Investigation of the Perforated Tube Configurations of Double-Pipe Heat Exchanger using CFD

Mohammad Asif, Vishwajeet Kureel

INTRODUCTION

Heat exchanger device is used to transmit heat energy from one fluid to another fluid. Condenser and evaporators used in various air conditioning units and refrigerators are very common heat exchangers used in our daily life. Heat exchangers are also used in many other industrial process in which a liquid or gas is required to be either cooled or heated. Boiler and condensers are also examples of large industrial heat exchangers. Heat exchangers are also used in automobiles in the form of radiator and oil cooler. Heat exchangers are abundantly used in chemical and process industries. There is a wide variety of heat exchangers available for the wide range of applications. However, heat exchangers can also be classified on the basis of fundamental design features instead of variety.

Improvised heat transfer technologies have been widely applied to heat exchanger applications like in refrigeration, automobile and process industries, etc in the recent years. The performance of a heat exchanger can be improved by economical design of the heat exchanger. With the use of heat transfer enhancement techniques, economic design of heat exchanger is obtained and thereby reducing the size which generally leads to energy, material and cost savings.

Double Pipe Heat Exchanger

Double pipe heat exchanger is very important in different industrial process. It is constructed by two concentric pipes. Two fluids flow through the inner and outer pipes. Usually fluids flow in opposite directions. This is the simplest type of heat exchanger, named double pipe heat exchanger because of the concentric tube construction. In such heat exchanger one fluid flows under the pipe and the other flows between the pipe and another concentric pipe which surrounds the first. The flow may be parallel or counter in double pipe heat exchanger.

Applications

- Concentric heat exchangers are commonly used in applications that involve relatively low flow rates and high temperatures or pressure for which they are best suited.
- Concentric tube heat exchanger mostly used for material processing, food preparation, and air conditioning.

Advantages

- The main advantage of a concentric tube heat exchanger is its simple design.
- The concentric tube heat exchanger has a low installation cost, ease of maintenance and flexibility.
- It is ideal for fluids that cause fouling.

Disadvantages

- Length of tube can be extended up to a limit.
- The rate of heat transfer is low in concentric tube heat exchangers.
- It posses high cost when considering amount of heat transfer.

In this work I have taken the concentric pipe heat exchanger with perforation in inner pipe. The perforation plate is provided at the middle of the heat exchanger. The perforated plate with 1mm thickness is set to be 5 holes. The liquid benzene is selected fluid to cool to desired temperature and set it up for outer annulus pipe in concentric pipe heat exchanger while water is selected as a cooling medium through which desired temperature of Benzene is achieved . The water is setup in the inner pipe of the heat exchanger.

Computational Fluid Flow Analysis

Fluid mechanics is the branch of science, which deals with fluid flow. The equations which describe the motion of fluid substances are Navier Stokes equations, which were developed by Navier and Stokes in the early 19th century. These equations are set of coupled differential equations and can be solved for a given flow problem by using methods from calculus. But, in practice, these equations are too difficult to be solved analytically. In the past, engineers made further approximations and simplifications to the equation set in order to develop a group of equations that they could solve. Recently, high-speed computers were used to solve approximations to the equations using a variety of techniques like finite difference, finite volume, finite element, and spectral methods. This area of study is referred to as Computational Fluid Dynamics or CFD.

Computational Fluid Dynamics (CFD)

CFD is a computational technology that enables the engineer to study the dynamics of things that flow. Using CFD, one builds a computational model that represents a system or device to be studied. Then the fluid flow physics is applied to this virtual prototype, and the software outputs a prediction of the fluid dynamics. A simplified and straight definition of the computational fluid dynamic would be that, CFD is, in part, the art of replacing the governing equations of fluid flow with numbers, and advancing these numbers in space and/or time to obtain a final numerical description of the complete flow field of interest [11]. The importance of CFD for the development of the fluid dynamic is enormous and it is seen as its “third dimension”, together with the experiments and the pure theory. CFD is a sophisticated analysis technique that not only predicts fluid flow behaviour, but also the transfer of heat, mass, phase change, chemical reaction, mechanical movement, and stress or deformation of related solid structures. These characteristics have rapidly transformed CFD into a very popular tool in engineering analysis.

