International Journal of Technology Research and Management
ISSN (Online): 2348-9006
Vol 2 Issue 8 Aug. 2015

# Cfd Analysis to Optimize the Heat Transfer Rate of Helical Fin-Tube Heat Exchanger with Different Fin Pitch

Sheeraz khan[1], Prof. Ravi vishwkarma[2]
Department of Mechanical Engineering[1]
Assistant Professor Department of Mechanical Engineering[2]
Shree Institute Of Science & Technology Bhopal MP( India)[1, 2]
Shirazkhan.4oct@gmail.com[1]

***Abstract:*** *The demand for effective and compact heat exchangers are continuously increasing in the automotive industry, refrigeration and air-conditioning applications has necessitated the use of various interrupted surfaces to augment air-side heat transfer. Fins are normally used to increase heat transfer coefficient and increasing efficiency of heat exchanger. We are using computational fluid dynamics (CFD) suitable software for analyse the heat transfer rate of heat exchanger with different boundary conditions, we can analyse easily all data as compared to experimental setup. And find out best feasible solution by CFD analysis. The objective of this study was to numerically simulate the heat transfer and flow characteristics for a helical fin-tube heat exchanger for different fin pitch (p) and different Reynolds no.(Re) using CFD analysis software Ansys FLUENT . The computational domain consists of air and copper pipe with external helical fins of copper material. The air inlet temperature are taken as constant & outlet air temperature , pressure drop , heat transfer coefficient ,nusselt number , friction factor are calculated subsequently with respect to different Reynolds number. .In this study we numerically simulate the heat transfer rate of helical fin of helical fin-tube heat exchanger for different fin pitch (p) and different Reynolds no.(Re) using CFD analysis software Ansys FLUENT.*

***Keywords:*** *CFD, Helical Fin-tube heat exchanger, Air velocity effect, Effective heat transfer rate.*

## I. Introduction

### 1.1 HEAT EXCHANGER:

A heat exchanger is a perfect for efficient heat transfer from one medium to another. The medium may be separated by a solid wall to prevent mixing or they may be in direct contact. They are widely used in space heating, refrigeration system, air conditioning system, power plants, chemical plants, petrochemical plants, petroleum refineries, natural gas processing, and sewage treatment. The classic example of a heat exchanger is found in an internal combustion engine in which oil using as engine coolant flows through radiator coils and air flows past the coils, which cools the coolant and heats the incoming air.

Heat exchangers are off-the-shelf equipment targeted to the efficient transfer of heat from a hot fluid flow to cold fluid flow, in most cases through an intermediate metallic wall and without moving parts. We are focus on the thermal analysis of heat exchangers, but proper design and use requires additional fluid dynamic analysis (for each fluid flow), mechanical analysis (for closure and resistance), materials compatibility, and so on.

Heat losses of a whole heat exchanger and environment can be neglected in comparison with the heat flow between both fluid flows; i.e. a heat exchanger can be assumed as a globally adiabatic.

Fig 1.1 heat exchanger

Saving material and energy are common objectives for optimization. Material selection of fins is very important factor for effective heat transfer. The optimization function can consider minimum weight for a specified heat flow, minimum mass, minimum pressure drop etc).we can increase convection coefficient with the help of growing the fluid velocity, widen temperature difference between surface and fluid or increase the surface area across which convection occurs. Extended surfaces, in the form of radial fins is common in applications where the need to enhance the heat transfer between a surface and an adjacent fluid exist

Fig 1.2 Fin tube heat exchanger

Fins are commonly used in extended surface exchangers. Conventional fin-tube exchangers often characterize the considerable difference between liquids' heat transfer coefficients. In a gas-to-liquid exchanger, the heat transfer coefficient on the liquid side is generally one order of magnitude higher than that on the gas side. To minimize the size of heat exchangers, fins are used on the gas side to increase the surface area and the heat transfer rate between the heat exchanger surface and the surroundings. Both the conduction through the fin cross section and the convection over the fin surface area take place in and around the fin. When the fin is hotter than the fluid to which it is exposed then the fin surface temperature is generally lower than the base (primary surface) temperature. If the heat is transported by convection to the fin from the ambient fluid, the fin surface temperature will be higher than the fin base temperature, which in turn reduces the temperature differences and the heat transfer through the fin. Exchangers with fins are also used when one fluid stream is at high pressure. The temperature value is limited by the type of material and production technique. All above causes that finned tube heat exchangers are used in different thermal systems for applications where heat energy is exchanged between different media. Applications range from very large to the small scale (tubes in heat exchangers, the temperature control of electronic components). The subject, who is investigated in the chapter, is inspired by the increasing need for optimization in engineering applications, aiming to rationalize use of the available energy. The performance of the heat transfer process in a given heat exchanger is determined for different fin profiles, considering the fluid flow as a variability often neglected for the fin optimization. The optimization task, defined in the chapter, is to increase heat transfer rates.

