Convective Heat Exchange with in a Compact Heat Exchanger

V. Balaji[1] D. Manivannan[2]
[1]PG Scholar 2Assistant Professor
[1,2]Department of Mechanical Engineering
[1,2]M.R.K. Institute of Technology, Kattumannarkovil

Abstract **- Compact heat exchangers have been receiving attention because of its high heat exchange area per unit volume and good heat transfer performance. In this project attempt was made to determine the average value of heat transfer coefficient between the air flow and the alumina walls within a single channel of a compact heat exchanger. Values obtained for average Nussel number are in a good agreement despite the assumption that thermodynamic properties does not change much with temperature.**

INTRODUCTION

A great number of numerical studies applying finite volume method have been conducted to investigate heat transfer and fluid flow in heat exchangers. To provide computational evidence for rational use of extended fin surfaces as a means to enhance heat transfer, simulated heat transfer problems in conjugate finned tube heat exchanger. investigated the effect of geometric characteristics (distance between fins, fin height, tube ellipticity, and tube thickness) on heat transfer and pressure drop characteristic in a gas-water plane fin type heat exchanger with one row tube configuration. simulate 3D flow through the single narrow passage between fins and obtained the distribution of the heat transfer coefficient on the fin surface and average heat transfer coefficient.

Aim of several numerical studies was to investigate mixing processes induced by the fins in channels and to verify influence of hydrodynamic regimes on the performance of heat exchangers. Numerical analysis of mixing processes, divided in a several steps due to complex fin geometry of oil-gas compact cross section exchangers, demonstrated effect of flow rate and fin geometry on heat transfer coefficient, pressure drop and fouling tendencies. To observe obstacles induced vortical flow, conducted dimensional numerical study of air through two-row cylinder tubes.

More detailed numerical analyses of turbulent flow in correlation with heat transfer are presented in.

There are several numerical simulations that analyzed transient heat transfer, but few of them focused on the controllability of heat exchangers. finite difference method to solve the transient heat transfer in the single tube and regulate the water flow rate by using PI regulator. The problem of two- dimensional laminar flow around array of heat generated cylinders in cross flow was also investigated by using the finite difference method.

While finite volume method has been widely used to simulate heat transfer and fluid flow, finite element method are less frequently applied to observe processes that take place in heat exchangers. determined wall sheer stresses and heat flux at a wall of a rectangular duct . Subjecting equations to Dirichlet boundary condition and assuming fully developed laminar flow as well as uniform wall heat flux significantly simplified the problem. proposed locally conservative Galerkin finite-element approach for one dimensional transient heat conduction and two dimensional convection-diffusion problems . Similar problems as in but with numerical technique based on finite element method were investigated.

LITERATURE REVIEW

Analysis of micro channel heat sinks for turbulent as well as laminar flow. They demonstrated improvement of previous studies by relaxing constraints on fin thickness/ pitch ratio and allowing turbulent flow. Copeland (1995) modified previous analyses for developing flow and calculated optimum fin thickness and pitch for silicon heat sinks cooled by fluorocarbon liquids. Lee (1995) analysed flow through parallel fin heat sinks in fully ducted and partially ducted flows.

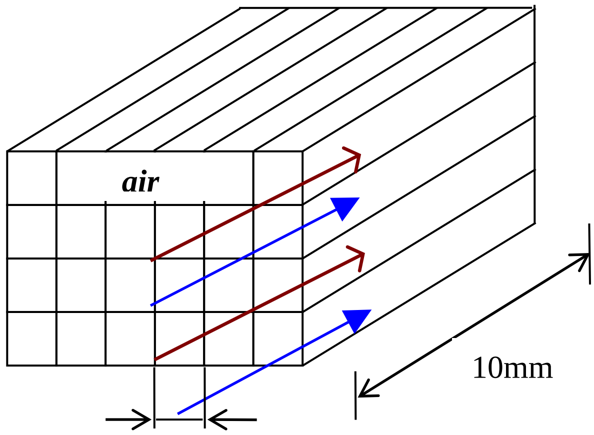
Unlike a fully ducted configuration, in partially ducted configuration at a fixed approach velocity, an optimum size of fin existed; thermal performance improves monotonically as fin pitch is decreased. . (1997) showed iso curves of pressure drop and fan power at fixed thermal resistance in addition to iso curves of thermal resistance at fixed pressure drop and fan power.

As pressure drop or fan/blower power increased, optimum fan thickness and pitch decreased, resulting in reduced thermal resistance. In addition to analysis, experimental and numerical studies were performed. (1997) performed experimental studies of compact heat sinks with fin thickness and pitch as small as 0.34mm and 0.70mm. Results correlated well with results from compact heat exchanger data. This compactness factor, defined as thermal conductance per unit volume, was three to seven times that of standard heat sinks.