Advantages of using CFD methods

1. There are many devices and systems that are very difficult to prototype. Often, CFD analysis shows on part of the system or phenomenon happening within the system that would not otherwise be visible through any other means. CFD gives one a means of visualizing and enhanced understanding of the designs.
2. CFD is a tool for predicting what will happen under a given set of circumstances. One provides the input variables and it provides the outcomes. In a short time, one can predict how a design will perform, and test many variations until an optimal result is obtained. To achieve these in physical prototyping and testing (done in the past) would require a huge amount of time and labour. The foresight gained from CFD helps the engineer to design better and faster.

3. The better and faster design or analysis leads to shorter design cycles. Time and money are saved. Products get to the market faster. Equipment improvements are built and installed with minimal downtime. Thus, CFD is a tool for compressing the design and development cycle.

RESEARCH METHODOLOGY

In all of these approaches the same basic procedure is followed.

- Preprocessing
  - ○ The geometry (physical bounds) of the problem is defined.
  - ○ The volume occupied by the fluid is divided into discrete cells (the mesh). The mesh may be uniform or non uniform.
  - ○ The Mathematical modelling is defined
  - ○ Boundary conditions are defined. This involves specifying the fluid behaviour and properties at the boundaries of the problem.
- Processing
  - ○ The simulation is started and the equations are solved iteratively as a steady-state
- Postprocessor
  - ○ Result is generated during processing is used for the analysis and visualization of the resulting solution.
  - ○ Parametric value is being generated.

VALIDATION OF CFD CODE

The present work, a CFD modelling is carried out to observe of the heat transfer rate for concentric pipe heat exchanger equipped with perforated plate at the middle span of inner pipe. This work is validated as below.

In order to show case the capability of the CFD in predicting the fluid flow and heat transfer in the tube equipped with perforated plate, the CFD code is validated with considering the plain tube and the simulated friction factor across the tube are compared with experiment carried out by [2]

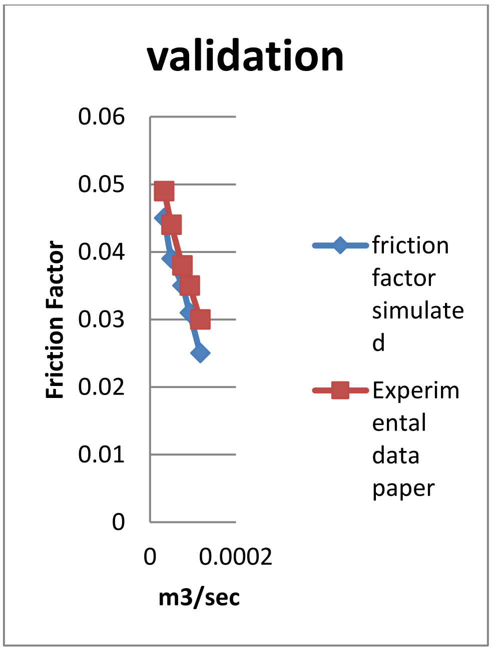


Figure 1 The comparison between the experimental and simulated

The simulated and experimental for friction factor is represented in fig. 06 The simulated result showing the 8% to9.5% percentage of error as compare to experimental data provided by Anas El Maakoul et al. With this percentage of error validated my CFD code for present work.
It can be seen from fig that experimental results are in good agreement with the CFD predicted values. The graph pattern of simulated result is almost same as that of experimental result.

RESULTS ANALYSIS

The numerical investigating is carried out with laminar flow in the inner pipe. The Reynolds no. is less than 2300. The Reynolds number Various from 65 to 80. Although the Reynolds number is less but is it sufficient to capture the effect of perforating material in the pipe.
The mass flow rate of Benzene it kept constant value 0.05 kg/s as it is specified for achieving the cooling of benzene for desired limit of temperature. The mass flow rate of water has been varied with certain diameter of perforating plate hole. The programme is converged in 50 iteration as shown in the fig 02.

