Research Paper

**USING COMPUTATIONAL FLUID DYNAMICS INVESTIGATE THE VARIOUS HEAT TRANSFER CHARACTERISTICS AND PRESSURE DROP IN THE COMPACT HEAT EXCHANGER**

DINESH SOLANKI ¹, RAM KUMAR VISHWAKARMA ², BHOOPENDRA PATEL ³

1. M.TECH, SCHOLAR, DEPARTMENT OF MECHANICAL ENGINEERING, SVNU, SAGAR, (M.P.)
2. ASST. PROF, DEPARTMENT OF MECHANICAL ENGINEERING, SVNU, SAGAR, (M.P.)
3. ASST. PROF., DEPARTMENT OF MECHANICAL ENGINEERING, SVNU, SAGAR, (M.P.)

**ABSTRACT**

Heat transfer is the most important industrial processes, distributed by heat exchangers in the different types of flow applications. There are various theoretical models are available for understand the heat transfer mechanisms. Due to increasing need for industrial applications requiring the design and improvement of heat exchangers. Heat exchangers are the wide range of applications together with power plants, nuclear reactors, refrigeration systems and air-conditioning systems, heat recovery systems, and chemical process and food industries. Helical coil configuration is incredibly effective for heat exchangers because they can accommodate a large heat transfer space in a small area, with high heat transfer coefficients. Main purpose of this study is to calculate the pressure drop and heat transfer characteristics for liquid ammonia owing through the desired interrupted fin heat exchanger at a given mass flux and simulate the average Stanton number considering all factors for turbulence modeling. We find by CFD results (contour images) pressure drop, static temperature, velocity profile, total energy turbulence kinetic energy etc.

**KEYWORDS**

Heat Exchanger, CFD, Mass Flux, Heat Transfer, Turbulence Modeling etc.
INTRODUCTION

A heat exchanger is a mechanical setup utilized for the procedure of heat trades between 2 or additional liquids that are at different temperatures. Heat exchange and trade among streaming liquids could be a standout amongst the most widely known and important physical procedure of concern and heat exchangers are utilized as a section of assortment kind of establishments and arrangements. Heat exchangers are valuable in varied designing procedures like individuals with vital power plants, refrigerating and aerating and cooling frameworks, conservative heat exchanger atomic force plants, nourishment preparing plants, concoction reactors and aeronautical applications. The explanation of building a heat exchanger is to induce effective strategy for modification of heat beginning with one liquid then onto following temperance of direct or in direct contact. There are 3 ways of modification of heat conductivity, convection and radiation. In initial 2 methods of heat exchange any medium is needed yet third one would like no medium, conductivity happen when heat exchange robust body to a different strong body and convective heat exchange occur when heat is exchanges liquid to strong or the other approach around. Be that as it could, in radiation no medium is needs the change heat.

TUBULAR HEAT EXCHANGERS:

Tubular heat exchangers are worked off for the most part of roundabout tubes albeit some other geometry has likewise been utilized as a part of various applications. This sort of structure offers a lot of adaptability in outline as the planning parameters, for example, width, length and the course of action can be effortlessly altered. These heat exchangers are utilized for liquid to-liquid (stage changing like build-up or dissipation) heat exchange. Again this sort is arranged into shell and tube, twofold pipe and winding tube or Shell and Tube heat exchangers.

SHELL AND TUBE HEAT EXCHANGER:

Shell And Tube heat exchangers are better than other straight heat exchangers when utilized in heat exchange applications. These Shell And Tube upgrading the heat exchange rate in light of radiating power because of the ebb and flow geometry of the comes about auxiliary stream improvement.

LITERATURE

Mrs. Shiva Kumar, K. Vasudevkarath (2013) have dealt with both straight tube and Shell And Tube heat exchanger. They saw that temperature drop for Shell and Tube heat exchanger is higher than the straight tube heat exchanger. This is a result of arch impact of the Shell and Tube. Liquid stream in the external layer of the channel moves speedier than the liquid stream in the inward layer. This difference in the speed will lay in an optional stream by which heat exchange will be expanded. [1] Fakoor, (2013) contemplated the weight drop highlight of nano liquid goes inside a vertical Shell And tube for laminar stream state.
Investigations were led by differencing the pitch circle distances across furthermore the tube widths. Results demonstrate that utilizing Shell And Tube tubes rather than straight tubes expands the weight drop quickly. [2] S.D. Sancheti, Dr. P.R. Suresh (2012) completed that the utilization of consistent qualities for the heat and transport properties of the heat transport medium results in projection of off base heat exchange coefficient. Likewise for projection of heat move in a state of liquid-to-liquid heat exchange, as it happens for the situation in a heat exchanger, subjective limit circumstance, for example, steady divider temperature consistent heat flux are not pertinent. In this condition it is key to display the hardware considering conjugate heat exchange. [3] Pramod Purandare (2012) considered that a near examination of the diverse relationships given by various specialists for Shell And Tube heat exchanger. They recognize that the Shell And Tube are effective for low Re. Likewise the extent of tube measurement to distance across ought to be sufficiently huge for substantial intensities of optional liquid keep running inside the tubes. [4] Ivaan Di Piazza (2010) manages computational results for turbulent stream and heat exchange along bended channels. Nusselt number and weight drop were ascertained by utilizing distinctive turbulent models. Out of those RSM-ω gave better results when contrasted with alternate models in the projection of nusselt number and heat exchange. [5] J.S. Jayakumar (2008) completed a test investigation of liquid to liquid heat exchange however a Shell And tube. Heat exchange qualities were likewise contemplated utilizing CFD code familiar. They watched CFD expectations coordinate sensibly with trial results for every single working condition. [6] Rahul Karath, Nitin Bhardwaj, R.S. Jha (2009) has manages the geometry of the heat exchanger and worked with enhancing a relationship for heat exchange coefficient for stream between concentric Shell And Tube s. They have build the crevice between concentric s when contrasted and the test results information. What's more, found that hole and tube measurement is the most imperative parameters for the heat exchange coefficient.