### 1.2 TYPES OF HEAT EXCHANGER

- ***Double pipe heat exchanger***
- ***Shell and tube heat exchanger***
- ***Plate heat exchanger***
- ***Finned Tube Heat Exchanger***
- ***Air Cooled Heat Exchanger***

## II. COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics (CFD) is a simulation method based on computer to analyze fluid flow, heat transfer, and related fact such as chemical reactions. This assignment uses simulation technique for analysis of flow and heat transfer. Examples of relevance areas are specially in aerodynamics (i.e. aero planes or windmill wings), compound process, heating/ventilation, and even biomedical manufacturing (simulating blood flow through arteries and veins). CFD analyses carried out in the many industries are used in R&D and aircraft manufacturing, ignition engines, as well as many other industrial products. It may be advantageous to use CFD over habitual experimental based analyses, since experiment has a cost which is directly proportional to the number of configurations required for testing, different with CFD, where many results can be obtained at practically no added expense. In this way, parametric study to optimize apparatus is not costly with CFD when compared to experiments. This section for a moment describes a universal concept and theory connected to using CFD to analyze fluid flow and heat transfer, as related to this assignment. It starts with a review of the kit needed for carrying out the CFD analyses and the processes required, followed by a summary of the governing equations and turbulence models and finally a discussion of the discretisation schemes and solution algorithms is presented. This segment describes the CFD kits needed to carry out a simulation and the process one follows in order to solve the problem using CFD. CFD technique fundamentally consist of three elements: the pre-processor, processor, and post-processor. There is a variety of industrial CFD software available such as Fluent, Ansys CFX, ACE, as well as a broad range of appropriate hardware and associated costs, depending on the complexity of the mesh and size of the calculations. Industrial CFD correspondence can cost up to about $20000 (US Dollars) per year for licenses, maintenance, and support. Complicated transient cases with fine meshes will require more powerful computer processors and RAM than simpler cases with rough meshes. A typical engineering workstation (i.e. 32 GB processing RAM with quad processors) at a cost of approximately $3000-$5000 (US Dollars), or a combination of several processors running in parallel, is probably the minimum investment needed to get started. The work for this project was carried out on a HP Pavilion laptop with dual processors totaling 2 GHz RAM, running on Linux Operating System downloaded free from Caelinux. The download from Caelinux included open-source software Salomé for geometry construction and meshing, Open FOAM for the CFD calculations, preview for visualization of results, along with other useful scientific and mathematics related software. Calculations for this project were carried out for approximately 50,000 cells (CFD calculations are often made for 1-2 million cells – or more). On my system, the steady-state solvers took between 1-3 hours to finish calculations, while the transient simulation took 2-3 days running in parallel on both processors. One of the purposes of this project is to use all open-source CFD software instead of commercial software for the simulations. This type of software is advantageous for smaller companies to use, as the cost of commercial CFD package licenses can be prohibitive. To run a simulation, three main elements are needed:

1. Pre-processor: A pre-processor is used to define the geometry for the computational domain of interest and generate the mesh of control volumes (for calculations). Generally, the finer the mesh in the areas of large changes, the more accurate the solution. Fineness of the grid also determines the computer hardware and calculation time needed. The open-source pre-processor used for this project is called Salomé.

2. Solver: The solver makes the calculations using a numerical solution technique, which can use finite difference, finite element, or spectral methods. Most CFD codes use finite volumes, which is a special finite difference method. First the fluid flow equations are integrated over the control volumes (resulting in the exact conservation of relevant properties for each finite volume), then these integral equations are discretised (producing algebraic equations through converting of the integral fluid flow equations), and finally an iterative method is used to solve the algebraic equations. (The finite volume method and discretisation techniques are described more in the next sections. Open FOAM CFD code is used for solving the simulations in this project.

