INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS)

www.ijprems.com
editor@ijprems.com

Vol. 03, Issue 07, July 2023, pp : 507-510

e-ISSN : 2583-1062

Impact Factor : 5.725

THERMAL CHARACTERISTICS PERFORMANCE OF HEAT TRANSFER UNDER FORCED CONVECTION IN A RECTANGULAR BODY WITH CIRCULAR FINS

Vivekanand Singh[1], Surjeet Singh Rajpoot[2]

[1]M. Tech Scholar, SCOPE College of Engineering, Bhopal, (India)

[2]Associate Professor, SCOPE College of Engineering, Bhopal, (India)

ABSTRACT

In this research perforated finned heat exchangers' diameter affected convective heat transmission. These holes were positioned on the circular fins so that they lined up with one another at the same predetermined angle. On the bottom of the fin, close to the heating tube surface, the perforations produced turbulence. The impact of this turbulence on pressure drop and heat transmission was then examined through a number of investigations. To ascertain the optimal heat transmission, several experiments were conducted at various diameters. In addition, a perforated finned heater was compared with an imperforate finned heater to observe the differences. In the cases of the Re above the critical value, Nusselt numbers for the perforated finned positions are higher than the Nusselt numbers for the imperforate state. Moreover, a correlation has been obtained between the Re and Nu in the Re number above the critical value and the Re below the critical value. Meanwhile, correlations regarding pressure drops in the flow areas have been obtained.

Key words: Effectiveness, Heat Transfer, Nusselt Number, Reynold Number, Temparature.

1. INTRODUCTION

Thermal energy is related to the temperature of matter. A given material and mass, the higher the temperature, the greater its thermal energy. Heat transfer is a study of the exchange of thermal energy through a body or between bodies which occurs when there is a temperature difference. When two bodies are at difference temperature, thermal energy transfers from the one with higher temperature to the one with lower temperature. Heat always transfers from hot to cold.

3

July

Fig. 1 Pin fin

The common SI and English units and conversation factors used for heat and heat transfer rates. Heat is typically given the symbols Q, and is expressed in joules (J) in SI units. The rate of heat transfer is measured in watts (W), equal to joules per second, and is denoted by q. The heat flux, or the rate of heat transfer per unit area, is measured in watts per area (W/m²), and uses q” for the symbol.

Conduction is at transfer through solids or stationary fluids .when you touch a hot object, the heat you feel is transferred through your skin by conduction. Two mechanisms explain how heat is transferred by conduction: lattice vibration and particle collision. Conduction through solids occurs by a combination of the two mechanisms; heat is conducted through stationery fluids primarily by molecular collisions.

Convection uses the motion of fluids to transfer heat. In a typical convective heat transfer, a hot surface heats the surrounding fluids, which is then carried away by fluid movements such as wind. the warm fluid is replaced by cooler fluid, which can draw more heat away from the surface. Since the heated fluid is constantly replaced by cooler fluid, the rate of heat transfer is enhanced.

Natural convection (or free convection) refers to a case where the fluid movement is created by the warm fluid itself .the destiny of fluid decrease as it is heated; thus hot fluids are lighter than cool fluids. warm fluids surrounding a hot object rises, and is replaced by cooler fluid. The result is a circulation of air above the warm surface.

The heat conducted through solids, walls or boundaries has to be continuously dissipated to the surroundings or environment to maintain the system in steady state conduction. In many engineering applications large quantities of heat have to be dissipated from small areas. Heat transfer by convection between a surface and the fluid surroundings it can be increased by attaching to the surface thin strips of metals called fins.

The fins increase the effective area of the surface thereby increasing the heat transfer by convection. The fins are also referred as "extended surfaces". Fins are manufactured in different geometries, depending up on the practical applications. Most of the engineering problems require high performance heat transfer components with progressively less weights, volumes, accommodating shapes and costs. Extended surfaces (fins) are one of the heat exchanging devices that are employed extensively to increase heat transfer rates. The rate of heat transfer depends on the surface area of the fin.

2. COMPUTATIONAL FLUID DYNAMICS

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 heat transfer, as relevant to this project. It begins with a review of the tools 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 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, since experiments have a cost directly proportional to the number of configurations desired for testing, unlike with CFD, where large amounts of results can be produced at practically no added expense. In this way, parametric studies to optimize equipment are very inexpensive with CFD when compared to experiments. This section briefly describes the general concepts and theory related to using CFD to analyze fluid flow and.

