Journal of Thermal Engineering, Vol. 3, No. 5, pp. 1442-1452, October, 2017
Yildiz Technical University Press, Istanbul, Turkey

NUMERICAL INVESTIGATION OF ISOTHERMAL FLOW AROUND IMPINGEMENT PLATES IN A SHELL AND TUBE EXCHANGER

R. S. Maurya[1] and S. Singh[2]

ABSTRACT

The impingement plate is an important protecting device of tube bank which is located on shell side at the nozzle inlet of shell and tube heat exchanger. Current work presents a 2D numerical investigation of the flow structure developing around different impingement plate design under different flow conditions using ANSYS Fluent. Location, size and arrangement of impingement plate are the investigated parameters which provides significant information for the selection of ideal location and size of required impingement plate. Study compares different plate geometry and concludes the best one which apart from protecting the tube banks, also removes the stagnation zone developing behind them because of their placements.

Keywords: *Impingement Plate, Heat Exchanger, Flow Distribution, TEMA, CFD, Numerical, Simulation*

INTRODUCTION

One of the most widely used process equipment in chemical and processing industries is shell and tube heat exchanger which is expected to work efficiently for years. The impingement plate placed inside the shell facing the shell inlet nozzle is used mainly to protect tubes against shell side inlet flow impingement. Tubular Exchanger Manufacturers Association (TEMA) recommends the use of impingement plate in shell-and-tube heat exchangers where ρV^2 value exceeds 2310 kg/ms^2 for non-corrosive, non-abrasive single phase fluid. For all other liquids and liquids near boiling point this value is close to 744 kg/ms^2. Fluid with such a high kinetic effect has a potential to damage the direct facing tube bank by their erosion, corrosion, vibration and scale formation. A nicely designed impingement plates stimulates fluid motion in the stagnation areas near the ends of the tubes to tube sheet which leads to effective heat transfer and ensures prolong tube life. The impingement plate is placed inside the shell facing the shell inlet nozzle and usually attached directly to the bundle. It is used mainly for the safety and protection of tube bank facing shell inlet nozzle. The fluid entering in the shell is directed towards the impingement plate which bears most of the damaging effect and deflects it.

In the process of protecting the tube bank from direct impact of the high velocity inlet fluid, the impingement plate helps in the development of a stagnant zone behind the plate. The stagnant areas create some serious problem from system's performance point of view.

- Under performance of tube bank section in low velocity zone.
- Corrosion and fouling processes accelerates under stagnant conditions
- Surface temperatures may appreciably exceed the dangerous level.

If the fouling and corrosion becomes severe, the heat exchanger needs to be completely shut down and retubed. These effects are due to loses in kinetic energy and pressure head due to irreversible vortex effect of fluid across impingement plate.

The device shell and tube heat exchanger has been intensively investigated by several researchers both through experimental and numerical methods. This led to a phenomenal growth in design and performance of the device. The investigation in the area of impingement plate is limited. In 1996, Tu and Wood [1] carried out an experimental investigation to determine the distribution of wall pressure developing on a flat plate caused due to normal impingement of turbulent. The investigation included the effect of wide range of Reynolds number and H/D (height / diameter) ratios of nozzle. In a numerical work done by Shi et al. [2], on a single semi-confined turbulence slot jet impinging normally on a flat plate, the effects of turbulence models, near wall treatments, turbulence intensity, jet Reynolds number, as well as the type of thermal boundary condition on the heat transfer were studied using the standard k-ε and RSM models. A hybrid RANS/LES turbulence model and one equation RANS model was successfully used by De Langhe et al. [3] to evaluate jet performance at Re = 20,000 which was validated well with experimental data using different level of mesh refinement.

This paper was recommended for publication in revised form by Regional Editor Balaram Kundu
[1,2] Department of Department of Mechanical Engineering, Sardar Patel College of Engineering, Mumbai, INDIA
**E-mail address: [1]r_maurya@spce.ac.in, [2]satinder89@hotmail.com*
Manuscript Received 15 May 2016, Accepted 27 August 2016

CFD codes have been successfully used by researchers in heat exchanger design to predict flow patterns and thermal fields thus allowing in determining the heat transfer characteristics and other properties. Perrotin and Clodic [4] numerically captured the physics of the flow and predicted the flow unsteadiness at higher Reynolds numbers. Their work was extended by Ozden and Tari [5] where the shell side design of a shell-and-tube heat exchanger such as baffle spacing, baffle cut and shell diameter dependencies of the pressure drop were investigated for a small heat exchanger. The differences between Bell-Delaware and numerically predicted value of the total heat transfer rate were below 2% for most of cases.

Badra et al. [6] carried out a basic study on jet impingement using numerical technique to study transient heat transfer between multiple jets and a moving plate. He carried out an optimization study to find H/D ratio based on stagnation Nusselt Number. The deflected flow of the fluid through the shell by-pass results in poor performance of the shell and tube heat exchanger. It was showed that the fluid bulk take a long path to reach the tubes in the middle of the tube bank. He was successful in obtaining 15 % increase in condenser performance by perforating the plate. An investigation on impingement plate modification was done by Karlsoon and Valming [7] where they successfully obtained 24 % increase in heat transfer using a perforated impingement plate as compared to a conventional solid one. Realizing the problem of low fluid velocity, stagnation and vortex formation below the impingement plate within the first few rows of tube bank, Al-Anizi and Al-Otaibi [8] proposed a modified double perforated impingement plate in place of simple flat plate. It was proved that by removal of stagnation areas flow pattern can be improved. Also non-isothermal flow has been addressed under different condition by Kandu [9] where he investigated the influence of the viscous dissipation on the limiting temperature profile for an unsteady shear driven and Poiseuille flow of Newtonian fluid between two asymmetrically heated parallel plates.