3. Post-Processor: The post-processor provides for visualization of the results, and includes the capability to display the geometry/mesh, create vector, contour, and 2D and 3D surface plots. Particles can be tracked throughout a simulation, and the model can be manipulated (i.e. changed by scaling, rotating, etc.), and all in full color animated graphics. Para View is the open-source post-processor used for this project

## III METHODOLOGY

In this section method which is used in the research is explained. In this research a CFD (Computational Fluid Dynamics) technique is used to optimize the heat transfer rate of helical fin of helical fin-tube heat exchanger.

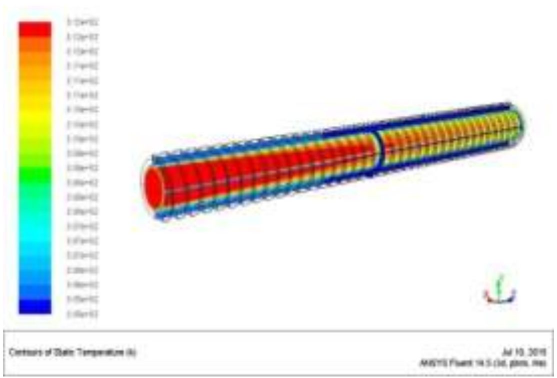
## IV. OUTCOMES

The result from this dissertation work is that the simulation technique (CFD) gives more accurate and consistent result as compare to the experimental technique. In this dissertation work it is analyzed that the experimental technique is time consuming and also the results are also not consistent so it is beneficial to use the simulation technique in case of flow analysis. This simulation analyzed the effective heat transfer rate from helical tube fin in the fin and tube heat exchanger. This study was to numerically simulate the heat transfer and flow characteristics for a fin-tube heat exchanger using CFD codes FLUENT. The computational domain consists of air and copper pipe with externally helical fins of copper material. The hot air is assumed to flow through the pipe at constant air inlet temperature (312.65 k). The air inlet temperature are taken as constant for all cases & outlet air temperature , pressure drop ,heat transfer coefficient, nusselt number, friction factor are calculated below subsequently with respect to different Reynolds number.

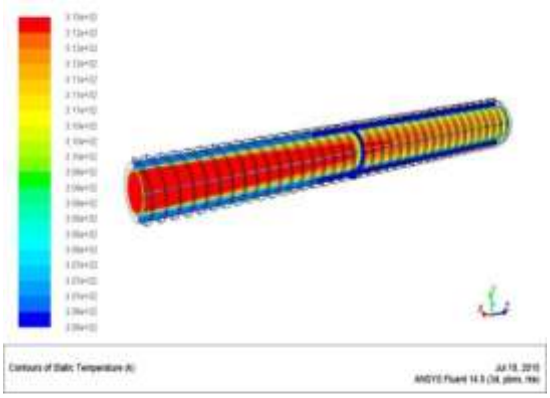
During CFD analysis with different velocities of air passing over the fin and tube and obtained temperature contours at different values of fin pitches (p) and different Reynolds number (Re). I would like to show some results from case-I, case-II , case -III and case –IV .

**TEMPERATUER COUNTOURS RESULTS**

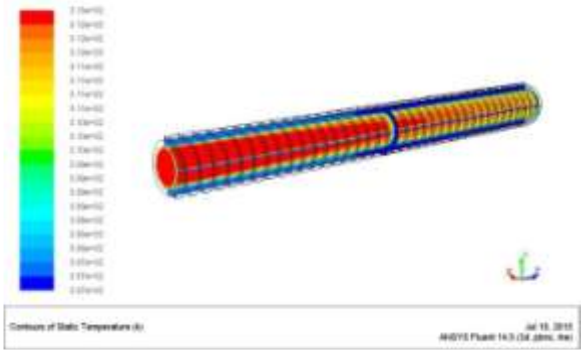
**For constant air inlet temperature condition the temperature contours are shown in below figure 5.1.**



