Proceedings of HT2003
2003 ASME Summer Heat Transfer Conference
July 21–23, 2003, Las Vegas, Nevada, USA

HT2003-47541

ANALYSIS OF FLOW AND HEAT TRANSFER AT A FINNED TUBE IN CROSSFLOW

Bharat K. Rangan
Indian Institute of Technology Madras
Mechanical Engineering Department
Chennai, Tamil Nadu – 600036
India
Ph: 91 (44) 2257 8530 Fax: 91 (44) 2257 0509 e-mail: bharat_iit@hotmail.com

Adarsh Krishnamurthy
Indian Institute of Technology Madras
Mechanical Engineering Department
Chennai, Tamil Nadu – 600036
India
Ph: 91 (44) 2257 8530 Fax: 91 (44) 2257 0509 e-mail: raguram@vsnl.com

Dr. Vijay R. Raghavan
Indian Institute of Technology Madras
Mechanical Engineering Department
Chennai, Tamil Nadu – 600036
India
Ph: 91 (44) 2257 8530 Fax: 91 (44) 2257 0509 e-mail: vrr@iitm.ac.in

ABSTRACT

In the present study, a numerical investigation has been carried out into the fundamental problem of airflow past and heat transfer from a circular finned cylindrical tube, placed in a duct. The simulation is carried out using a finite volume method, based on laminar calculation of the transport quantities and employs an unsteady, 3-D, second order upwind scheme. As the work has importance in applications of air-cooled heat exchangers, practical values have been chosen for air velocity, air temperature, fin spacing and clearance between fin outer diameter and duct wall. In experimental determination of the performance of a finned-tube bundle, only overall average values such as drag coefficient and overall Nusselt number are possible. Local measurements are well nigh impossible, as any measurement instrument introduced into the narrow fin space will immediately change the flow field. This work gives an insight into variations of shear and heat transfer that will help the designer to optimize the fin spacing. The validity of the results for instantaneous velocity profiles and Nusselt number distribution comes from their physical plausibility. The agreement of the logical behavior of the studied variables when the fin space or fin clearance is modeled confirms the adequacy of the numerical simulation. The strong viscous effects caused by decreasing the fin space result in an increase in C_d and change in its frequency. Vortices generated on the rear section of the tube are damped, but the flow still shows small oscillations downstream of the tube, which indicates that vortex generation still exists, but has changed its location. Vortices augment the Nusselt number locally with increasing fin distance. Simultaneously, the opposite effect of converging boundary layers and therefore accelerated core flow in the fin space yields a maximum in the average Nusselt number as a function of fin space. The impact of decrease in the clearance between the tube and the duct wall is not as important as the effects of fin space.

KEYWORDS

Finned cylinder, crossflow, heat transfer, numerical solution

INTRODUCTION

Heat exchangers form an important part of any application, which involves heat transfer. Of these, circular finned tubes are of particular interest as they are commonly used due to their features like manufacturing simplicity, low pressure drop and augmentation. Prediction of flow and heat transfer in such configurations can lead to an optimized heat exchanger design. The purpose of the present study is to simulate and examine the airflow past a circular finned cylindrical tube for application in heat exchangers. Even though the physics of the problem has been reasonably well understood, mathematical solution and experimental data are not available. Therefore a numerical solver formulation has been applied.

The present study is an interesting problem in both fluid dynamics as well as in heat transfer. The clearance space between the fin and the duct wall force the flow into the fin space thereby increasing heat transfer. But if the fins are very close to each other the resistance to flow will be higher in the fin-space. The majority of the flow will take place outside the fin-space, reducing the heat transfer. Obstruction to flow by the tube forms an inherent part of the problem and the role of turbulence is unclear

The geometric dimensions and air velocity chosen for the investigation are based on practical heat exchangers. Water is the tube-side fluid and forced circulation of air is assumed to take place over the tubes.

To aid numerical solution the control volume is discretized into many small elements. All the governing equations are solved for each of these elements. This is done iteratively until a suitable convergence criterion is met. The Finite Volume Method is adopted because of its ability to accommodate any type of numerical grid.