3. CONSERVATION EQUATIONS

The conservation laws of physics form the basis for fluid flow governing equations (previously listed as Equations 1-3 in Section 2.1: Governing Equations and Numerical Schemes). The laws are:

• Law of Conservation of Mass: Fluid mass is always conserved. (Equation 1)

$$\frac{\partial(\rho \mathcal{U}i)}{\partial xi} = 0$$

• Newton's 2nd Law: The sum of the forces on a fluid particle is equal to the rate of change of momentum. (Equation 2)

$$\frac{\partial}{\partial xi}(\mathcal{P}\mathcal{U}i\mathcal{U}j) = \frac{\partial}{\partial xi}(\mu\frac{\partial uj}{\partial xi}) - \frac{\partial \rho}{\partial xj}$$

• First Law of Thermodynamics: The rate of head added to a system plus the rate of work done on a fluid particle equals the total rate of change in energy. (Equation 3)

$$\frac{\partial}{\partial xi}(\rho\mathcal{U}i\mathcal{T}) = \frac{\partial}{\partial xi}(\frac{\mathcal{K}}{\mathcal{C}p} - \frac{\partial uj}{\partial xi})$$

The fluid behavior can be characterized in terms of the fluid properties velocity vector **u** (with components *u*, *v*, and *w* in the *x*, *y*, and *z* directions), pressure *p*, density ρ, viscosity μ, heat conductivity *k*, and temperature *T*. The changes in these fluid properties can occur over space and time. Using CFD, these changes are calculated for small elements of the fluid, following the conservation laws of physics listed above.

The changes are due to fluid flowing across the boundaries of the fluid element and can also be due to sources within the element producing changes in fluid properties. This is called the Euler method (tracking changes in a stationary mass while particles travel through it) in contrast with the Lagrangian method (which follows the movement of a single particle as it flows through a series of elements).

Boundary condition

A 50W heater is used to heat the base wall. Fin base is heated with constant heat flux. Heat transferred from the base to the fin by conduction and from fin to atmosphere by convection. Some amount of heat is also removed by convection from fin base where fin is not attached. Air enter into the rectangular duct with 300 K temperature and air exit from duct at atmospheric pressure.

INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS)