**Re=4000**

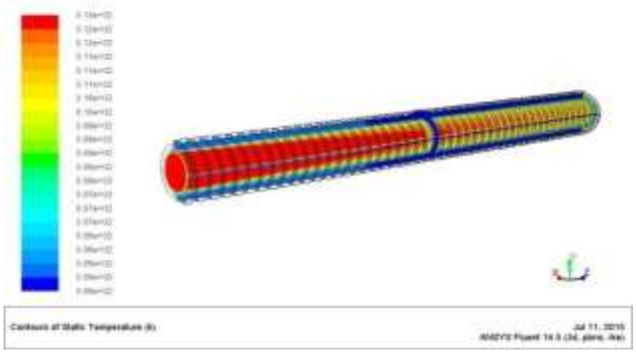


**Re=6000**

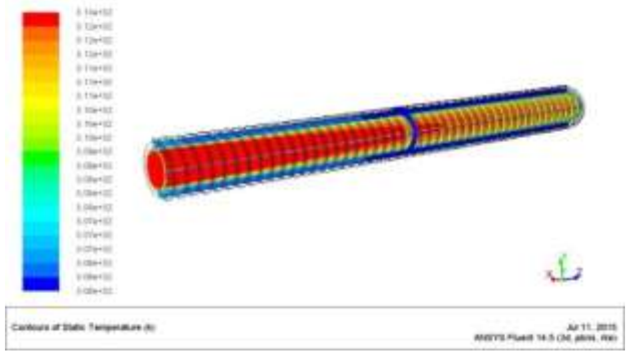


**Re=8000**

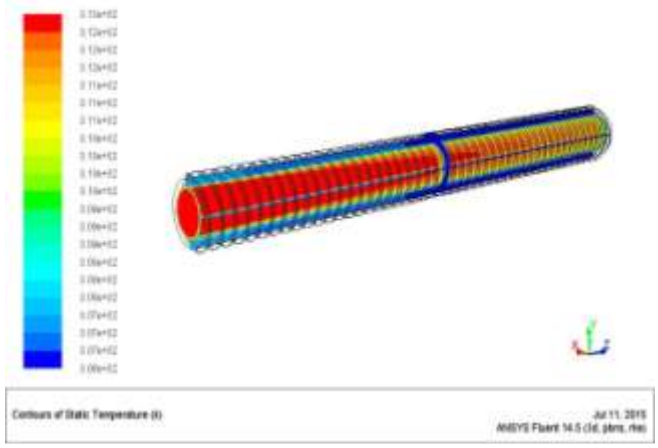
**Figure 4.1(a) Temperature contour for pitch(p) = 10 at constant inlet temperature($t_i$) = 312.65 K**



**Re=4000**

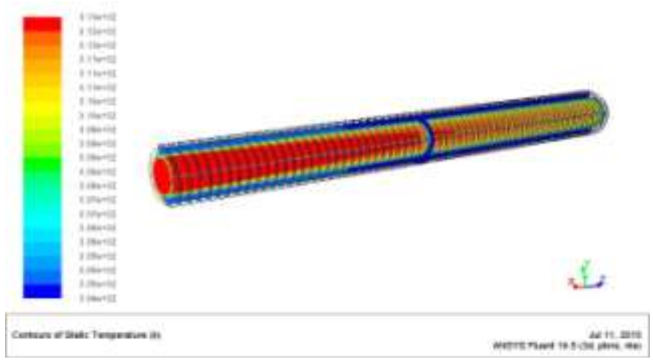


**Re = 6000**

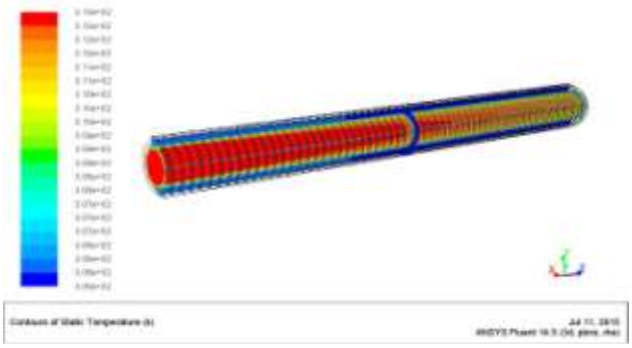


**Re = 8000**

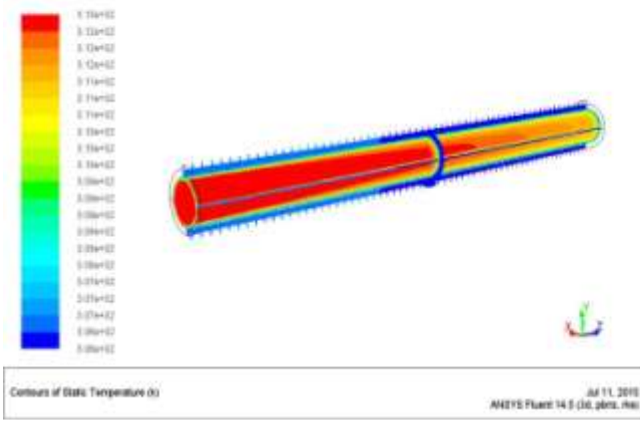
**Figure 4.1(b) Temperature contour for pitch(p) = 8 at constant inlet temperature($t_i$) = 312.65 K**