M Bisht and K S Mehra (2014) number of experiments and published a paper on optimisation of working of processor by fins using FEM in International Journal for Research in Applied Science and Engineering Technology. The computational fluid dynamics is concentrated on the forced air cooling of the CPU using a heat sink. This paper utilizes CFD to identify a cooling solution for a desktop computer, which uses an 80 W CPU. In this study a complete computer chassis with different heat sinks are investigated and the performances of the heat sinks are compared.

DESIGN OF HEAT SINK

A Model Of A Parallel -Flow Heat Exchanger Is Presented



Schematic of in-Line Pin-Fin Heat Sinks

The geometry of an in-line pin-fin heat sink is shown in Fig 4.The dimensions of the base plate are L × W × tb, where L is the length in the stream wise direction, W is the width and tb is the thickness. Each pin fin has diameter D and height H. The longitudinal and transverse pitches are SL and ST respectively. The approach velocity of the air is U app. The direction of the flow is parallel to the x-axis.

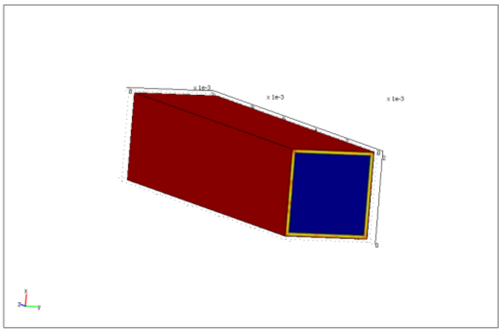
The base plate is kept at constant heat flux and the top surface (y = H) of the pins is adiabatic. The average local wall temperature of the pin surface is Tw(x). The heat source is idealized as a constant heat flux boundary condition at the bottom surface of the base plate. The mean temperature of the heat source is Ts.

OBJECTIVE

For co-current and counter-current operation of the equipment and flow rates (hot and cold air) specified by the instructor, determine: The heat lost to the surroundings. The overall efficiency. The temperature efficiency for the hot and cold air. The overall heat transfer coefficient U determined experimentally. The overall heat transfer coefficient U determined theoretically. Compare with the experimental one.

Solution using FEMLAB

FEMLAB heat transfer application mode is applied in the analysis of heat convection and conductance within a single channel. Dimension of a square cross section channel are 2mm x 2mm x 100mm with wall thickness of 0.1 mm.In order to solve equations and obtain the stationary analysis of the model Stationary nonlinear solver and Lagrange – Quadratic element type are used



3D temperature distribution in a single channel

For the air stream and alumina the following thermodynamic characteristics are adopted.

Air:

k=0.0505 (w/m K) - thermal conductivity;

c= 1529 (J/kg K) – specific heat capacity;

ρ= 0.8824 (kg/m^3) – density;

W max = 1.4 (m/s) – velocity.

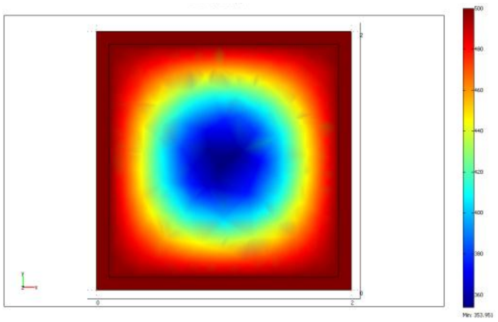
Alumina:

k=155 (w/m K) - thermal conductivity

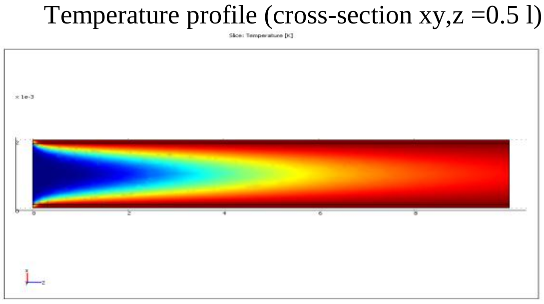
c= 895 (J/kg K) – specific heat capacity;

ρ= 2730 (kg/m^3) – density;

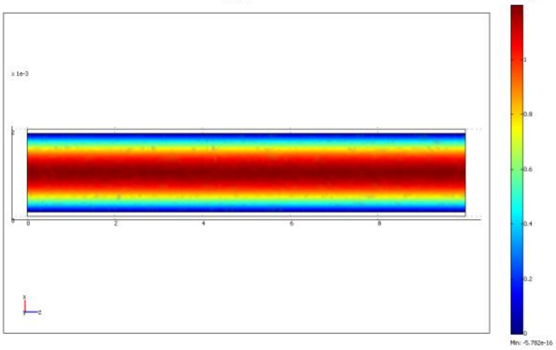
Maximum velocity for which the model gave the stable solution was of 2.2m/s. For faster air stream FEMLAB solution could not converge. Thermodynamic properties of the air and alumina are assumed to be constant and are adopted for mean temperature of the air and alumina. While properties of the air and alumina do not differ much from real values, the air velocity of 2.2 m/s is lower by 50-100% that gas velocity common for this type of heat exchangers.