www.ijprems.com
editor@ijprems.com

Vol. 03, Issue 07, July 2023, pp : 507-510

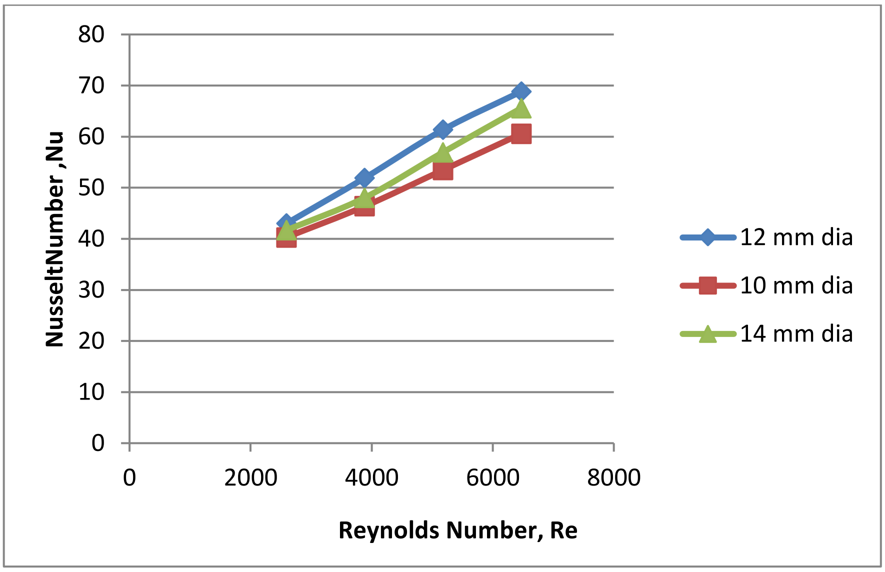
e-ISSN : 2583-1062

Impact Factor : 5.725

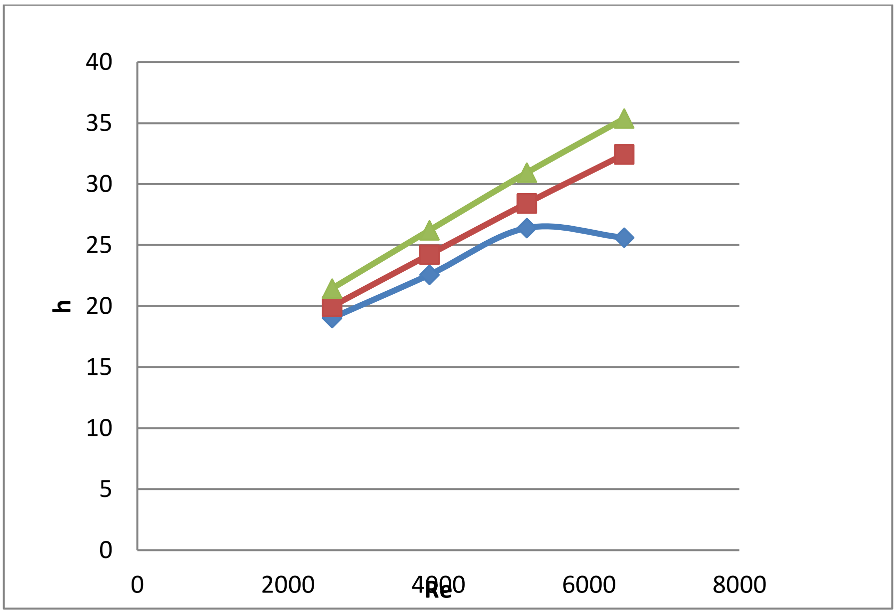
4. RESULT & DISCUSSION

The amount of heat transferred in any process can be defined as the total amount of transferred energy excluding any macroscopic work that was done and any energy contained in matter transferred. For the precise definition of heat, it is necessary that it occur by a path that does not include transfer of matter. As an amount of energy (being transferred), the SI unit of heat is the joule (J). The conventional symbol used to represent the amount of heat transferred in a thermodynamic process is Q. Heat is measured by its effect on the states of interacting bodies. Heat transfer coefficients are calculated both for fin and fin base. Q is the total heat transfer from the fin or fin base. And As is the surface area. **Δ**T is the temperature difference between inlet air and the average temperature of fin or fin base.

The velocity profile of air inside the duct at midsection. From the velocity profile it is clear that the velocity between the two fins is always greater than the free steam velocity. Velocity after the fin is very small. This may be because of recirculation zone. From Velocity profile we can conclude that velocity in front and rear of the fin are very small and maximum air velocity occurs at fin side wall. Heat removal rate is high at sidewall of the fin. It is possible to enhance heat transfer by increasing surface area at sidewall though introducing elliptical or aerofoil shape which is supported by several research article.



The velocity profile of air inside the duct at midsection. From the velocity profile it is clear that the velocity between the two fins is always greater than the free steam velocity. Velocity after the fin is very small. This may be because of recirculation zone. This is supported. From Velocity profile we can conclude that velocity in front and rear of the fin are very small and maximum air velocity occurs at fin side wall. Heat removal rate is high at sidewall of the fin. It is possible to enhance heat transfer by increasing surface area at sidewall though introducing elliptical or aerofoil shape which is supported by several research article.



5. CONCLUSION

This paper analyses computational results of flow characteristics along with and heat transfer characteristics of rectangular body with staggered arranged circular fins subjected to forced convection. All the results calculated separately for the base and fins make this paper distinct.

- Heat transfer co-efficient and Nusselt number increase with the increase of velocity.
- Pressure drop across the fin was significant with the increase of inlet velocity. As the velocity increases, the heat transfer rate from the base increases and from the fin decreases. We observed
- We observe that Nusselt number increasing with increasing Reynolds number.
- It is observed that heat transfer from the fin is higher than the base.
- The inlet Air velocity increases, the heat transfer rate from the fin decreases and heat transfer rate from the base increases.

6. REFRANCES

[1] D.Q. Kern, A.D. Kraus, Extended Surface Heat Transfer, McGraw-Hill, New York, 1972.

[2] A.D. Krause, A. Bar-Cohen, Design and Analysis of Heat Sinks, Wiley, New York, 1995.

[3] P. Razelos. A critical review of extended surface heat transfer, Heat Transfer Eng. 24 (6) (2003) 11–28.

[4] F.P. Incropera, D.P. Dewitt, Introduction to Heat Transfer, Wiley, New York, 1985.

[5] A.F. Mills, Heat Transfer, second ed., Prentice-Hall, New Jersey,1999.

[6] Wang, F., Zhang, J., & Wang, S. (2012). Investigation on flow and heat transfer characteristics in rectangular channel with drop- shaped pin fins. Propulsion and Power Research, 1(1), 64–70.doi:10.1016/j.jppr.2012.10.003

[7] Zhou, F., & Catton, I. (2011). Numerical evaluation of flow and heat transfer in plate-pin fin heat sinks with various pin cross- sections. Numerical Heat Transfer, Part A: Applications, 60(2), 107–128. doi:10.1080/10407782.2011.588574

[8] Peles, Y., Koşar, A., Mishra , C., Kuo , C.-J., & Schneider, B. (2005). Forced convective heat transfer across a pin fin micro heat sink. International Journal of Heat and Mass Transfer, 48(17), 3615–3627. doi:10.1016/j.ijheatmasstransfer.2005.03.017

[9] Al-Sallami, W., Al-Damook , A., & Thompson, H. M. (2016). A numerical investigation of thermal air flows over strip fin heat sinks. International Communications in Heat and Mass Transfer, 75, 183–191. doi:10.1016/j.icheatmasstransfer.2016.03.014

[10] G.J. Van Fossen , Heat transfer coefficients for staggered arrays of short pin fins, Journal of Engineering for Power104 (2) (1982) 268–274.

[11] D.E. Metzger, R.A. Berry, J. P. Bronson, Developing heat transfer in rectangular ducts with staggered arrays of short pin fins, Journal of Heat Transfer 104 (1982) 700–706

[12] D.E. Metzger, C.S. Fan, S.W. Haley, Effects of pin shape and array orientation on heat transfer and pressure loss in pin fin arrays,Journal of Engineering for Gas Turbines and Power 106 (1984) 252–25

[13] A.E. Bergles, R. M. Manglik, Current progress and new developments in enhanced heat and mass transfer, J. Enhanc. Heat Transfer. 20 (1) (2013) 1–15.