The most accurate approach to simulate any kind of flow (including turbulence) is to solve the Navier-Stokes equations without any approximations. This method is called Direct Numerical Simulation (DNS). This method is both time consuming and needs enormous computational power. It has been shown that when considering problems where flow takes place around bluff bodies these turbulent-models are often either unstable or not adaptable [1].

Like the DNS-model, the laminar model solves the governing equations without any additional averaging or approximations. Hence the laminar model can be suited to model the sub-critical flow in this situation and it has shown accurate results in modeling fluid flow up to $Re=10^5$ for flow past bluff bodies [2].

NOMENCLATURE

C_d Drag Coefficient
D_f Outer diameter of fin
D_T Tube diameter
h Heat transfer co-efficient
k Thermal conductivity of air
m_c Mass flow rate in clearance
m_f Mass flow rate in fin space
Nu Nusselt number ($h \times D_T/k$)
Re Reynolds number ($U_\infty \times D_T/\nu$)
U_∞ Approach velocity
δ_f Fin space
δ_c Clearance space between fin and duct
ν Kinematic viscosity of air

Domain Geometry

The domain geometry and the boundary conditions applied are discussed in this section. The geometry needs to be as simple as possible to reduce storage requirements and also computational time. However, every simplification made makes the model more unrealistic. Hence an optimum level of simplification is required. The control volume considered in this analysis is shown in Fig. 1.

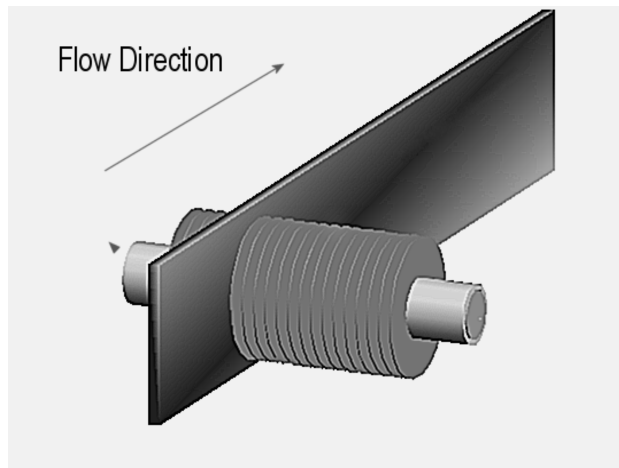

Figure 1: Domain Geometry

The computational domain is bounded by the mid-planes of two adjacent fins. The control volume extends to sufficient distance past the fins to capture the vortices. As the flow before the fins is essentially plug flow, the length considered before the fins is small. Such a control volume considered serves a number of purposes. The domain taken can be considered as representative of the flow between any two fins in a large row of fins. Such a domain may also be taken as a first approximation to a fin tube bundle and greatly reduces the computational time and memory which would otherwise be required if we were to model it.

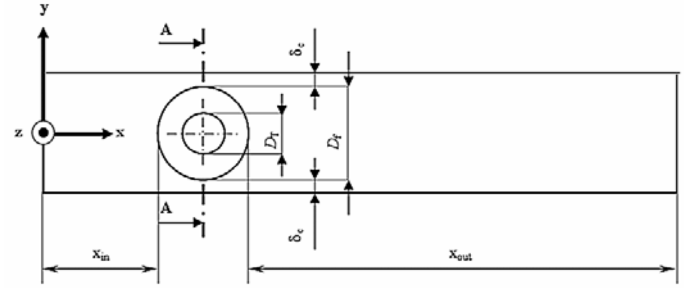

Figure 2: Dimensional details - Elevation

The boundary conditions are applied to this domain taking into account that the model is one of a series of fins. Hence the fluid faces on the sides are made periodic such that the conditions on the left wall are reflected on the right wall. The inner wall of the tube is set at constant temperature, which is generally the case in a typical heat exchanger. Since the control volume passes through the middle of the fin, the fin mid-plane can be considered insulated on account of symmetry.