Review of literature reveals that sound understanding of developing flow structure due different impingement plate geometries is only partially understood. It needs further investigation. A systematic investigation of impingement plate induced flow structure is expected to improve the shell and tube heat exchanger design. The present work tries to address this issues by numerical investigation and propose a better design of impingement plate which can serve its purpose and simultaneously lead to better system performance. The work investigates impingement plate size, location, arrangement under different flow conditions and finally recommends the best one.

Problem Definition

In order to get the insights into the flow structure developing around the impingement plate of different configuration, 2D computational domain is considered where flow enters in semi-circular shell through a nozzle and passes through an impingement plate located a little distance downstream. The turbulent flow below the impingement plate is investigated with and without tube banks. For the completeness of the investigation a three dimensional computational domain is also considered for the investigation. The proposed work deals with providing a numerical investigation based better design of impingement plate which fulfils its objective without hampering performance. It optimizes impingement plate location and its size with respect to nozzle inlet from flow distribution view point where Re is varied from 110000 to 610000. $\mathrm{Re} = \mathrm{VD}/\nu$, is defined for water flow through inlet nozzle at ambient temperature where V is inlet velocity and D is diameter of the nozzle. The quantitative analysis of impingement plate is presented under the parameters - pressure loss and kinetic energy loss and flow penetration across impingement plate.

NUMERICAL DETAILS

The flow is assumed to be due to single phase Newtonian fluid with constant fluid properties under isothermal, steady, incompressible and turbulent flow conditions. Body forces arising due to gravitational effect and the effect of viscous dissipation are minimal. The developing flow structure is only due to the basic conservation law where viscous force and pressure gradient plays significant role.

To be able to determine the renewable energy potential of Turkey, general information and maps were given according to four renewable energy types; wind, wave, current and solar energy. The main aim of this study is to find the most convenient location for a floating renewable energy platform. For this reason, five different regions were selected to analyze and to be able to make a comparison, the power data of the different energy types were converted to same unit and the regions were ranked with respect to these results.

Governing Equations

A time averages conservation equation can be written as follows.

Mass Balance Equation:

$$\operatorname{div}\mathbf{U} = 0 \tag{1}$$

Momentum Equation

$$\operatorname{div}(U\mathbf{U}) + \operatorname{div}\left(\overline{u'\mathbf{u}'}\right) = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu \operatorname{div}\left(\operatorname{grad}(U)\right) \tag{2}$$

$$\operatorname{div}(V\mathbf{U}) + \operatorname{div}\left(\overline{v'\mathbf{u}'}\right) = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu \operatorname{div}\left(\operatorname{grad}(V)\right) \tag{3}$$

$$\operatorname{div}(W\mathbf{U}) + \operatorname{div}\left(\overline{w'\mathbf{u}'}\right) = -\frac{1}{\rho}\frac{\partial p}{\partial z} + \nu \operatorname{div}\left(\operatorname{grad}(W)\right) \tag{4}$$

where U, V and W are velocity components and p indicates local pressure. The third term on the left hand side of the equation represents the presence of turbulent stresses.

In order to capture turbulent features of flow, k-ε model of turbulence is used. So complete mathematical model includes two more transport equation – turbulent kinetic energy (k) and dissipation rate (ε).

$$\frac{\partial(\rho k)}{\partial t} + \operatorname{div}(\rho k\mathbf{U}) = \operatorname{div}\left(\frac{\mu_t}{\sigma_k}\operatorname{grad} k\right) + 2\mu_t S_{ij}\cdot S_{ij} - \rho\varepsilon \tag{5}$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \operatorname{div}(\rho \varepsilon\mathbf{U}) = \operatorname{div}\left(\frac{\mu_t}{\sigma_k}\operatorname{grad} \varepsilon\right) + C_{1\varepsilon}\frac{\varepsilon}{k}2\mu_t S_{ij}\cdot S_{ij} - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{6}$$

where $\mu_t = \rho \cdot C_\mu \dfrac{k^2}{\varepsilon}$ and the value of the constant appearing in the equation is obtained from the benchmark results.

The boundary condition needed to carry out the investigation is based on experimental work of Tu and Wood [1] for the inlet condition i.e. velocity inlet (1.426 m/s) at nozzle inlet. All along the solid surfaces no-slip wall condition and pressure outlet at outlet of the flow from the domain are used.

Numerical Implementation

Present numerical investigation uses ANSYS Fluent 13.0 as analysis tool box. The investigation starts with a 2D analysis without tube banks behind the impingement plate and later tube bank is added to see its effect on flow distribution. Figure 1 shows 2D computational domain with its dimension and meshed zone. Appropriate mesh size is selected to capture the flow features arising inside the domain. As the flow is assumed to be isothermal, steady and incompressible with flow turbulence, a pressure based SIMPLEC algorithm is used as solver. Convective terms are treated by QUICK scheme. Water is considered as the working medium for this simulation.