**Re=4000**



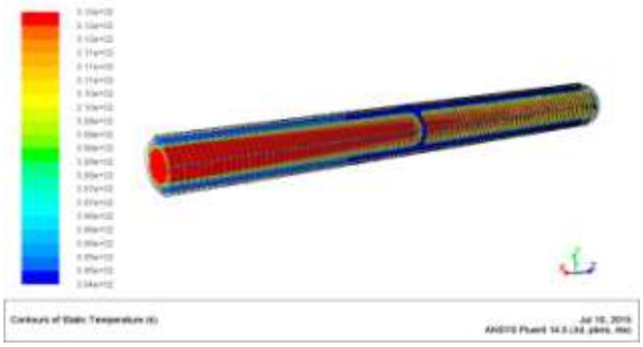
**Re=6000**



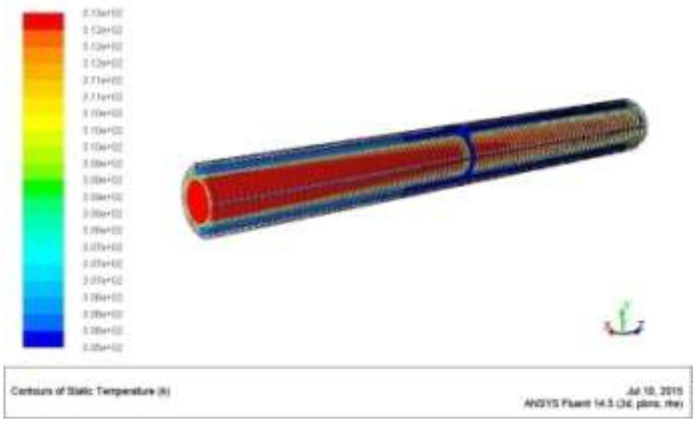
**Re=8000**

**Figure 4.1(c) Temperature contour for pitch (p) =6 at constant inlet temperature ($t_i$) = 312.65 K**

**Re=4000**



**Re=6000**



**Re = 8000**

**Figure 4.1(d) Temperature contour for pitch(p) = 4 at constant inlet temperature($t_i$) = 312.65 K**

In the above figure 4.1 temperature contour of helical fin tube for different Reynolds number at constant air inlet temperature of 312.65 K is shown. In figure 4.1(a) temperature contour of helical fin tube for pitch(p) = 10 mm & thickness(t)= 1mm shown, where we conclude that with increase in velocity of air flow or the Reynolds number, mean temperature at outlet increases. This is because; with increase in Reynolds number velocity of flow increases with increase in velocity of flow time available for heat transfer between two fluids decreases.

Fig 4.1 (b) shows the temperature contour of helical fin tube for p= 8 mm. By comparing the fig 4.1(a) & 4.1(b), we find that for same Reynolds number (for e.g. Re= 6000) outlet temperature for p= 8 mm is less than outlet temperature for p= 10 mm.This is due to the reason that with increase in pitch size the length of the tube of the exchanger increases, which increases the surface area of contact of heat exchanger. With increase in area of contact the heat transfer between two fluid increases so, the outlet temperature of air increases with increase in pitch size

It is also shown in Fig.4.1(c-d) when pitch size decreases from 6 to 4 the outlet temperature also decreases.