**METHODOLOGY**

Pressurized liquid ammonia at 240 K is to be heated in the heat exchanger and that the fin walls are at a constant temperature of 350 K. The mass flux of liquid ammonia through the heat exchanger is 303.14 kg/s-m2 and the hydraulic diameter is 3.51 mm. Considering the viscosity of liquid ammonia, 0.000152 kg/m-s, the Reynolds number based on hydraulic diameter is 7000 and falls in the weakly turbulent region (i.e., low Reynolds number turbulent regime).

A sketch of the geometry, with dimensions in mm, is shown in Figure 1. The geometry contains symmetry boundary conditions at the top and bottom planes.
Periodic boundaries will be used at the beginning and ending planes of the domain and a periodic mass flow rate of 1.385 kg/s will be used. The performance of this heat exchanger will be expressed in terms of a drag coefficient, CD and a Stanton number.

\[
CD = \frac{D}{\frac{1}{2} \rho A U^2}
\]

The compact heat exchanger for this problem utilizes interrupted and staggered fins that enhance the heat transfer relative to a continuous fin heat exchanger by promoting turbulent mixing in the wake region behind each fin and taking advantage of the relatively high heat transfer coefficients as the boundary layers continuously re-form on the interrupted fins.

Computational fluid dynamics (CFD) is a computer-based simulation method for analyzing fluid flow, heat transfer, and related phenomena such as chemical reactions. This project uses CFD for analysis of flow and heat transfer. Some examples of application areas are: aerodynamic lift and drag (i.e. airplanes or windmill wings), power plant combustion, chemical processes, heating/ventilation, and even biomedical engineering (simulating blood flow through arteries and veins). CFD analyses is carried out in the various industries are used in R&D and manufacture of aircraft, combustion engines, as well as many other industrial products. It can be advantageous to use CFD over traditional experimental-based analyses, whereas experiments cost is directly proportional to the number of configurations desired for testing, unlike with CFD. In CFD large amounts of results can be produced at practically with no other expense. In this way parametric studies to optimize equipment are very inexpensive with CFD when compared to experiments.

**COMPUTATIONAL FLUID DYNAMICS**

Computational fluid dynamics, sometimes abbreviated as CFD, could be a branch of mechanics that use numerical analysis and algorithms to unravel and analyze issues that involve fluid flows. Computers are accustomed perform the calculations needed to simulate the interaction of liquids and gases with surfaces outlined by boundary conditions. With high-speed supercomputers, higher solutions may be achieved. Current analysis yields software system that improves the accuracy and speed of
complicated simulation situations like sonic or turbulent flows. Initial experimental validation of such software system is performed employing a structure with the ultimate validation coming back in complete testing, e.g. flight tests.

**RESULTS**
CONCLUSION

Our main purpose of study is to evaluate the pressure drop and heat transfer characteristics for liquid ammonia owing through the specified interrupted fin heat exchanger at a given mass flux.

We simulate the average Stanton number considering all factors for turbulence modeling, as well as our time and hardware constraints. As we from the CFD results (contour images) pressure drop, static temperature, velocity profile, total energy turbulence kinetic energy etc.

1. The pressure contours show a region of high pressure near the stagnation point at the front of the fin, and regions of relatively low static pressure where the flow accelerates around the curved front portion of the fin.

2. The velocity contours show the stagnation and flow acceleration regions at the front of the fin, the growth of a thin boundary layer along the length of the fin, and a wake region immediately downstream of the fin. The wake region promotes turbulent mixing and enhances overall heat transfer.

For this flow rate, the wake region settles out before the next fin is encountered, and ensures that a new boundary layer with a relatively high heat transfer coefficient will grow on subsequent fins.

In this tutorial liquid ammonia owing through an interrupted fin heat exchanger is simulated using ANSYS FLUENT and the pressure drop and heat transfer characteristics are evaluated. For liquid ammonia at Re = 7000 over heat exchanger surface.

REFERENCES


4. Pramod S. Purandare, Mandar M. Lele Rajkumar Gupta, 2012,


12. R. Thundil Karuppa Raj, Manoj Kumar S., Aby Mathew numerical analysis of Shell And Tube heat exchanger using cfd techniques VOL. 9, NO. 3, MARCH 2014 ISSN 1819-6608