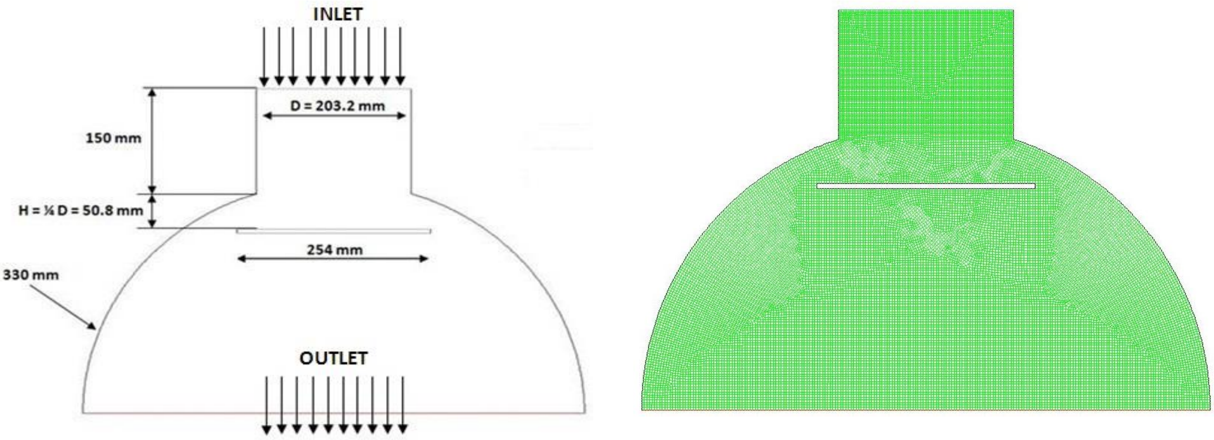


Figure 1. 2-Dimension computational domain and meshed zone

Grid Independence Test and Model Validation

In order to check mesh sensitivity, pressure head loss ($(\Delta P/\rho g)$ occurring in the flow due to impingement of flow on plate is considered. Result is shown in Figure 2(a) where grid appears to be insensitive after 22000 cells. The model is validated with an experimental investigation carried out by Tu and Wood [1] to find pressure variation on an impingement plate where H/D equal to 2 was considered for investigation. A close match between simulated data and experimental data Figure 2(b) depicts.

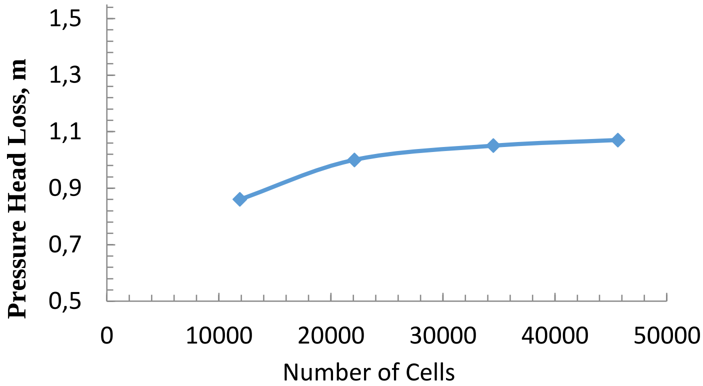


Figure 2(a). Grid independence test

The distance(H) of the impingement plate in a shell and tube heat exchanger from inlet nozzle is significant for better thermal performance of the system. A severe loss of pressure head across conventional impingement plate is undesirable. Keeping Reynolds number equal to 11,000, the H/D ratio is varied to 0.25, 0.55, 1.0, 1.6, and 2.0. The corresponding pressure distribution on the plate is presented in Figure 3. With increase in H/D ratio, the maximum pressure can be seen to occur at the centre of the impingement plate. Maximum pressure reduces with decreasing H/D ratio. Also the pressure distribution above plate gets more uniform with decreasing H/D ratio. This trend is expected due to decreasing inertial forces which is the function of incoming velocity. Due to increasing size of the nozzle, the incoming velocity reduces and approaches zero. As velocity gets lower and lower in magnitude the stagnation effect occurring during impingement dies out.