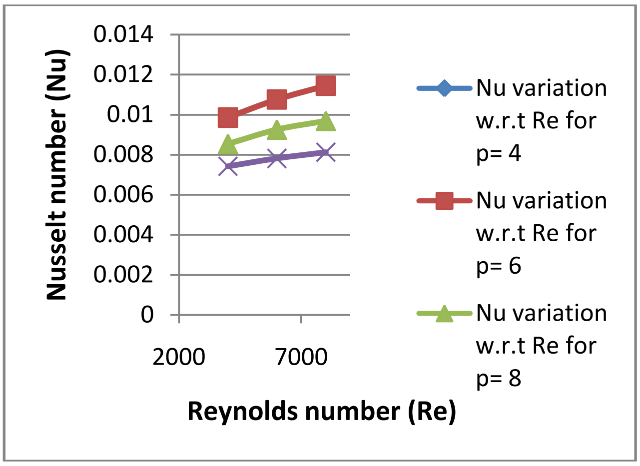



**Figure 4.2 Variation of Nusselt number(Nu) with respect to Reynolds number(Re) for different fin pitch**

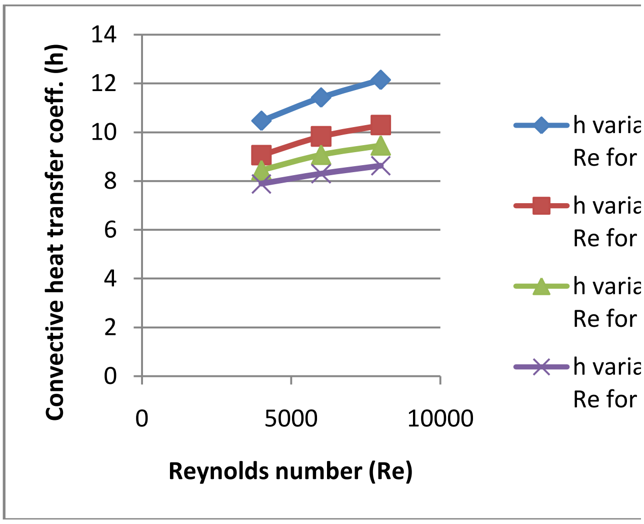



**Figure 4.3 Variation of Convective heat transfer coefficient (h) with respect to Reynolds number(Re) for different fin pitch**