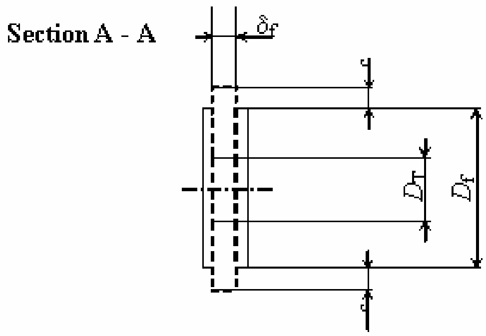

Figure 3 : Dimensional details - Side View

Air at the inlet has a constant velocity. Heat flow through the inner tube, the inner face of the fins and the fin tips are considered for the analysis. The entire duct is taken into account, as the duct mid-plane cannot be assumed to have symmetry, due to vortex shedding.

The upper and lower walls are considered as real walls and no-slip condition is applied to them. The intention in the present work is to analyze the flow over a single cylinder. The physical understanding obtained may be used to interpret the flow over a bundle of finned tubes.

The dimensions and the initial conditions chosen are taken to be typical of that of commercial applications such as cross flow finned air-coolers. The velocity of air along the major flow direction is taken to be 2.5m/s. The velocity in the other directions at inlet is assumed to be 0, representing a plug flow. The inlet temperature of air is taken as 293K and the temperature of the inner wall of the tube is taken to be a constant 323K. All the dimensions, except the fin spacing and fin clearance, are kept constant. The various dimensions are marked in the Fig.2 and Fig.3 and their values are given in Table 1.

Parameter	Value (mm)
D_T	25
D_f	57
X_{in}	$\approx 1.25\ D_f = 100$
X_{out}	$\approx 6.5\ \ D_f = 400$
δ_f	variable (1.5, 2, 2.5)
δc	variable (1, 2, 3, 4, 5)

Table 1 : Domain Dimensions

The variation in the fin clearance and the fin spacing leads to fifteen different configurations for which models were created and analyzed. Further, different meshing schemes were used to ensure mesh independence of the model. One such scheme used for meshing the fin region is shown in Fig.4.

Solution modeling

The flow is assumed to be Newtonian, incompressible and is modeled by the Navier-Stokes equation.

$$\rho\ \frac{Du}{Dt} = -\frac{\partial p}{\partial x} + \mu\ \nabla^2 u + \rho\ b_x \tag{1}$$

$$\rho\ \frac{Dv}{Dt} = -\frac{\partial p}{\partial y} + \mu\ \nabla^2 v + \rho\ b_y \tag{2}$$

$$\rho\ \frac{Dw}{Dt} = -\frac{\partial p}{\partial z} + \mu\ \nabla^2 w + \rho\ b_z \tag{3}$$

where *u, v, w* are the velocity components and *b* represents the body forces.

Fluent v6.0, a commercial CFD solver is used for solving the present problem. As mentioned previously, the laminar model is chosen to analyze the present problem. As the flow is bound to shed vortices, unsteady formulation is used to model the problem. The segregated scheme is used which solves the momentum and energy equations individually.

As the velocity and pressure terms are coupled in the Navier-Stokes equations, a solution requires one of them to be guessed to solve the other. The assumption is then rechecked with the obtained values. This is implemented in the pressure-velocity coupling algorithm. The method of coupling used in this case is the Pressure-Implicit Splitting of Operators (PISO). This method is most suitable for time dependent problems. In this method, the pressure field is first assumed and is used to obtain the velocity components. The values of velocities obtained are then used to solve the pressure equation. This process is repeated till convergence. The advantage of the PISO algorithm is that under-relaxation factors of 1.0 for momentum and pressure may be used if the mesh is not highly distorted. For the used mesh the under-relaxations factors are set to 1.0 as the mesh is structured without unusual distortion.

