

# Optimization of Heat Transfer Rate of Helical Fin-Tube Heat Exchanger with Different Fin Pitch using CFD Analysis

Ashok sahu<sup>#1</sup>, Anita Ahirwar<sup>\*2</sup>, Ravi Vishwakarma<sup>#3</sup>  
 Mechanical Engineering  
 yogeshlnct65@gmail.com

**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 flow 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

1) *Sub Section 2 Heading:* A level-2 heading must be in Italic, left-justified and numbered using an uppercase alphabetic letter followed by a period. For example, see heading "C. Section Headings" above.

2) *Sub Section 3 Heading:* A level-3 heading must be indented, in Italic and numbered with an Arabic numeral followed by a right parenthesis. The level-3 heading must end with a colon. The body of the level-3 section immediately



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.

### ***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 techniques 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

*References*

The heading of the References section must not be numbered. All reference items must be in 8 pt font. Please

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

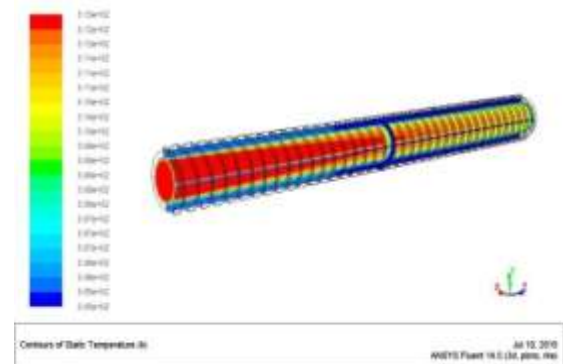
The result from this dissertation work is that the simulation technique (CFD) gives more accurate and consistent result as compare to the experimental technique. In this dissertation work it is analyzed that the experimental

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.

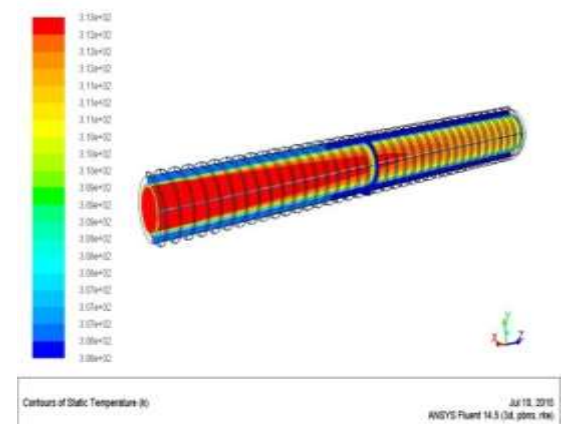
During CFD analysis with different velocities of air passing over the fin and tube and obtained temperature contours at different values of fin pitches ( $p$ ) and different Reynolds number ( $Re$ ). I would like to show some results from case-I, case-II , case -III and case -IV

#### TEMPERATURE COUNTERS RESULTS

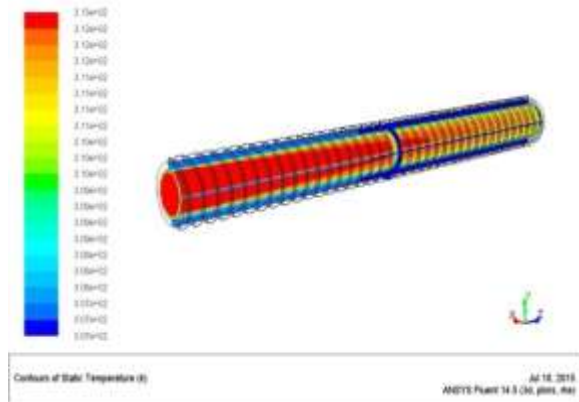
For constant air inlet temperature condition the temperature contours are shown in below figure 5.1.