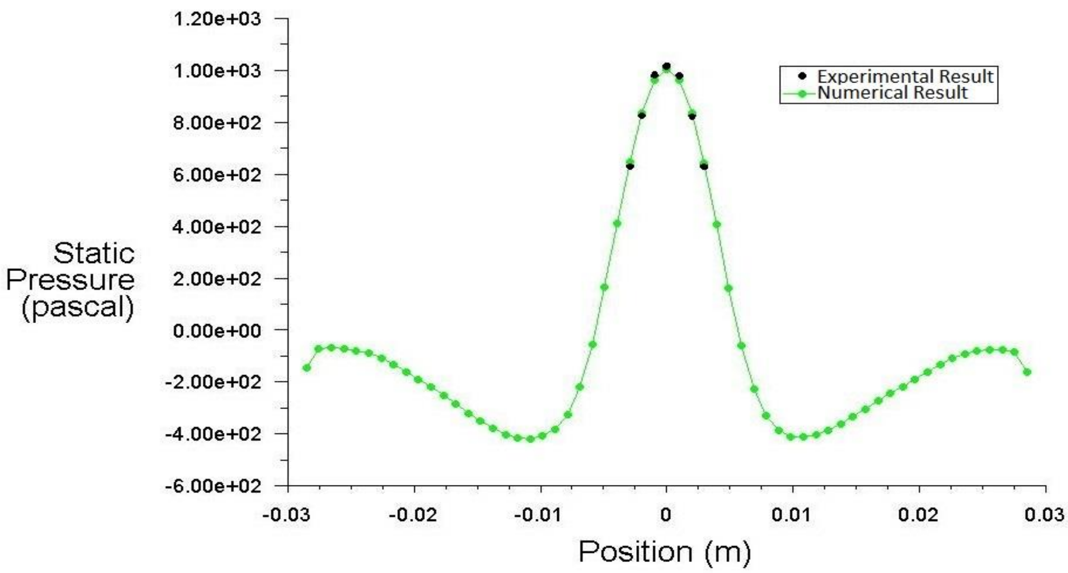


Figure 2(b). Validation of simulated data with experimental

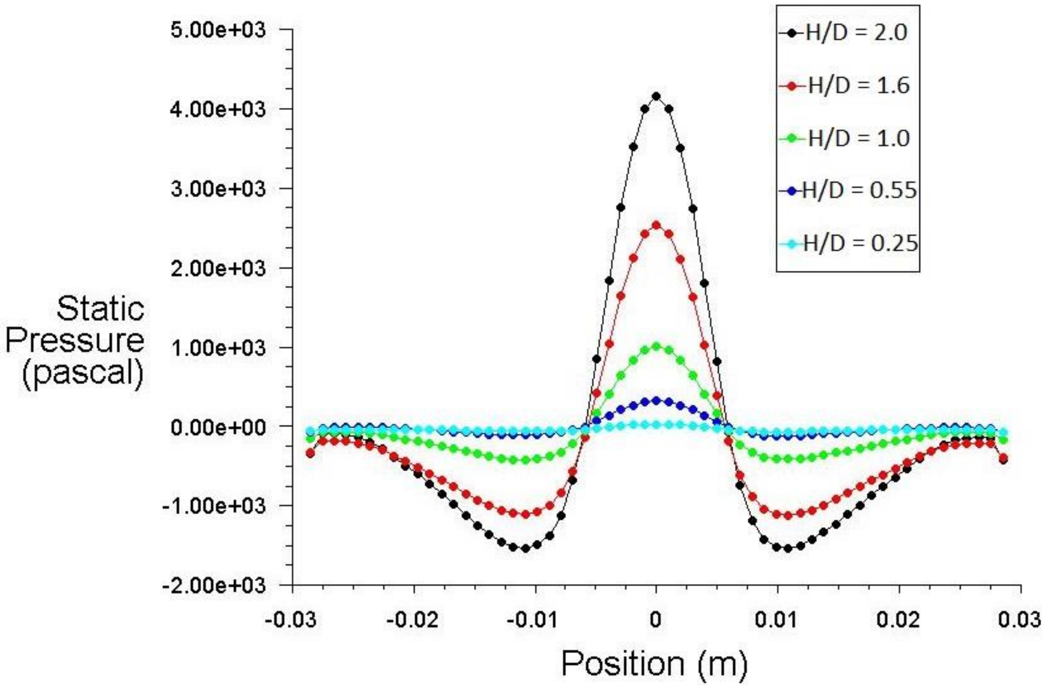


Figure 3. Effect of H/D ratio on the pressure distribution

RESULTS AND DISCUSSION

Investigation of significant parameters which influences the flow structure developing around an impingement plate is presented in this section. To evaluate the effect of impingement plate on the performance of shell and tube heat exchanger the following qualitative/quantitative terms have been considered from flow distribution point of view.

$$\text{Pressure head loss: } (P_2 - P_1)/\rho g$$

It is used to describe the decrease in pressure from one point to another point downstream. Pressure drop is the result of frictional forces on the fluid as it flows through the system. The frictional forces are caused by a resistance to flow. Fluid always flows in the direction of least resistance (less pressure).

$$\text{Kinetic head loss: } (V_2^2 - V_1^2)/2g$$

It is the loss experienced by the flow due to resistance in its path. Velocity data above and below the plate was used to calculate kinetic energy loss. The design with least amount of loss is desirable.

Flow penetration: The lowest point in the tube bank with no stagnation was used to decide the flow penetration. Design with maximum flow penetration is desirable.

Effect of Impingement Plat Size

Tubular Exchanger Manufacturers Association (TEMA) standard recommends impingement plate to be of size 1.26D where D is a nozzle diameter. In order to know the effect of investigation plate size on flow pattern, the plate size of 0.25D, 0.75D, 1.0D, 1.26D and 2.0D is considered here for investigation. The variations of pressure head and kinetic energy losses for different plate size are presented in Figure 4. Pressure head indicates loss of pressure head due to the presence of impingement plate which is calculated by taking pressure difference before and after impingement. Kinetic energy loss is the loss of dynamic potential of flowing stream due to the presence of impingement plate

A drastic change in pressure head can be observed for plate sizes greater than D as shown in Figure 8. At a bigger plate size, fluid rebounds after striking the plate at the plate corners. This increases the pressure head loss. At smaller plate size fluid finds lesser surface area to strike which reduces rebound tendencies of the fluid.

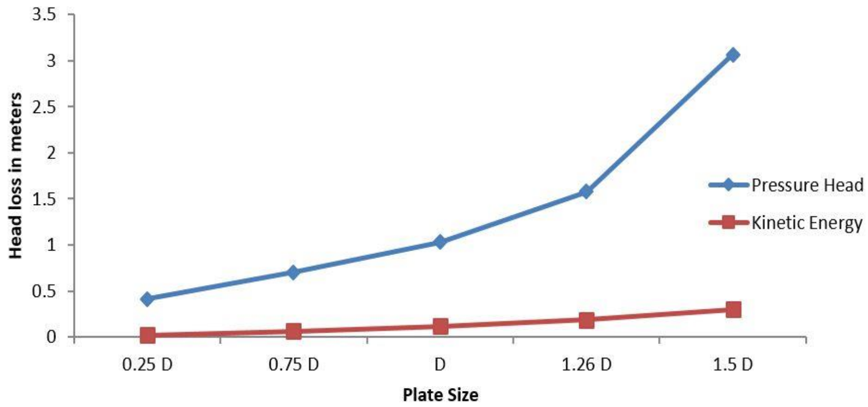


Figure 4. Variation of pressure head and kinetic energy loss with plate size

Effect of Inlet Reynolds Number

To investigate the effect of inflow condition on flow behaviour around the impingement plate, the Reynolds number is varied as 122182, 244364, 366546, 488728 and 610910 which correspond to inlet velocities of 0.5, 1.0, 1.5, 2.0 and 2.5 m/s respectively. Figure 5 shows the variation of pressure head loss and kinetic energy head loss with inlet velocity.