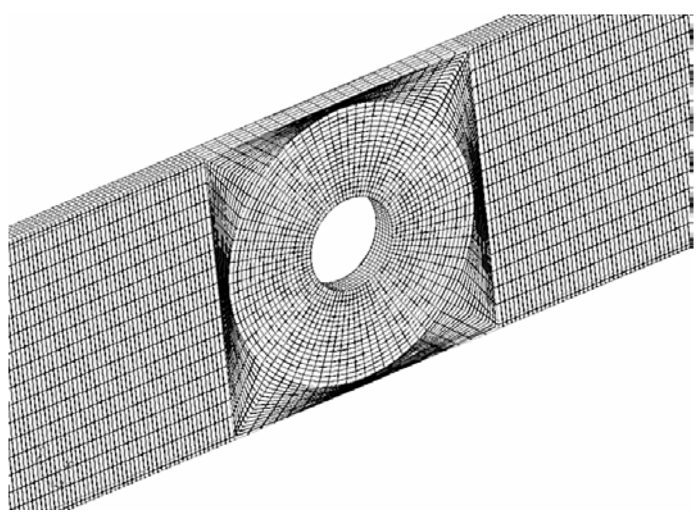

Figure 4 : Meshing of model

For better accuracy, the quadratic upwind interpolation scheme is used in the solver. The standard properties for the fluid medium (water) and the solid medium (aluminum) are taken.

To model transient phenomenon, it is necessary that the time step taken is smaller than the smallest time constant in the system. This criterion is given more explicitly in the Courant number. Effectively, the Courant number ensures that the time step is smaller than the time the fluid needs at its highest velocity to cross the smallest cell of the domain. For such a criterion, the Courant number needs to be taken less than 1 [3]. Taking a value of 0.5 for the Courant number, we obtain a time step of 2.0×10^{-4} seconds, which is used in the calculations.

Grid independence

To ascertain that the solutions obtained are independent of the grid, the problem was solved using two grid distributions. One was solved using the grid obtained by dividing the space between the fins into ten cells and the other was solved with five cells. The results obtained were comparable, with 4 mm fin space giving the maximum error band of –1% to 1% and 2 mm fin space giving a minimum error band of 0 to 0.2% in heat flux.

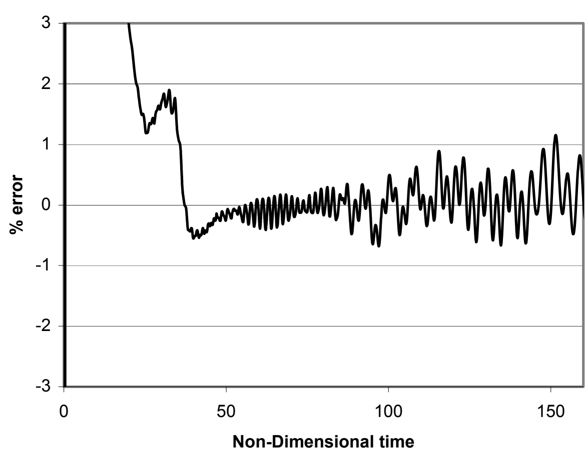

Figure 5 : Difference in average heat flux for 4mm fin space between two grid sizes

From Fig.5 and Fig.6, the solution obtained can be assumed to be independent of the grid within reasonable accuracies.

Variation of Drag Coefficient with Fin Space

The flow, which takes place over the cylinder, is essentially asymmetric due to vortex shedding. Once asymmetry is initiated alternate shedding occurs from the two sides of the cylinder. As the flow stabilizes, the C_d values start oscillating about a mean value. This mean C_d value gives an indication about the force acting on the tube and hence gives a good estimate of the pressure drop.

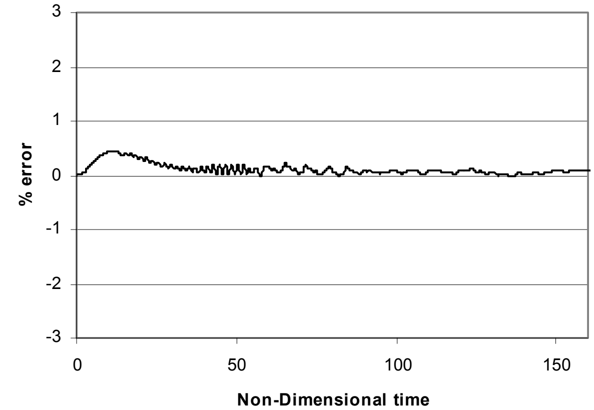

Figure 6 : Difference in average heat flux for 2mm fin space between two grid sizes

From Fig.7, it is clear that the average C_d decreases exponentially as the fin space increases.