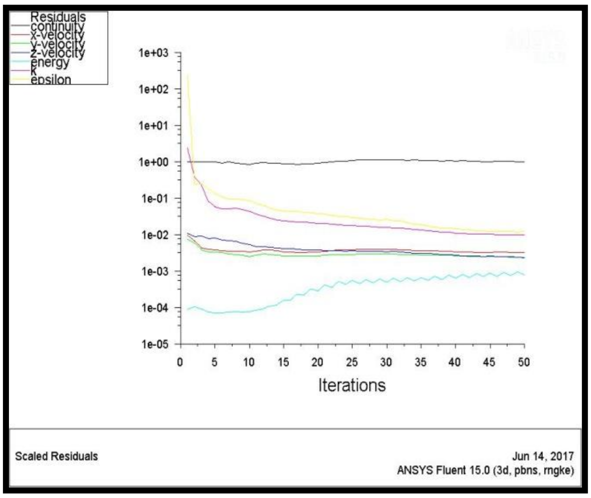


Figure 2 convergence graph

The following graph and temperature distribution identified at various location during numerical investigation.

The fig 3 shows the temperature contours at the mid location of the heat exchanger. It is the location where I have inserted the perforated plate with 5 holes. It can be seen that with the insertion of plate the more temperature gradient at the location. Direction of temperature gradient move toward the perforation area.

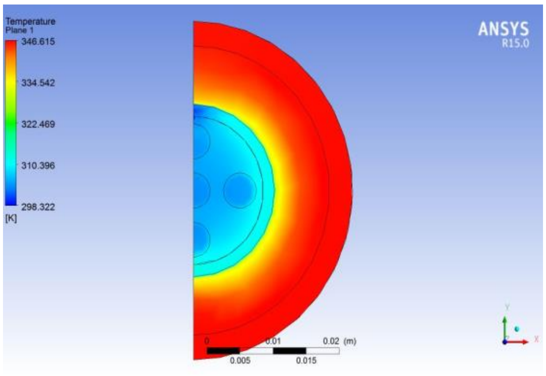


Figure 3 Temperature Contours at perforating plate

In Fig 9 shows temperature contours at the location 800mm from inlet of water. It is location where no perforation is provided, It is clearly visualised that less temperature gradient in inner pipe toward centre of pipe. The temperature gradient exists only near the wall of inner tube.

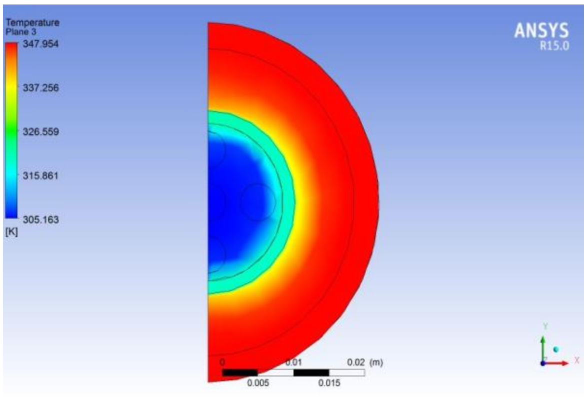


Figure 4 Temperature Contours 800 mm from water Inlet

The fig 5 & 6 shows velocity profile at the region of perforating plate. It is clearly shows that perforating hole leads to the change in the velocity profile in the pipe as compare to other location of the pipe. The change in the velocity would cause to change the heat transfer coefficient hence it increase the heat transfer rate from water to benzene liquid (heat absorbing fluid). Therefore the more temperature gradient in perforating region as compare to other region of the pipe.

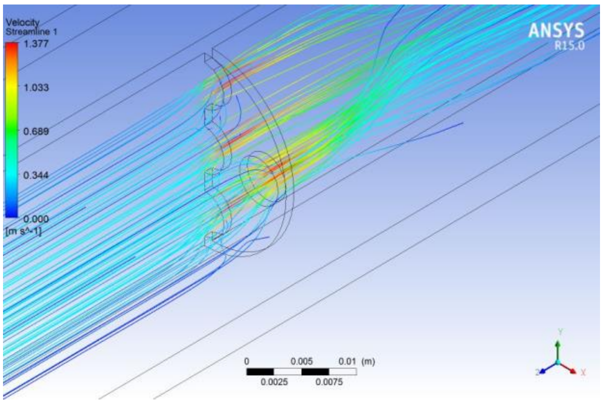


Figure 5 Velocity at Perforating Region.