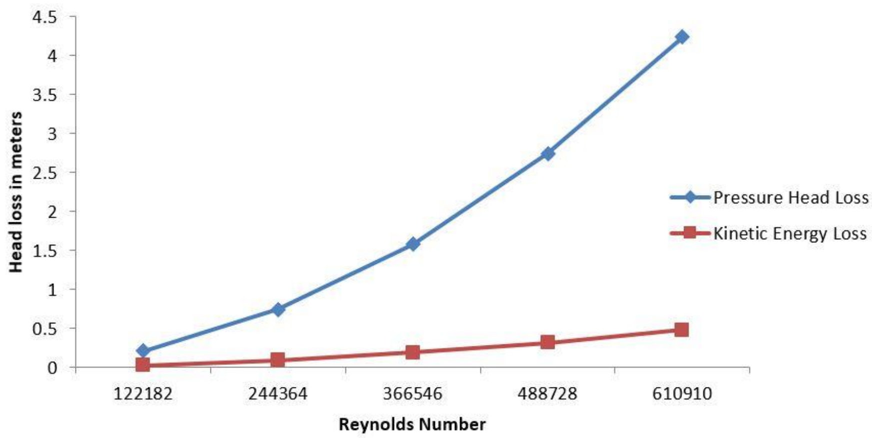


Figure 5. Variation of pressure head loss and kinetic energy head loss with Reynolds number

Increase in Reynolds number increases the flow velocity at inlet which alters the flow pattern. It was observed that both pressure and kinetic energy losses increased monotonically with increasing in Reynolds number..Pressure head losses contribute to the major part of the loss.

Effect of Impingement Plate Geometry

In order to find a better impingement plate design that could provide better flow distribution compared to conventional impingement plate along with satisfying its basic function, several arrangements of impingement plate are considered. A two dimensional sketch of these arrangement is shown in Figure 6. These arrangements aim at improving flow distribution in tube area which is partially shadowed by the presence of impingement plate. Flow condition remains same for all the cases.