It can be seen that for a constant fin space, decreasing the clearance forces the flow into the fin space. The division of flow between the fin space and clearance is not proportional to the area, but rather to the flow resistance. The decrease in fin space has a much greater effect than a reduction in clearance in the distribution of flow.

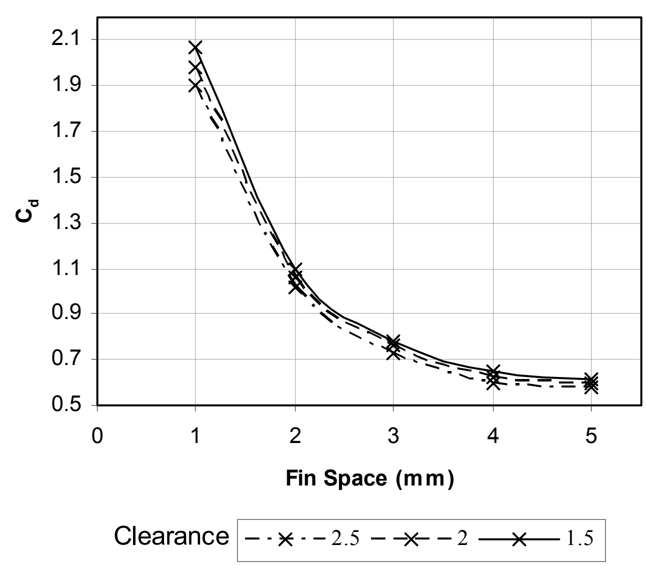

Figure 7 : Drag Co-efficient variation

The mass flow rate shown in Fig.8 is obtained by integrating over the cross sectional area of the fin space and clearance. The resulting values are checked for consistency by ensuring that the total mass flow rate equals that at the inlet with reasonable accuracy.

$$\dot{m}_C^* = \frac{\dot{m}_C}{\dot{m}_{inlet}} \text{ for the clearance} \tag{4}$$

$$\dot{m}_f^* = \frac{\dot{m}_f}{\dot{m}_{inlet}} \text{ for the space} \tag{5}$$

As it can be observed that a major portion of the flow occurs through the fins, it is inferred that the fin clearance does not greatly affect the flow.

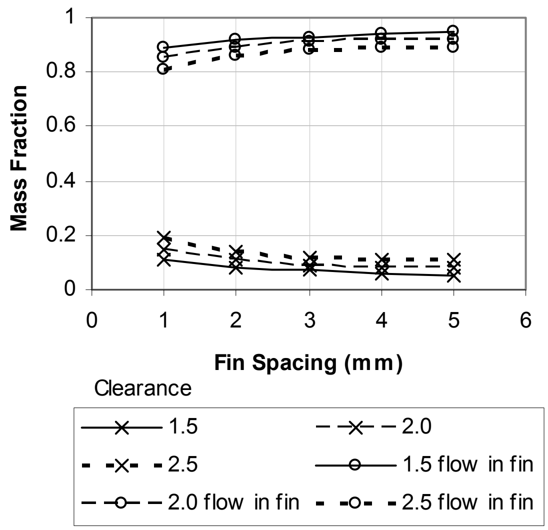

Figure 8 : Flow rate fraction through fins

Path Lines

To get an idea on how unsteady flow develops on entering the fin space or clearance, the path lines are shown for every fin space with a clearance δ_C = 1.5 mm in Fig.9. The path lines give a first impression of flow behavior in a domain. The tangent on a path line shows the direction of the velocity vector, whose magnitude usually varies along the path line.