**Re=4000**

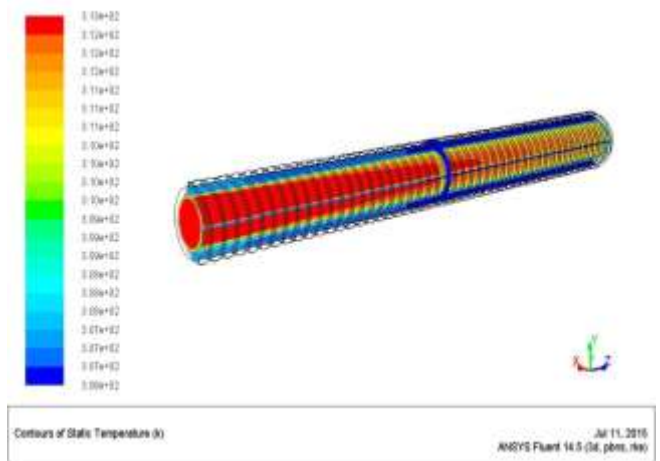


**Re=6000**



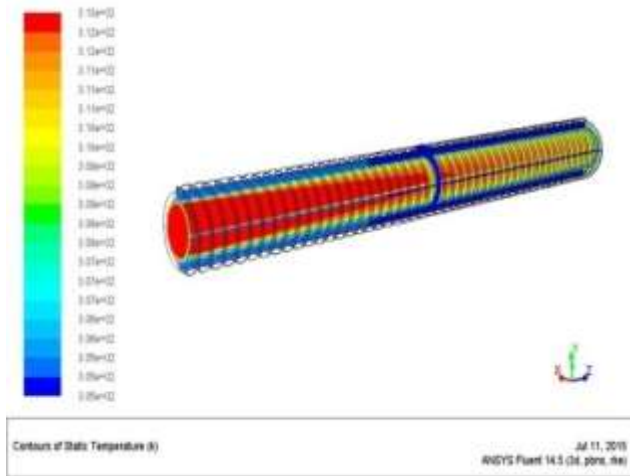
Re=8000

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

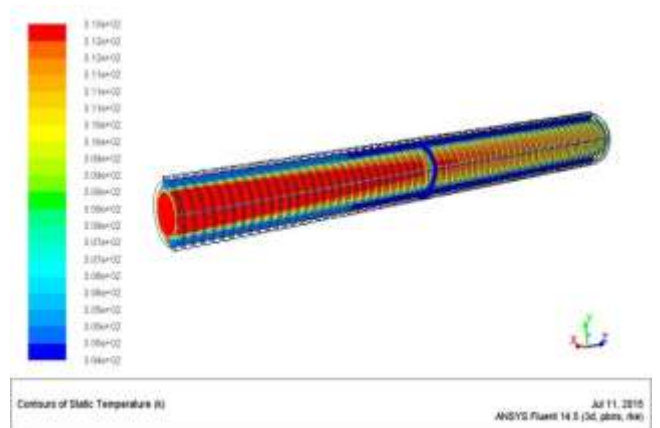


Re = 8000

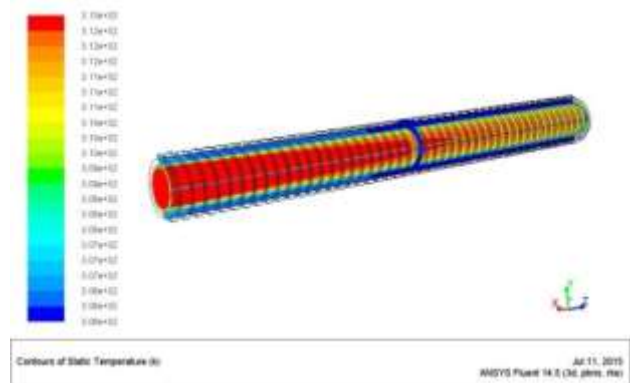
Figure 4.1(b) Temperature contour for pitch(p) = 8 at constant inlet temperature( $t_i$ ) = 312.65 K



