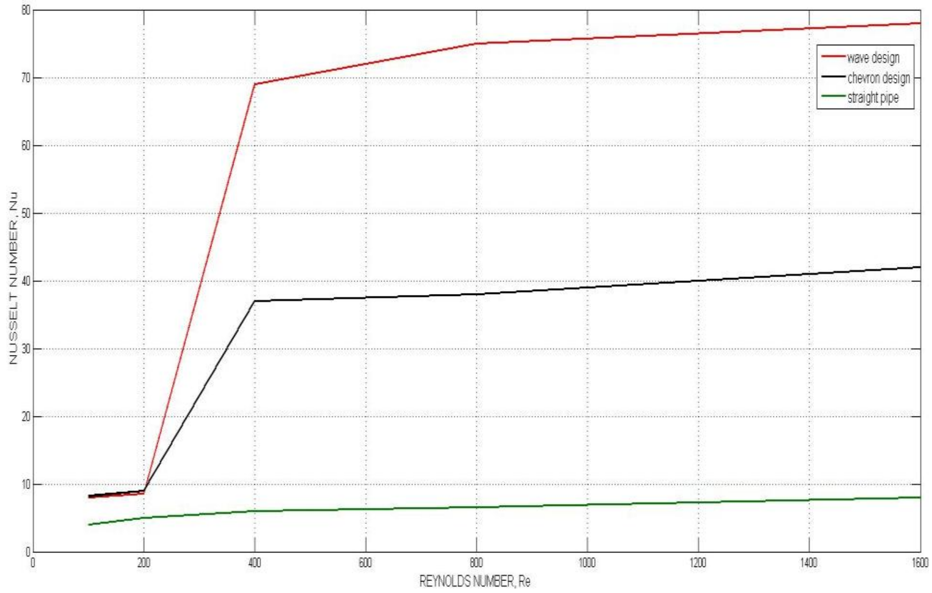

Figure 5: Nusselt Number vs Reynolds Number

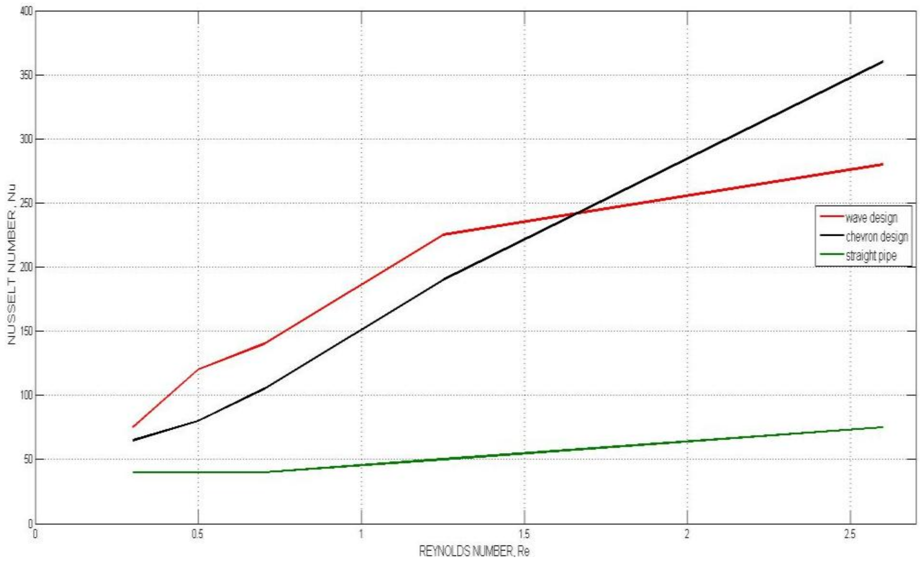

Figure 6: Nusselt Number vs Reynolds Number

REFERENCES

1. B. Sunden and L. Wang., ‘ Optimal design of plate heat exchanger with or without pressure drop specifications’ Applied Thermal Engineering,23 (2003) pp.295-311

2. Davidov, B. I., ‘On the statistical dynamics of an incompressible fluid’ Doklady Academiya Nauk SSSR, Vol.136 (1961) p.47

3. Fluent 12.1 ‘User’s manual and work bench tutorials’

4. Jain, S etal., ‘A new approach to numerical simulation of small sized plate heat exchangers with chevron plates’ Journal of Heat transfer, 129(2007) pp291-297

5. Jones, W. P and Launder, B. E ‘The prediction of laminarization with a two-equation model of turbulence’ International Journal of Heat and Mass Transfer,15(1972) pp 301-314

6. Kolmogorov, A.N., ‘Equations of turbulent motion of an incompressible fluid’ Izvestia Academy of Sciences, USSR physics, 6(1942),1 pp.56-58

7. Launder, B.E., and Sharma, B.I., ‘Application of the energy dissipation model of turbulence to the calculation of flow near a spinning disc’ Letters in Heat and Mass Transfer, 1(1974),2 pp.131-138

8. Lee, Y. S., etal., ‘ Experimental study of heat transfer of corrugated channels’ First International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics, Krugar Park, South Africa (2002)