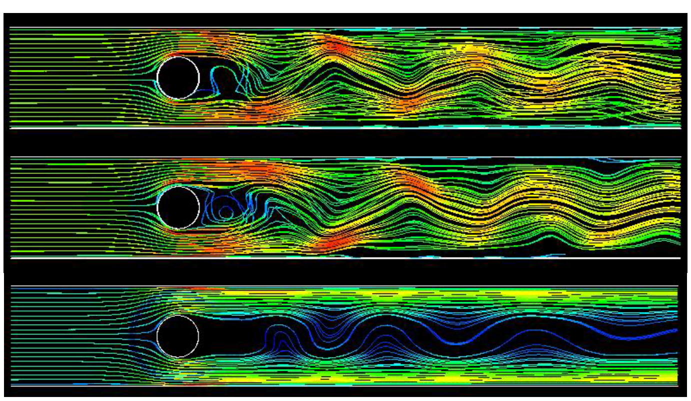

Figure 9 : Path Lines colored by X-Velocity for 5mm, 3mm and 1mm Fin Spacing

The distance between path lines lying next to each other shows fluid acceleration or deceleration for convergence or divergence of path lines respectively. To intensify the impression of the magnitude of velocity the path lines are colored. The path lines are colored with regard to the x-velocity, so that backflow is more clearly seen behind the cylinder.

For 2D problems the path lines can be displayed as they result out of the problem solution. In 3D this is not quite correct as the 3rd dimension is very difficult to display and just neglecting it can place a wrong impression. For the present problem the velocities in z-direction being small compared to the main flow velocity, conclusions arrived on the basis of 2D path lines are quite realistic.

The temperature contours also follow a similar trend as the velocity contours. From these plots we can conclude that the laminar model used to solve the problem is valid.

Velocity profile at Y Line

The velocity distribution along the centerline of the tube shows a maximum close to the tube wall. Due to the no slip condition, the velocity goes to zero both at the bottom and top walls as well as at the tube wall. This maximum is due to the combination of two complementary effects. One is the acceleration of the flow due to the obstruction of the flow by the tube. The second is due to the increase in the path length of the flow near the smaller radii of the tube rather than at the larger radii. The second fact is clearly illustrated by Fig.10. The flow decelerates due to the boundary layer and by continuity, reduced flow in the boundary means that the remaining flow should take place in the core. This increased flow in the core accelerates the fluid. Thus longer the boundary, greater is the flow acceleration.