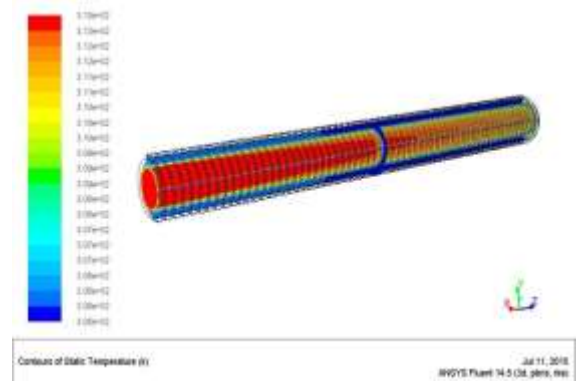
Re=4000



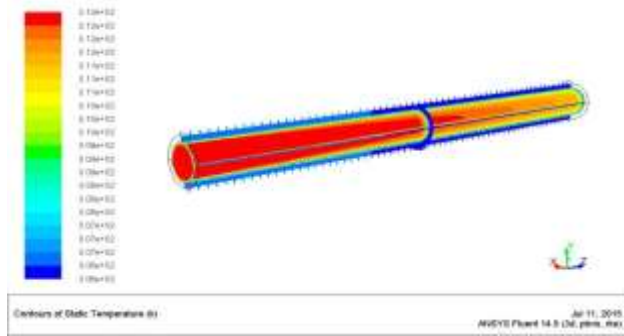
Re=4000



Re = 6000

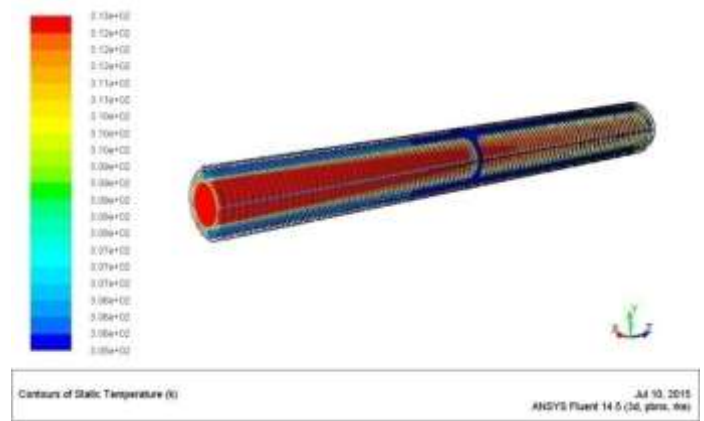


Re=6000



Re=8000

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

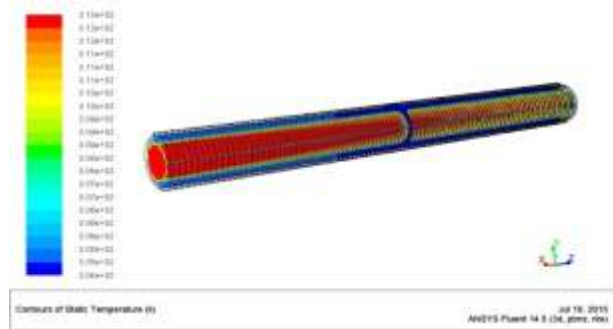


Re = 8000

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



Re=4000



Re=6000

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)= 1 mm 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.

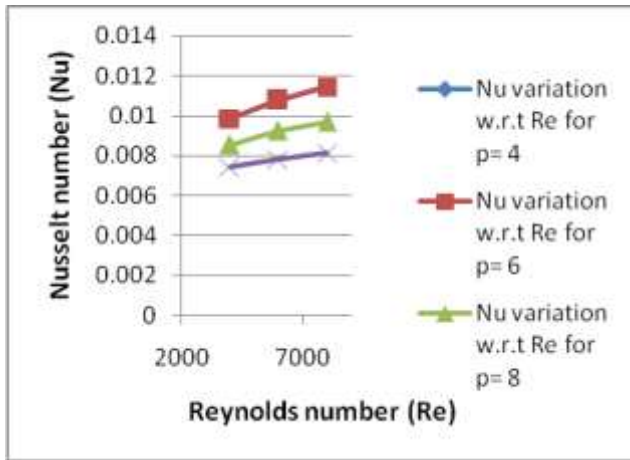


Figure 4.2 Variation of Nusselt number(Nu) with respect to Reynolds number(Re) for different fin pitch