(a) Without Tube Arrangement

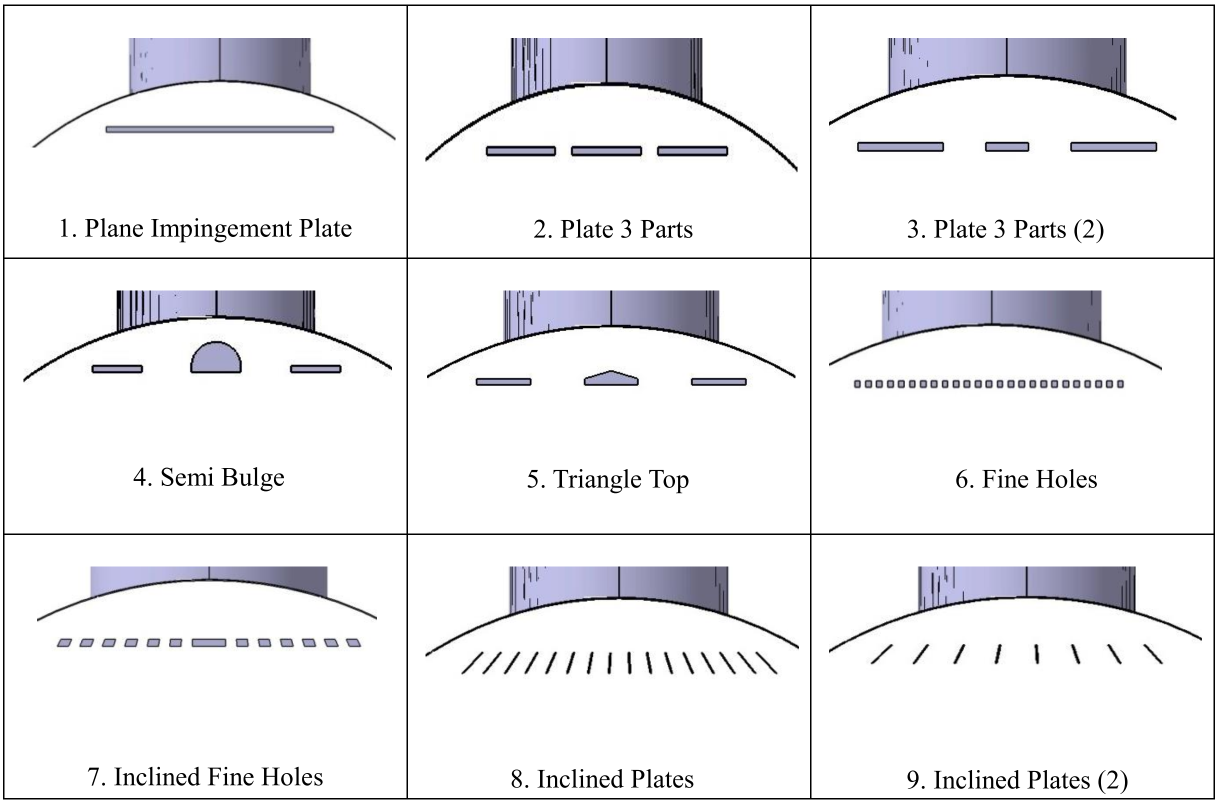


Figure 6. 2D Impingement plate geometries

Considering the evolution of symmetric flow structure around the impingement plate a two dimensional analysis with different geometries is presented here. The plate geometries which are investigated are numbered from 1 to 9 as shown in Figure 6. The plate geometry 1 indicates conventional unmodified plate. Changes in impingement plate geometries is proposed based on the consideration that a significant portion of tube bank remain under utilized due the stagnant fluid zone developing under the impingement plate. Present numerical analysis carried out on the proposed impingement plate geometry investigates the pressure and velocity variation around it. An account of energy loss due to the presence of different plate geometry provides insight into the selection of optimum one. The effect of providing a nozzle kind of perforation in the impingement plate on the flow pattern is depicted in Figure 7 where velocity vectors are compared with that for the conventional plate geometry. The flow gets altered by the perforation leading to minor velocity and pressure changes and indicates the scope for better thermal performance. Velocity contours around impingement plate for different plate geometries 1-9 is shown in Figure 8.