Temperature profile (cross-section xy,z =0.5 l)

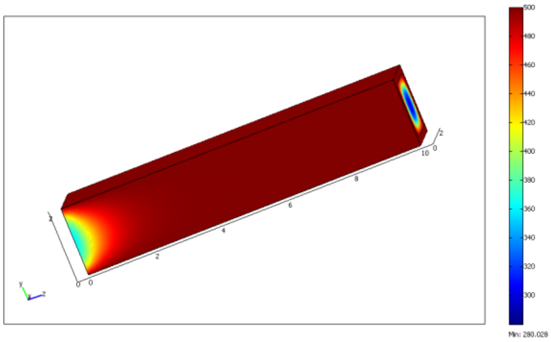


Temperature profile (cross-section x z, x = d/2)



Velocity profile (cross-section yz, y = d/2)

3D temperature distribution in a single channel



FEMLAB results:
$\int T_o dA$ [Km^2]
$\int W dA$ [$m/s\ m^2$]
Mass flow rate:

$$\dot{m} = \rho \cdot A \cdot W$$

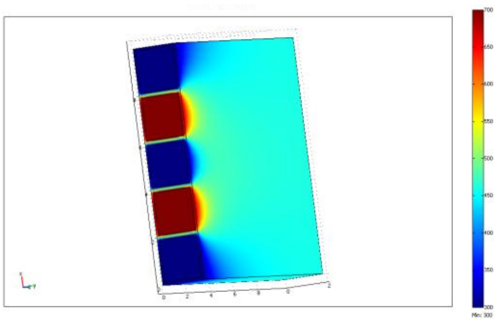
Amount of heat transferred to the fluid:

$$Q = \dot{m} \cdot c_p \cdot (To - Tin)$$

Average heat transfer coefficient:

$$\frac{1}{\alpha} = \frac{Q}{A_o \cdot (Tw - Tm)} - \frac{\delta}{\lambda}$$

Thermodynamic properties of the air are adopted for the T= 400 [K]. This assumption will probably not yield a great error in a case a) where mean temperature changes in the relatively narrow range 385-392 [K]. However, in the case b) where temperature of the air changes significantly 345-480[K] inaccuracy is larger.



Temperature field in a heat exchanger section
(T hot=700 [K], T cold=300K, v = 1.2 m/s)

CONCLUSION

According to the presented results we can conclude Nusselt number calculated using FEMLAB results and the simplified procedure has value 3.18w/m^2K. This value is in a satisfactory agreement with those obtained from literature. In order to determine more accurately Nu, it is necessary to calculate local values of dimensionless heat transfer coefficient which requires more complex analysis of temperature field within the channel.

Heat transfer coefficient increases both with air flow velocity and the wall temperature. However, the heat transfer coefficient change with the wall temperature change should be taken with caution because of the assumption that thermodynamic properties are constant. Main disadvantage of the model is the failure to implement Navier –Stokes equation into model.

To improve the existing model, it is necessary to implement the temperature dependence of thermodynamic air properties both in model and parametric study.

$$Cp = a_1 + a_2T + a_3T^2 + a_4T^3 + a_5T^4$$

REFERENCES

[1] S.F. Tsai, T.W.H. Sheu, S.M. Lee, Heat transfer in a conjugate heat exchanger with a wavy surface, International Journal of Heat and Mass Transfer 42 (1999) 1735-1745.
[2] Aytunc¸ Erek, Barıs Ozerdem, Levent Bilir , Zafer Ilken , Effect of geometrical parameters on heat transfer and pressure drop characteristics of plate fin and tube heat exchangers, Applied Thermal Engineering 25 (2005) 2421–2431
[3] D. Taler, Determination of heat transfer correlations for plate-fin-and-tube heat exchangers, Heat and Mass Transfer 40 (2004) 809–822.
[4] E. Carluccio, G. Starace , A. Ficarella, D. Laforgia, Numerical analysis of a cross-flow compact heat exchanger for vehicle applications, Applied Thermal Engineering 25, (2005) 1995–2013.
[5] Tony W. H. Sheui, S. F. Tsai, and T. P. Chiang, Numerical Study of Heat Transfer in Two-Row Heat Exchangers Having Extended Fin Surface, Numerical Heat Transfer, Part A, 35:797 - 814, 1999
[6] M. Yaghoubi, M. Rahnema, Numerical study of turbulent flow and heat transfer from an array of thick plates, International Journal of Thermal. Science, (2000) 39, 213–224.