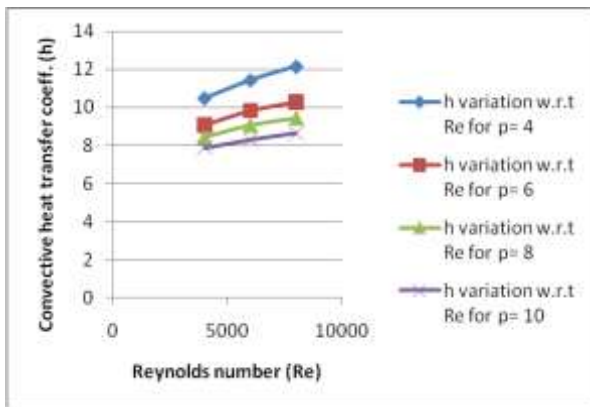


Figure 4.3 Variation of Convective heat transfer coefficient (h) with respect to Reynolds number(Re) for different fin pitch

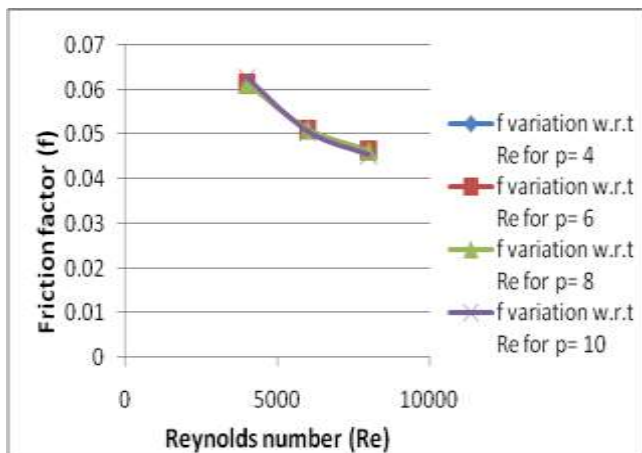


Figure 5.6 Variation of friction factor (f) with respect to Reynolds number (Re) for different

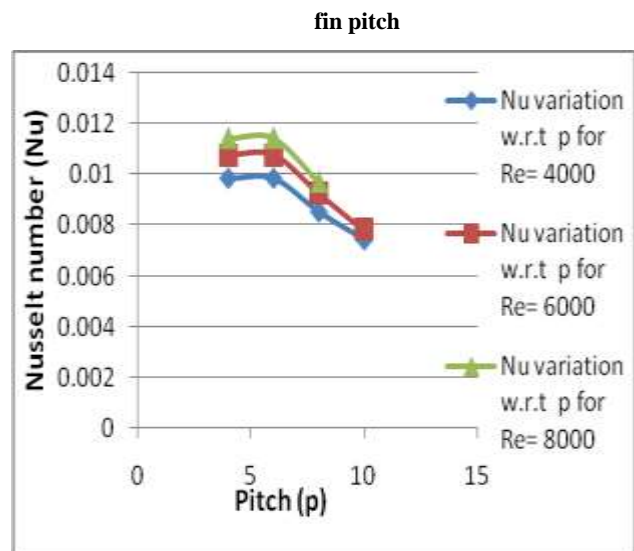


Figure 5.8 Variation of Nusselt number (Nu) with respect to Pitch (p) for different Reynolds numbers

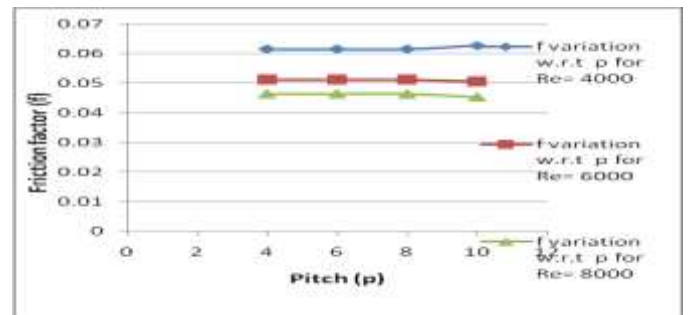


Figure 5.9 Variation of Friction factor (f) with respect to Pitch (p) for different Reynolds numbers