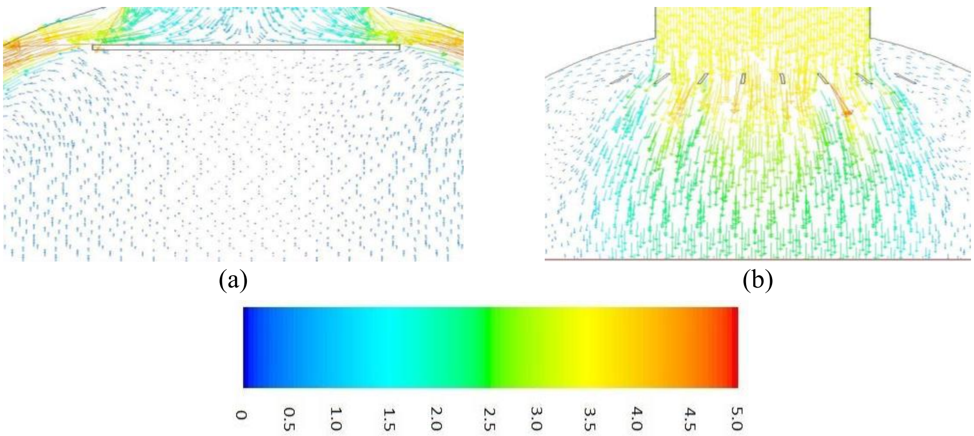


Figure 7. Velocity Vectors for (a) Unmodified and (b) Modified Designs

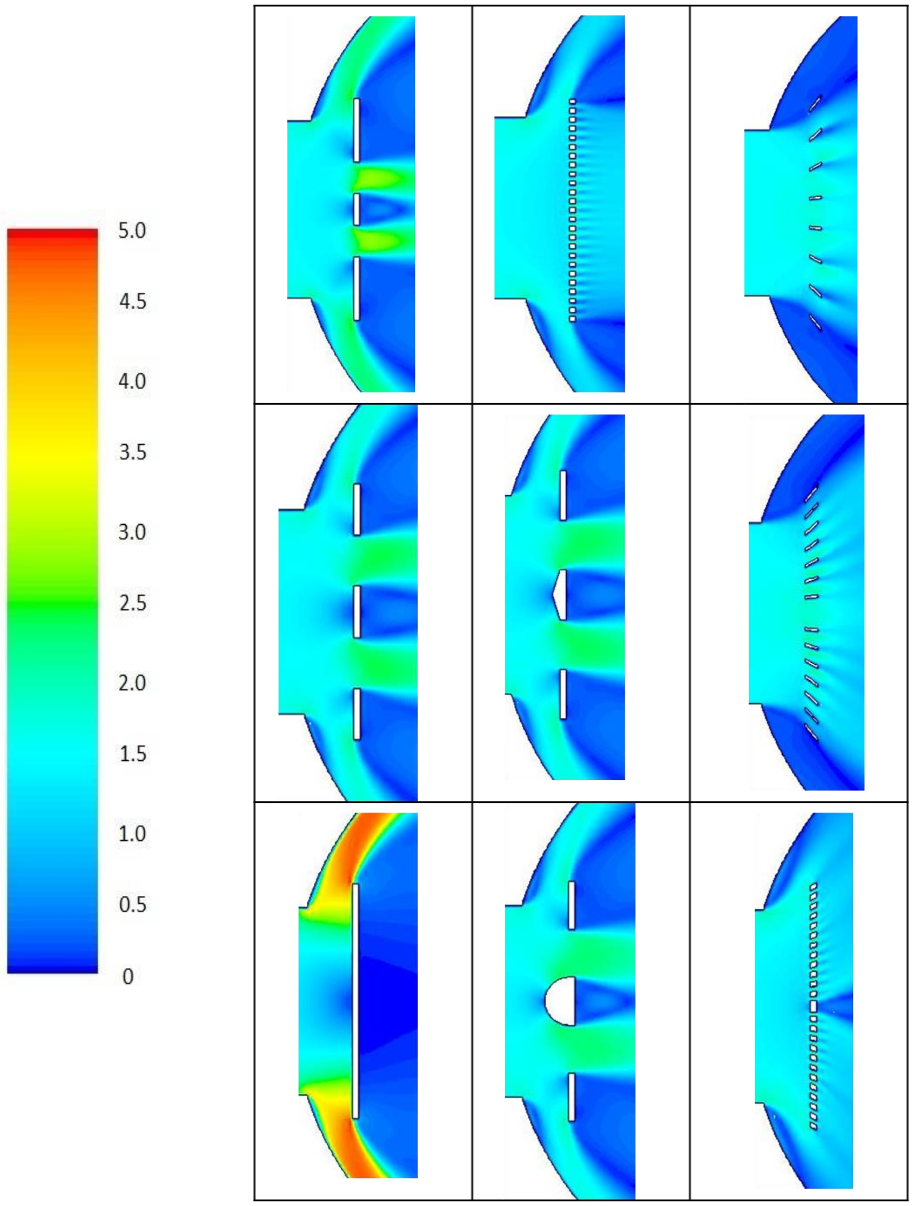


Figure 8. Velocity contours for modified plate designs

A quantitative analysis of average loss of energy is pressure and kinetic head for all geometries of impingement plate is depicted in Figure 9 It is observed that the kinetic head loss is insignificant for most of the plate geometries compared to respective pressure head loss. The effect of perforation is obvious from this figure. The least pressure drop is observed for plate geometry 9.

Figure 10 shows the variation of pressure on top and bottom impingement plate area for the two extreme geometries i.e. 1 and 9, which are designated as unmodified and modified geometry. In the modified geometry, the presence of perforation guiding flows to spread in downward direction does not allow to develop the stagnation zone around impingement plate. A considerable increase in average velocity due to channeling effect is observed. It provides better scope for the heat transfer.

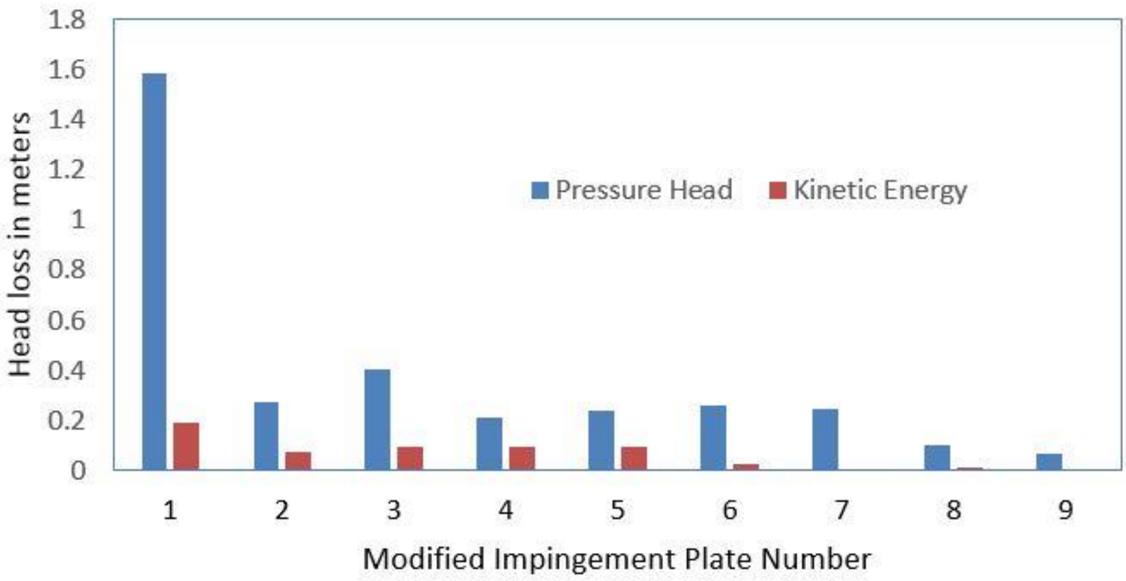


Figure 9. Kinetic and pressure head loss across different impingement plate geometries