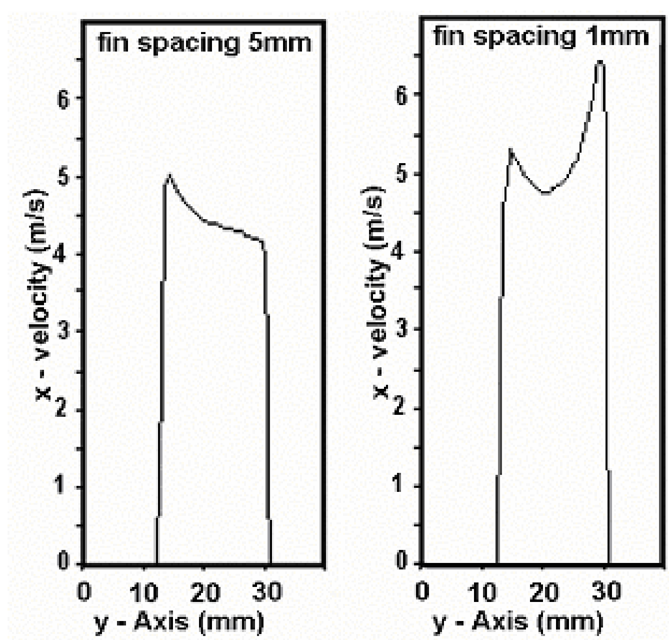

Figure 10 : Velocity along Y-axis

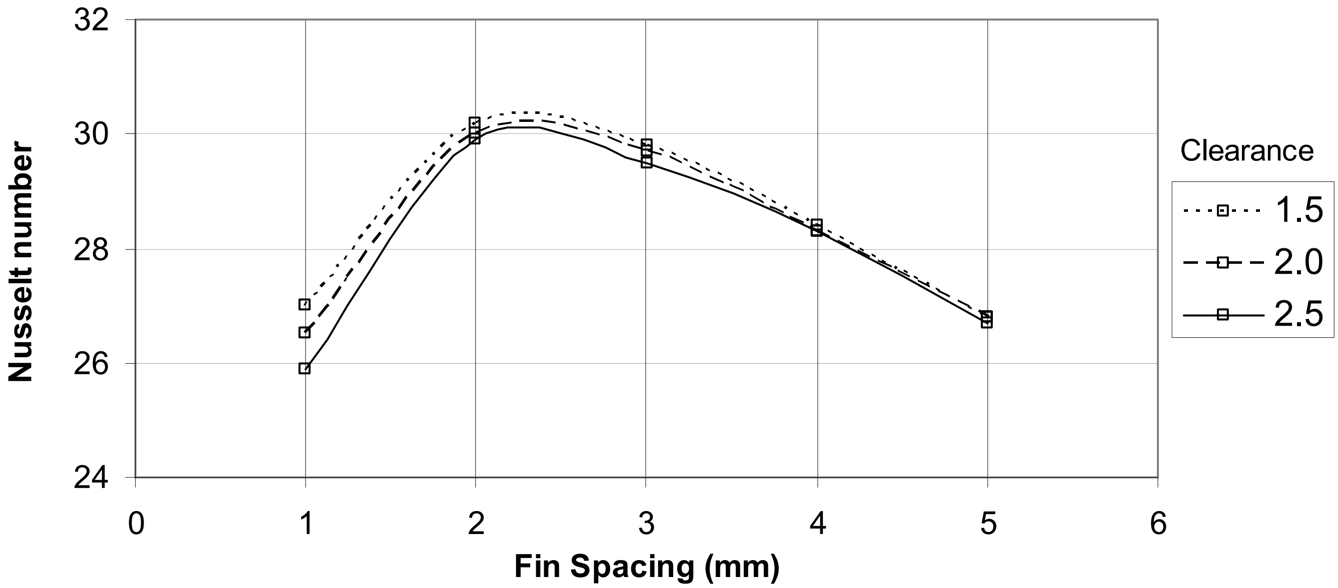

Figure 11 : Nusselt number Variation

In Fig.10 the x-velocity is given for 1 mm fin space. It can be seen from the data for mass flow distribution (Fig.8) that the interstitial mass flow rate is less for 1 mm fin space as compared to 5 mm fin space. This is clearly on account of the greater flow resistance. However, the mid-plane velocities are in general higher in the 1 mm fin space.

For a smaller fin space, the boundary layer occupies a relatively larger part of the flow cross section, thus severely constricting the core flow. The acceleration of flow at all radii in the core is observed. The minimum in the mid-plane is the result of two contradictory effects. The first is the acceleration of the core flow as stated above. The second is the momentum transfer from the flow in the clearance. The mass flow distribution (Fig.8) also shows that a higher fraction of flow takes place in the clearance.

Average Nusselt number

From Fig.11, it can be seen that at any fin space decreasing the clearance redirects the flow through the fin space, increasing the Nusselt number. However this increase is not significant. Hence the effect of clearance on the heat transfer can be ignored.

On decreasing the fin space, the Nusselt number first increases up to a fin space of 2 mm but then drops steeply. This can be due to the fact that the boundary layers, growing from two adjacent faces of the fins, converge and the flow is similar to the fully developed flow between two parallel plates. A Hele –Shaw type of flow might be considered to occur at these low fin spacings.

Conclusions

The optimum fin spacing for the cases studied is found to be 3mm. The pumping power moderately decreases from 2mm to 4mm fin space. Hence, an optimum value of fin spacing in this range depends on the application. Decreasing the fin space below 2mm leads to a high pressure drop as well as lower heat transfer.

ACKNOWLEDGMENTS

The help of Mr. Bansode Annasaheb Sampat, graduate student, in running the simulations is gratefully acknowledged.

REFERENCES

[1] Jalaiah, N., and Raghavan, V.R., 2002, ”Effects of blockage on flow and heat transfer over a tube in cross flow”, Proc., 12th International Heat Transfer Conference, Grenoble, France, pp. 711-716.

[2] Jalaiah, N., and Raghavan, V.R., 2002, ” Numerical simulation of flow past a circular cylinder with integral wake splitter”, Proc., International Conference on Applied Computational Fluid Dynamics, Beijing, China, pp. 170-177.

[3] Stebener H., 2001, “*Numerical analysis of flow past a circular finned cylindrical tube*,” Diplomarbeit, Indian Institute of Technology Madras and University of Hamburg.