- 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

### V. CONCLUSION

Heat transfer behaviors for different boundary conditions are predicted and optimize condition for maximum Nusselt number (Nu) and minimum friction factor (f) was plotted against Reynolds number. Following are the outcome of above numerical study; As the fin pitch size increases the variation of Nusselt number for a particular value of Reynolds number is less or insignificant. The Nusselt number is maximum for all values of Re for fin pitch (p= 4) and it decrease with increase in pitch size. The variation of Nu number with respect to Re number between p= 8 and p= 10 is less and with further increases in pitch size, The variation almost vanishes. The variation of convective heat transfer coefficient(h) with respect to Reynolds number(Re) between p= 8 mm and p= 10 mm is less as comparison to p=4 mm and p= 6mm. As the value of fin pitch increases the variation of Convective heat transfer coefficient for a particular value of Reynolds number is less At Reynolds number (Re=4000 to Re= 6000) the variation of friction factor is maximum and then the variation of friction factor slightly decreases for Reynolds number ( Re=6000 to Re= 8000) . The best values of Nusselt number Nu, Convective heat transfer coefficient h, Friction factor f can be found at pitch = 4 mm and there is not so much variation occur at the values of the same at p = 4 & 6, Thus the design is satisfactory at p= 4mm and p= 6 mm.

### FUTURE WORK

The optimization condition presented in this study is by intersection of two graphs. More accurate method should be developed to determine the maximum condition. Along with numerical analysis experimental validation should be required. In this study analysis of Copper fin at fin pitch values 4, 6, 8, 10 mm is carried out and in future in place of fin size, changing in values of tube diameter of fin may be considered.

### VI. REFERENCES

- 1) Gaurav tiwari, Dr. Nitin shrivastava, Gaurav tiwari Int. Journal of Engineering Research and Applications ISSN : 2248-9622, Vol. 4, Issue 10( Part - 5), October 2014, pp.108-114
- 2) Rudra Nandan Pramanik, Smruti Ranjan Pradhan and Prafulla Kumar Sahoo, International Journal Of Applied Research In Mechanical Engineering (IJARME) ISSN: 2231 –5950, Vol-2, Iss-2, 2013
- 3) V. S. Kumbhar, D. G. Mali, P. H. Pandhare& R. M. Mane , Department of Automobile Engineering, Gourishiv Polytechnic Khatav, Satara, Maharashtra. International Journal of Instrumentation, Control and Automation (IJICA) ISSN: 2231-1890, Vol-1 Iss-3,4, 2012.
- 4) Shane Curtis , Mark Owen, Terrence Hess and Scott Egan, Brigham Young University Provo, Utah. December 5, 2008.
- 5) AchintaSarkar, Achin Kumar Chowduri, Arup JyotiBhowal and Bijan Kumar Mandal, International Journal of Scientific & Engineering Research, Volume 3, Issue 6, June-2012 1 ISSN 2229-5518.
- 6) C. AnandaSrinivasan and C.G. Saravanan, Department of Mechanical Engineering, Annamalai University, Annamalai Nagar , Tamil Nadu, India. Journal of Sustainable Energy & Environment 1 (2010) 85-91.
- 7) AlvydasPikunas, SaugirdasPukalskas, JuozasGrabys, Vilnius Gediminas Technical University, J.Basanaviciaus Str.28, LT-2006 Vilnius, Lithuania. Journal of KONES Internal Combustion Engines 2003, vol. 10, 3-4.
- 8) Maher A.R. Sadiq, Yassar K. Ali and AimenRashad Noor, College of Engineering, University of Al-Kufa / Najaf Ashraf 2006.,
- 9) Viral K Pandya, Shailesh N Chaudhary, Bakul T Patel, Parth D Patel, LaljibhaiChaturbhai Institute of Technology, Bhandu, Gujarat, India, International Journal of Advances in Engineering & Technology, Nov 2011.IJAET ISSN: 2231-1963.
- 10) Ali Barzegar, Mohsen Mazloum ,Farsibaf Mona, Golchinpour ,Department of industrial engineering , Iran university of science & technology(IUST), Proceedings of the 41st International Conference on Computers & Industrial Engineering.