

Experimental and Numerical Investigation of Convective Heat Transfer in a Circular Pipe with Internal Ring ribs

دراسة عملية ونظرية لانتقال الحرارة داخل أنبوب دائري ذو أضلاع حلقيّة

Pro.Dr Arkan Al-taee
University of technology
Email:arkanaltaie @yahoo.com

Ass.Lec. Hasasn Ali Jurmut
Technical college
Email:Hasan_jurmut@yahoo.co

Abstract

This paper studied an experimental and CFD investigation of heat transfer characteristics of horizontal circular pipe 500mm long using internal rings ribs of 80mm width ,80mm height, $n=7$, $p =100$ mm, with air as the working fluid. Reynolds number 31170 was taken. The steel pipe(ASM4120) was subjected to different constant surface temperatures(573,873,and 1173K^o). The experimental data obtained were compared with plain (without ribs)case. Based on the same coolant flow, the pipe with internal ring ribs was found to possess the highest performance factors for turbulent flow. The results show a good agreement between theoretical and experimental by factor 3.6%. The heat transfer rates obtained is 32.22times over the smooth channel. for a given Reynolds number. The use of internal ring ribs improved the heat transfer in circular pipe. All studies where carried out using workbench programFIUENT14.5.

Keywords-cooling enhancement, internal ring ribs , heat transfer and turbulent flow.

المخلص

تمت في هذا البحث دراسة عملية ونظرية لانتقال الحرارة خلال أنبوب دائري طوله (500ملم) باستخدام أضلاع حلقيّة (القطر الخارجي 49 ملم وقطر داخلي 33 ملم) داخل الأنبوب بعدد(7). المانع المستخدم (الهواء) أخذ رقم رينولد 31170 وباستخدام درجات حرارة ثابتة لسطح الأنبوب(573,873,1173 كلفن). وقد أظهرت النتائج اقتراب جيد بين الجانب النظري والعملية بمقدار 3.6% وان كمية الحرارة المنتقلة بمعدل 32.22 مرة مقارنة بالأنبوب بدون أضلاع. وان استخدام الاضلاع يحسن من انتقال الحرارة وقد استخدم برنامج فلونت 14.5 لكافة الحسابات.

1. Introduction

In the design of channel heat exchangers, rib, fin or baffle turbulators are often employed in order to increase the convective heat transfer rate leading to the compact heat exchanger and increasing the efficiency. For decades, rib turbulators have been applied in high-performance thermal systems due to their high thermal loads. The cooling or heating air is supplied into the channels with several ribs to increase the stronger turbulence intensity of cooling or heating levels over the smooth wall channel. Ribs placed in tandem in the channels interrupt hydrodynamic and thermal boundary layers because downstream of each rib the flow separates, re-circulates, and impinges on the channel walls that are the main reasons for heat transfer enhancement in such channels. The use of ribs not only increases the heat transfer rate but also substantially the pressure loss. In particular, the rib geometry, the rib-to-channel height ratio and the rib pitch-to-height ratio are the parameters that affect the heat transfer rate and the thermal performance.

Many attempts have been made to study the effect of these parameters of ribs on heat transfer and friction factor for two opposite roughened surfaces. Han et al. [1,2] investigated experimentally thermal characteristics in a square channel with angled ribs on two walls and found that the angled ribs and 'V' ribs yielded higher heat transfer than the continuous ribs and the highest value is at the 60° amongst the angled ribs. For heating either only one of the ribbed walls or both of them, or all four channel walls, they reported that the former two conditions resulted in an increase in the heat transfer with respect to the latter one. For broken ribs with $e/D=0.0625$ and $P/e=10$ placed in a square channel, Han and Zhang [3] also found that the 60° broken 'V' ribs give higher heat transfer

at about 4.5 times the smooth channel and perform better than the continuous ribs. Liou and Hwang [4,5] carried out an experiment to study the performance of square, triangular and semi-circular ribs by using a real time Laser Holographic Interferometry to measure the local as well as average heat transfer coefficients. They reported that the square ribs give the best performance among them. The heat transfer behaviors in a ribbed square channel with three e/D ratios ($e/D=0.083, 0.125$ and 0.167) and a fixed $P/e=10$ using a liquid crystal technique were examined by Taslim et al. [6]. They found that the average Nusselt number increased with the rise in e/D ratio and the best e/H ratio was seen to lie between 0.083 and 0.125 . Turbulent convective heat transfer behaviors in square ducts with ribs on two opposite walls and discrete angled ribs on one wall were numerically investigated by Saidi and Sunden [7] and Tatsumi et al. [8], respectively. They found that noticeable heat transfer enhancement is obtained downstream of the ribs due to strong secondary flow motion.