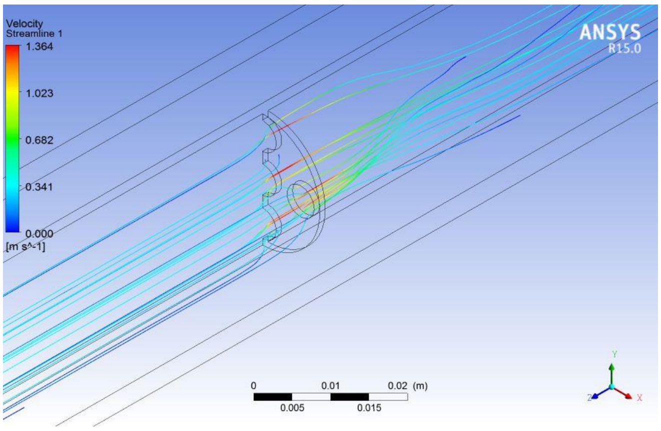


Figure 6 Variation in Velocity at Perforating Region.

Fig 7 shows the graph between mass flow rate of water and its heat transfer coefficient with diameter of perforation on perforating plate. It shows that heat transfer coefficient increases with mass flow rate of water for particular perforating diameter. For 3 mm perforation diameter shows more heat transfer coefficient as compare to other 4mm and 5 mm diameter. It show that as we decrease the perforation diameter the heat transfer coefficient increases and it lead to increase the heat transfer rate from water, Hence it increases the cooling rate of benzene.

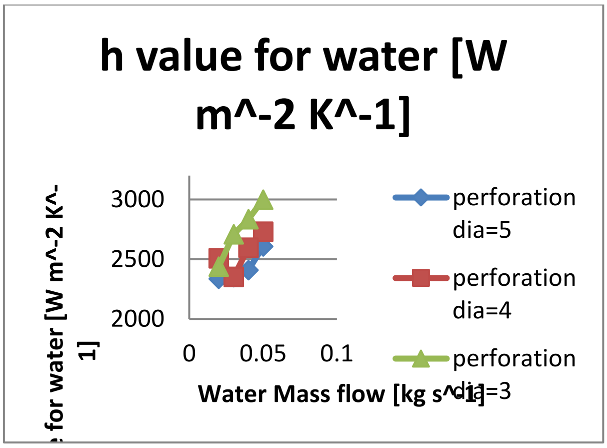
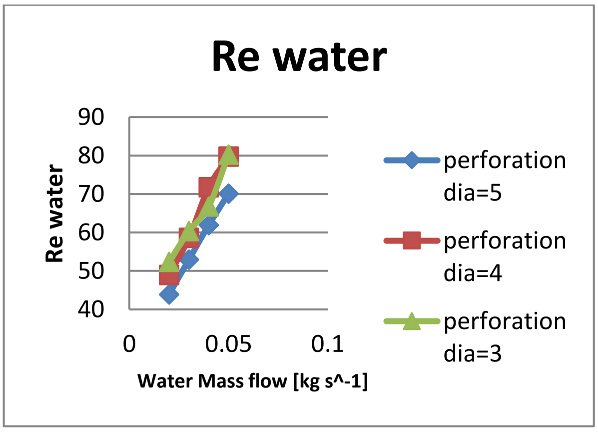


Figure 7 Graph between Heat transfer coefficient and mass flow rate of water



.

Figure 8 Graph between Re no. and Mass flow rate of water

The fig 8 shows graph between Average Reynolds number and mass flow rate of water with different perforation diameter in perforating plate. It depicts that as the increase of mass flow rate of water Reynolds no. is also increased. The Reynolds number is almost same for3 mm and 4 mm perforation diameter. The change in Reynolds no. would increase in heat transfer rate from water.