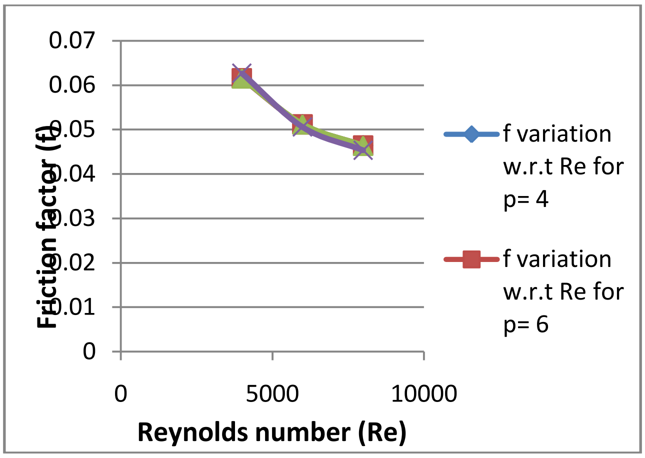



**Figure 5.6 Variation of friction factor (f) with respect to Reynolds number (Re) for different fin pitch**

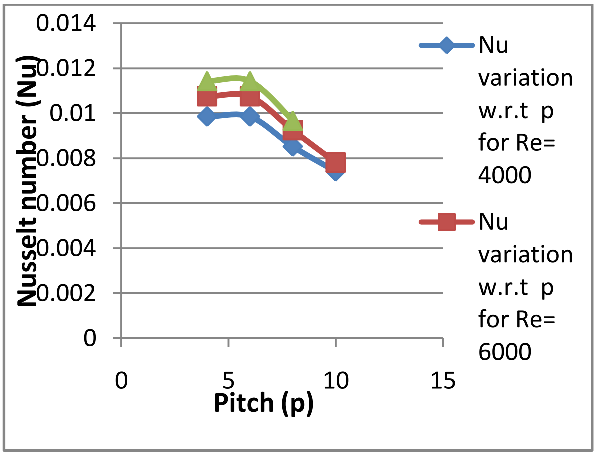



**Figure 5.8 Variation of Nusselt number (Nu) with respect to Pitch (p) for different Reynolds numbers**

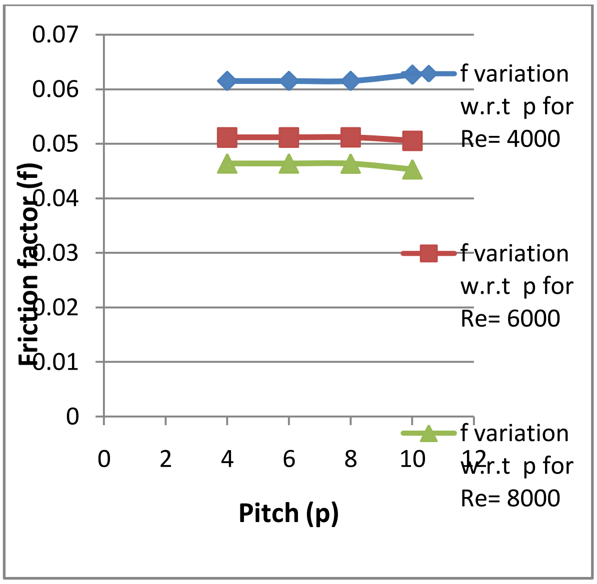



**Figure 5.9Variation of Friction factor (f) with respect to Pitch (p) for different Reynolds numbers**

- **Fin material=Copper**
- **Fin thickness = 1 mm**
- **Thermal conductivity = 387.6 W/m-K**

| Fin pitch (p) (mm) | Reynolds number Re | Air inlet velocity (m/s) | Mass flow rate of air (kg/sec) | Air inlet temp. K | Air outlet temp. K |
|---|---|---|---|---|---|
| 10 | 4000 | 2.3 | 0.001421125 | 312.65 | 309.5947 |
| 10 | 6000 | 3.45 | 0.002131688 | 312.65 | 310.3896 |
| 10 | 8000 | 4.6 | 0.00284225 | 312.65 | 310.834 |
| | | | | | |
| 8 | 4000 | 2.3 | 0.001421125 | 312.65 | 309.4435 |
| 8 | 6000 | 3.45 | 0.002131688 | 312.65 | 310.2424 |
| 8 | 8000 | 4.6 | 0.00284225 | 312.65 | 310.7061 |
| | | | | | |
| 6 | 4000 | 2.3 | 0.001421125 | 312.65 | 309.3177 |
| 6 | 6000 | 3.45 | 0.002131688 | 312.65 | 310.1237 |
| 6 | 8000 | 4.6 | 0.00284225 | 312.65 | 310.5966 |
| | | | | | |
| 4 | 4000 | 2.3 | 0.001421125 | 312.65 | 308.9565 |
| 4 | 6000 | 3.45 | 0.002131688 | 312.65 | 309.8434 |
| 4 | 8000 | 4.6 | 0.00284225 | 312.65 | 310.3318 |

| Fin pitch (p) (mm) | Reynolds number Re | Pressure drop (Pa) | Convective heat transfer coefficient h ($W/m^2K$) | Nusselt number Nu | Friction factor f |
|---|---|---|---|---|---|
| 10 | 4000 | 3.0664189 | 7.882998 | 0.00742336 | 0.062655594 |
| 10 | 6000 | 5.4835078 | 8.295981 | 0.007812263 | 0.050570364 |
| 10 | 8000 | 8.6780167 | 8.624774 | 0.008121885 | 0.045346443 |

| (p) (mm) | Re | P (Pa) | h ($W/m^2K$) | Nu | f |
|---|---|---|---|---|---|
| 8 | 4000 | 3.0677335 | 8.446286 | 0.008527407 | 0.061514425 |
| 8 | 6000 | 5.6050477 | 9.074656 | 0.003249903 | 0.0511842 |
| 8 | 8000 | 808367395 | 9.456071 | 0.009684017 | 0.046388518 |
| | | | | | |
| 6 | 4000 | 2.9958563 | 9.055405 | 0.009856158 | 0.061507526 |
| 6 | 6000 | 5.5357738 | 9.822637 | 0.010754363 | 0.05117545 |
| 6 | 8000 | 8.8568716 | 10.28363 | 0.01143677 | 0.046413165 |
| | | | | | |
| 4 | 4000 | 2.9955254 | 10.46643 | 0.009856158 | 0.061507526 |
| 4 | 6000 | 5.5348334 | 11.42025 | 0.010754363 | 0.05117545 |
| 4 | 8000 | 8.86133 | 12.14491 | 0.01143677 | 0.046413165 |

## REFRENCES:

[1] Dhananjay kumar, Prof. Alok choube, review of research on experimental study of heat transfer enhancement in fin tube heat exchanger by vortex generator.may2014 international journal of engineering sciences & research technology.