2.Theoretical Model and Numerical Solution

This study presents a description of the mathematical basis for a comprehensive general purpose model of fluid flow and heat transfer from the basic principles of conservation of mass, momentum, and energy. This leads to the governing equations of fluid flow and heat transfer used for the analysis of steady state , three-dimensional, turbulent and incompressible flow in addition to thermal performance and cooling effectiveness of pipes with different ribs configuration heat exchanger .

The results obtained from the experimental investigation work show the behavior of the temperature distribution in the pipe flow region (smooth and rough). The temperature distribution affects the heat transfer coefficients effectiveness estimation values.

The computational fluid dynamics (CFD) becomes one of the most useful tools for complex phenomena without resorting to expensive prototype and difficult experimental measurement. Numerical prediction using **FLUENT** can be performed to determine the temperature distribution and for better understanding the losses of heat transfer. The flow may be considered incompressible as velocity tend to be constant. Like many common fluids such as water, air is a Newtonian fluid, displaying a linear relationship between shear and strain.

This study also presents the governing equations of fluid flow to solve the interaction of coolant air with configuration of ribs. To demonstrate the effect of the turbulence model that involves the solution of two transport equation ($k-\epsilon$) model is used, thus the numerical solution by finite volume and explicit methods will solve these Cartesian coordinate system . Three dimensional geometry is generated and the effects of ribs shape are to be studied. **FLUENT** version (14.5),

3.Governing equation

3.1The Heat transfer Equation:The heat-transfer rate is the amount of heat that transfers per unit time (usually per second). If a hot metal bar has a surface temperature of T_o on one side and T_i on the other side, the basic heat-transfer rate due to conduction can be given by:

$$Q = UA\Delta T \dots\dots\dots(1)$$

If a hot wall at a temperature T_o is exposed to a cool fluid at a temperature T_i on one side, the *convective* heat-transfer rate can be given by:

$$Q = hA\Delta T \dots\dots\dots(2)$$

The conventional expression for calculating the heat transfer coefficient in fully developed turbulent flow in smooth pipes is the Dittus Boelter . Because of the many factors that affect the convection heat-transfer coefficient (h) , calculation of the coefficient is complex. However, dimensionless numbers are used to calculate (h) for both free convection and forced convection .

$$Nu = 0.023Re^{0.8} Pr^{0.4} \dots\dots\dots(3)$$

D_h is the hydraulic diameter express as:

$$D_h = \frac{4A}{P_w} \quad \text{Prandtl number express as:}$$

$$Pr = \frac{\mu c_p}{K} \dots\dots\dots(4)$$

Reynolds number express as:

$$Re = \frac{\rho u D}{\mu} \dots\dots\dots(5)$$

The definition of mass flow is important for correlations. Mass flow is defined in equation :

$$m = \rho u A \dots\dots\dots(6)$$

3.2Energy Balance

Energy balance is applied to the air flowing in the pipe. The energy which is produced by the heaters is carried away by the air flowing inside the pipe and only a small portion of the heat produced is lost by natural convection from the outer surface of the pipe. The energy balance principle:

$$Q = mc_p (T_o - T_i) \dots\dots\dots(7)$$

Furthermore Q is the total heat transferred to air by forced convection and is given by:

$$q = Q / \pi D L \dots\dots\dots(8)$$

$$h = q / (T_{w,x} - T_{w,bx}) \dots\dots\dots(9)$$

4.FLUENT Software Package:

FLUENT is a commercial CFD software package developed by FLUENT, it also includes a mesh generation module called GAMBIT and a solver. The general modeling capabilities of FLUENT relevant to this research are the following:

1. 3-D flows.
2. Finite volume method based on fully unstructured meshes.
- 3.Choice of segregated and coupled solvers - In a segregated solver, the governing equations are solved sequentially whereas in a coupled solver they are solved simultaneously.
4. Steady state or transient analysis.