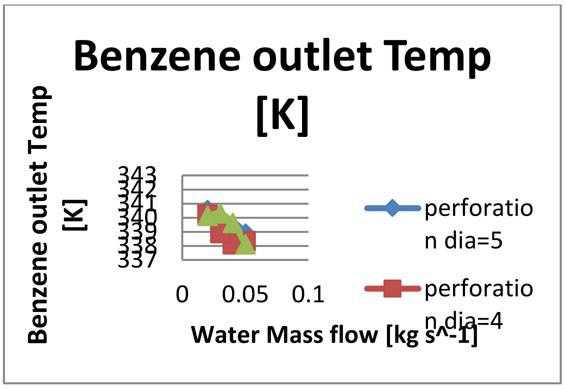


Figure 9 Graph between Benzene outlet temperature and Mass flow rate of water

The fig 9 shows benzene outlet temperature with perforation diameter. For perforation diameter 3mm shows minimum temperature of benzene. Although 4mm diameter is almost same as for 3 mm perforation diameter.

CONCLUSION AND SCOPE FOR FUTURE WORK

The investigations of heat transfer coefficient and heat transfer rate for perforated plate fitted in inner pipe of the concentric pipe heat exchanger at mid span of heat exchanger have been studied. Different diameter of perforation hole in the plate with different mass flow rate of water (inner pipe fluid) has been numerically investigated. In post analysis it has been identified that for increase of mass flow rate of water increase the heat transfer rate for lower diameter of perforation. Hence decrease of perforation diameter enhance the heat transfer rate as compare to higher diameter of perforation

The present work can further be extended with more number of perforation plates. The perforation diameter in perforating plate may be varied at different location. In this work i have considered the circular perforation has been used but it can be modified with notched circular hole in the perforation plate. It may increase the turbulent and hence heat transfer. These variant perforating plates can be used for heat transfer augmentation studies also in refrigeration system.

REFERENCES

1. Mohamad Omidi et al. “A comprehensive review on double pipe heat exchangers” Applied Thermal Engineering Science Direct

2. Anas El Maakoul et al. ” Numerical design and investigation of heat transfer enhancement and performance for an annulus with continuous helical baffles in a double-pipe heat exchanger.” Energy Conversion and Management 133 (2017) 76–86 ScienceDirect

3. M. Sheikholeslami et al. “Experimental study on turbulent flow and heat transfer in an air to water heat exchanger using perforated circular-ring” Experimental Thermal and Fluid Science 70 (2016) 185–195 ScienceDirect

4. P.V. Durga Prasad et al. "Investigation of Trapezoidal-Cut Twisted Tape Insert in a Double Pipe U-Tube Heat Exchanger using Al2O3/Water Nanofluid." Procedia Materials Science 10 (2015) 50 – 63, 2nd International Conference on Nanomaterials and Technologies (CNT 2014)

5. Ojha Pramod Kailas et al. " Design and experimental analysis of pipe in pipe heat exchanger" | IJMER | ISSN: 2249–6645 | Vol. 5 | Iss.3| Mar. 2015 | 42|

6. Tzu-Chen Hung a et al. "Optimal design of a concentric heat exchanger for high-temperature systems using CFD simulations" Applied Thermal Engineering 75 (2015) 700e708, ScienceDirect

7. Jibin Johnson et al. "CFD Analysis of Double Pipe Heat Exchanger" International Journal of Science, Engineering and Technology Research (IJSETR), Volume 4, Issue 5, May 2015, ISSN: 2278 – 7798

8. H.A. Mohammed et al. "Heat transfer enhancement of nano fluids in a double pipe heat exchanger with louvered strip inserts." International Communications in Heat and Mass Transfer xxx (2012) xxx–xxx, ICHMT-02703; No of Pages 11 SciVerse ScienceDirect

9. Ebru Kavak Akpinar "Evaluation of heat transfer and exergy loss in a concentric double pipe exchanger equipped with helical wires." Mechanical Engineering Department, Firat University, TR-23119 Elazig, Turkey Received 24 December 2004; received in revised form 14 July 2005; accepted 24 December 2005
10. Shah RK & Sekulic DP. "Fundamentals of heat exchanger design". John Wiley &Sons; 2003.