[2] Arafat a. bhuiyana, a. k. m. sadrulislama, m. ruhulaminb, numerical study of 3d thermal and hydraulic characteristics of wavy fin and tube heat exchanger, global digital central issn: 2151-8629

[3] Alahasan, kai sireen, raed abed mehndi, performance investigation of plain and finned tube evaporative cooled heat exchangers ,11 October 2002. Printed with permission from applied thermal engineering 23 (3), pages 325-340.

[4] Aossainnematic and mohammad moghimi, numerical study of flow over annular-finned tube heat exchangers by different turbulent models,www.cfdl.issres.net vol. 6 (3) – September 2014

[5] Vladimir glažar, Bernard franković, Paolo blemish. numerical and experimental analysis of fin and tube heat exchanger. Faculty of engineering, university of Rijeka, vukovarska 58, hr-51000 Rijeka.

[6] M. r. jafari nasr and a. t. zoghi, full analysis of low finned tube heat exchangers, 232 - vol. 16, no. 3, September 2003,www.sid.ir

[7] Ahmed f. khudheyer and mahmoud sh., numerical analysis of fin-tube plate heat exchanger by using cfd technique, vol. 6, no. 7, July 2011

issn 1819-6608 , arpn journal of engineering and applied sciences.

[8] Parinya pongsoi, Somchai wongwises, determination of fin, accepted November 19, 2014 vol. 1, special issue 1, pp. 251-262, February, 2015. For maximum performance index of l-footed spiral fin-and-tube heat exchangers,

[9] Ankit uppal, Dr. vinod kumar, dr. chanpreet singh, CFD analysis of heat transfer enhancement in a heat exchanger using various baffle arrangements,ijrmet vol. 4, issue 2, may - October er 2014

[10] Dilekkumlutas¸ ziyahaktankaradeniz, numerical model on frost height of round plate fin used for outdoor heat exchanger of mobile electric heat pumps,hindawi publishing corporation advances in mechanical engineering volume 2012, article id 863731, 7 pages doi:10.1155/2012/863731

[11] Seong won hwang1, dong hwan kim1, june kee min2 and ji hwan jeong1, generators has also been investigated. delta winglet vortex generators have recently attracted research interest, s. w. hwang et al. / journal of mechanical science and technology 26 (9) (2012) 2949~295, accepted April 16, 2012

[12] M. perčić1 – k. lenić1 – a. trp1, a three-dimensional numerical analysis of complete cross flow heat exchangers with conjugate heat transfer, engineering review, vol. 33, issue 1, 23-40, 2013.

[13] Sandeep k. patel, Professor alkesh m. mavani, shell & tube heat exchanger thermal design with optimization of mass flow rate and baffle spacing, Patel et al, international journal of advanced engineering research and studies ,ijaers/vol. ii/ issue i/oct.-dec.,2012/130-135.

[14] J. e. o'brien, m. s. sohal, t. d. foust, p. c. heat transfer enhancement for finned-tube heat exchangers with vortex generators: experimental and numerical results august 18, 2002. 12th international heat transfer conference.

[15] M. r. jafari nasr and a. t. zoghifull analysis of low finned tube heat exchangers,232 - vol. 16, no. 3, September 2003 ije transactions a: basics

[16] Rathodpravin p, ravi kumar, akhilesh gupta Enhancement of condensation heat transfer over horizontal integral-fin tubes- a review study. journal of engineering research and studies e-issn 0976-7916

[17] Avinash d. jadhav, tushar a. koli, vijay h. patil, critical review on different heat exchangers used for heat transfer between two fluids,avinash d. jadhav et al., international journal of engineering, business and enterprise applications, 8(2), march-may., 2014, pp. 149-152

[18] Prof. arvind s. sorathiya, manankumar b. josh, prof. .et.al [2014]:- heat transfer augmentation of air cooled 4 stroke si engine through fins- a review paper conditions, materials etc.ijreat international journal of research in engineering & advanced technology, volume 2, issue 1, Feb.-mar, 2014 issn: 2320 - 8791

[19] Jignesh m. chaudhari.heat transfer enhancement in fin and tube heat exchanger.vol.8, no3, march 2013 issn 1819-6608 arpn journal of engineering and applied sciences.

[20] Santosh d katkade, prof j h bhangale, prof d d palande, experimental investigation of spiral fin tube heat exchanger with different fin thickness, international journal of research in advent technology, vol.3, no.1, January 2015 e-issn: 2321-9637.