There are three main steps in any CFD analysis

1. Pre-Processing.2. Solver Execution.3. Post-Processing.
- 1-Pre-Processing** is the step where the modeling goals are determined and computational grid is created. In the second step, numerical models and boundary conditions .
- 2- The Solution** module solves Navier-Stokes ,continuity ,momentum equations as well as the turbulent flow model , heat transfer and temperature distribution.
- 3- The general postprocessor** module is used to display the analysis results .

4.1System Geometry:

The system geometry shown in Figure (1) consists of pipe of 500mm long for the hot air at (constant temperature), pipe with inner and outer diameter (63.2 , 49) mm respectively for coolant air and the same model geometry of ribs as shown in figure (2). The system geometry is drawn by using (Auto CAD 2011).

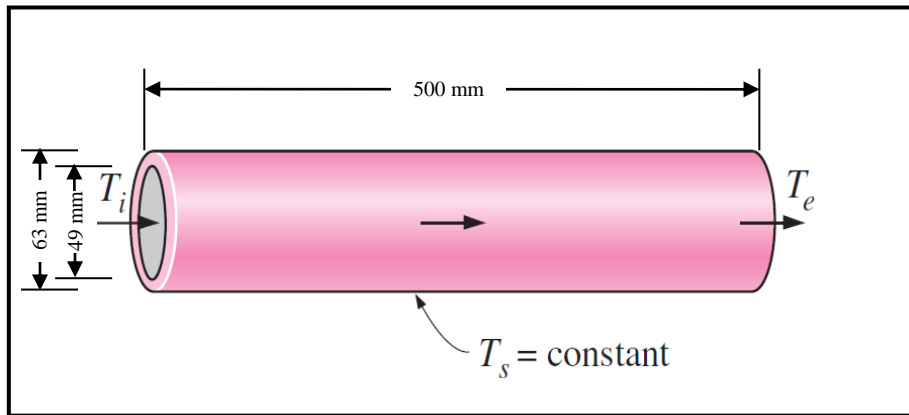


Figure (1) shown the geometry of test section



Figure (2) shown the geometry of ribs

4.2 Mesh Generation:

There are mainly two types of approaches in volume meshing, structured and unstructured meshing. **FLUENT** can use grids comprising of tetrahedron or hexahedron cells in three dimensions. The type of mesh selection depends on the application as shown in figure (3).