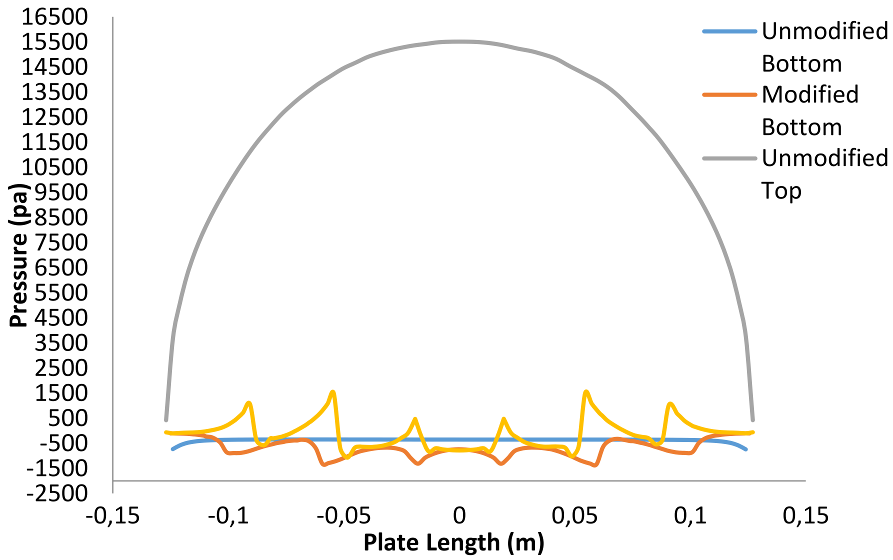
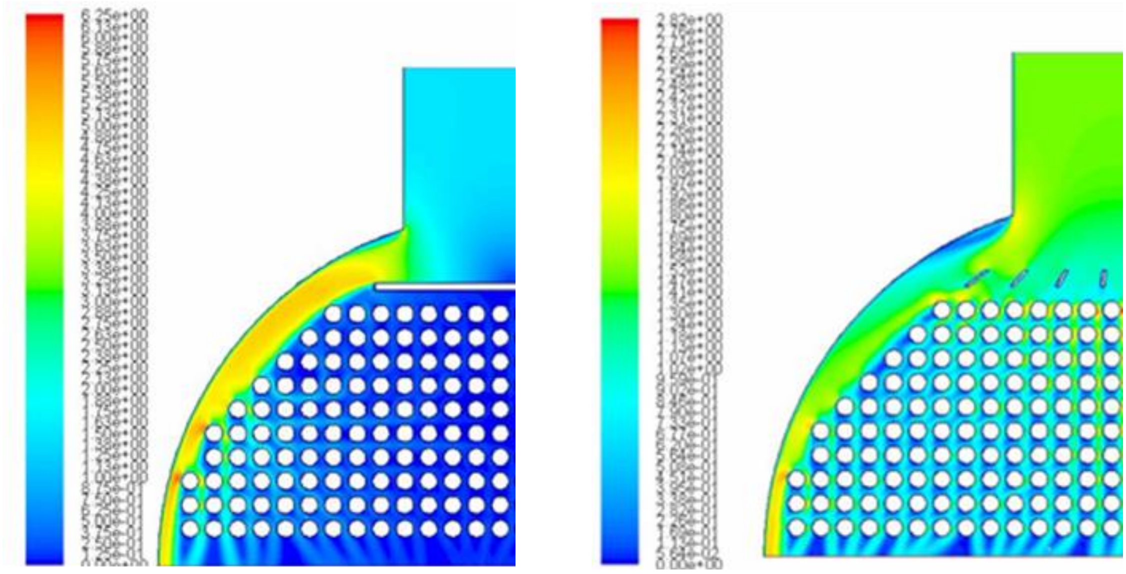


Figure 10. Pressure variation plots combined

(b) With Tube Arrangement:

The placement of tube bank below the impingement plate gets influenced by the shadow effect of the plate which needs to be minimized. A qualitative investigation of the situation is presented in Figure 11. The unmodified version of impingement plate influences a big region of tube bank where fluid is almost stagnant. The convective mode of heat exchange in this region is severely affected leading to lower overall thermal performance of the system. A significant improvement in velocity distribution can be observed for the modified version. Fluid has penetrated almost every corner of plate shadow region providing scope for better heat exchange.



(a) (b)

Figure 11. Velocity contours with (a) Unmodified (b) Modified Plate geometries

CONCLUSION

A two dimensional numerical investigation of flow structures evolving round different layouts of the impingement plate with and without tube bank brings out following important conclusions.

- The pressure distribution on impingement plate becomes more and more uniform as it comes closer to the inlet nozzle
- Size, location and design of the impingement plate are an important consideration while designing the shell and tube heat exchanger for a specific application.
- The size of impingement plate should be close to nozzle inlet size to reduce kinetic head loss and pressure head loss. Compared to kinetic head loss, pressure head loss is more dominant.
- Instead of using simple flat plate geometry as an impingement plat, it is always better to use segmented inclined plate geometry. It provides better flow penetration to tube bank and also better heat transfer due to higher velocity around tube.

NOMENCLATURE

U	Velocity Vector
ρ	Density
p	Local Pressure
ν	Kinematic Viscosity
k	Turbulent Kinetic Energy
ε	Turbulence Dissipation
D	Diameter of Nozzle

REFERENCES

[1] Tu, C.V. and D.H. Wood, *Wall pressure and shear stress measurements beneath an impinging jet*, Experimental Thermal and Fluid Science, 1996. 13: p. 364-373.
[2] Shi, Y.L., M.B. Ray and A.S. Mujumdar, *Computational study of impingement heat transfer under a turbulence slot jet,* Industrial and Engineering Chemistry Research, 2002, 41: p. 4643-4651.
[3] De Langhe, B. Merci, and E. Dick, *Application of RG hybrid RANS/LES model to swirling confined turbulent jets*, Journal of Turbulence, 2006. 7(56): p. 1-19.
[4] Perrotin, T. and D. Clodic, *Thermal-hydraulic CFD study in louvered fin and flat tube heat exchangers*, International Journal of Refrigeration, 2004. 27(4): p. 422-432.
[5] Ozden, E. and I. Tari, *Shell side CFD analysis of a small shell and tube heat exchanger*, Energy Conversion and Management, 2010. 51(5): p. 1004-1014.

[6] Badra, J. A.R. Masri, and M. Behnia, *Enhanced heat transfer from arrays of jets impinging on a moving plates*, Heat Transfer Engineering, 2013, 34(4): p. 361-371.
[7] Karlsson T. and L. Vamling, *Flow fields in shell and tube condensers: comparison of a pure refrigerant and a binary mixture,* International Journal of Refrigeration, 2005. 28(5): p. 706-713.
[8] Al-Anizi, S.S. and A.M. Al-Otaibi, *Double perforated impingement plate in shell and tube heat exchanger,* 7th International Conference on Heat Exchanger Fouling and Cleaning- Challenges and Opportunities. 2007. Tomar, Portugal.
[9] Kandu B, *Semi analytical methods for heat and fluid flow between two parallel plates,* Journal of Thermal Engineering, 2015, 1(3): p. 175-181.