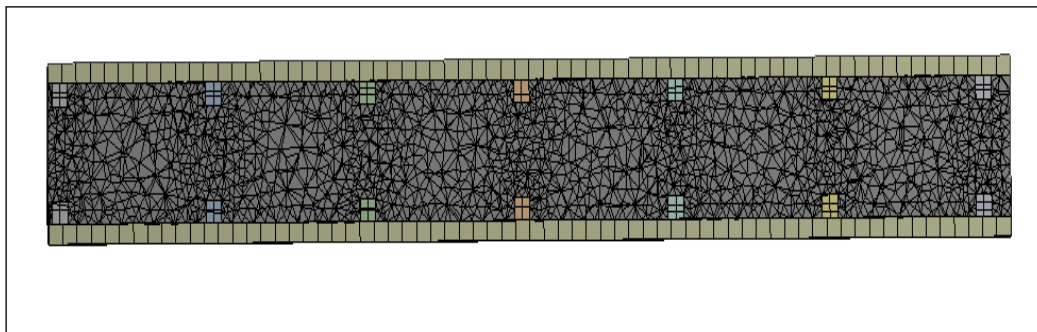


Fig. (3): Mesh Generation

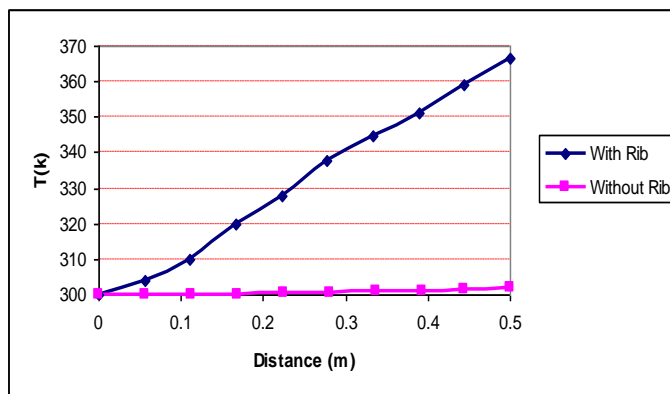
5. Experimental work

The test rig Fig.(14.B) consists of a blower unit fitted with a pipe, which is connected to the test section located in horizontal orientation. The blower we have used is an ordinary dust blower. It has a converging nozzle (400mm) long outlet which Air was made to flow through the test pipe by means of blower motor with velocity (10 m/sec). A heat input of 1000 W was given to the nichrome heating wire on the test pipe by adjusting the dimmer stat. The test section was insulated in order to avoid the loss of heat energy to the surrounding. Thermocouples (2 to 4) were fixed on the test surface and thermocouples(1and5) were fixed inside the pipe (inlet and outlet) respectively. The readings of the thermocouples were observed every (10 minutes) until the steady state condition was achieved. Three thermocouples (T_2 , T_3 and T_4) at a distance of (15 , 30 , and 45 cm) from the origin of the heating zone are embedded on the walls of the pipe and two thermocouples

are placed in the air stream, one at the entrance (T_1) and the other at the exit (T_5) of the test section to measure the temperature of flowing air as shown in Fig.(14.A). The blower consists a plate , which controls the airflow rate through the pipe and an orifice meter to find the volume flow rate of air through the system.

6.Result and discussion

Figure (4), shows the temperature distribution of cooling air along the pipe, without rings and with rings. It shows that cooling air temperature increase from 300K at entrance to 302.04K at exit for that without rings. While for that with rings ,temperature increase from 300K at entrance to 366.28K at exit, i.e an increase of 66.28K is experienced .This gives indication of the amount of heat carried away with cooling air due to the addition of rings. Addition of ring increase surface area of heat transfer and hence will reduce metal temperature. Reducing metal temperature will increase its integrity and life.



Figure(4) Temperature distribution along the pipe for surrounding temperature of(573K)

Figure (5), shows the temperature distribution of cooling air along the pipe, without rings and with rings. It shows that cooling air temperature increase from 300K at entrance to 304.63K at exit for that without rings. While for that with rings ,temperature increase from 300K at entrance to 439.45K at exit, i.e an increase of 139.45K is experienced .This gives indication of the amount of heat carried away with cooling air due to the addition of rings. Addition of ring increase surface area of heat transfer and hence will reduce metal temperature. Reducing metal temperature will increase its integrity and life.

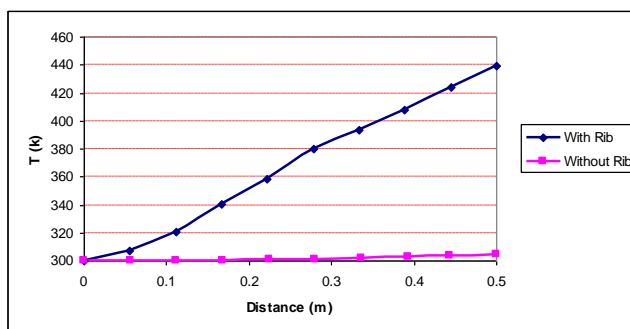


Fig. (5) Temperature distribution along the pipe for surrounding temperature of(873K)

Figure (6), shows the temperature distribution of cooling air along the pipe, without rings and with rings. It shows that cooling air temperature increase from 300K at entrance to 307.35K at exit for that without rings. While for that with rings ,temperature increase from 300K at entrance to 513.46K at exit, i.e an increase of 213.46K is experienced .This gives indication of the amount of heat carried away with cooling air due to the addition of rings. Addition of ring increase surface area of heat transfer and hence will reduce metal temperature. Reducing metal temperature will increase its integrity and life.

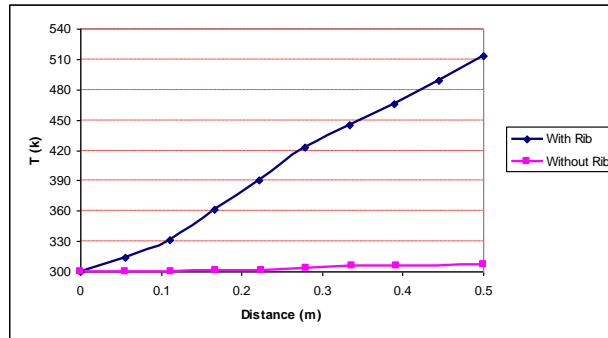


Fig. (6) Temperature distribution along the pipe for surrounding temperature of (1173K)

Figures (7 to 9) show the temperature contours inside the pipe, for cooling air temperature of (300K) and surface temperature of (573,873 and 1173K) ,respectively. Cooling air velocity is (10m/s). The disturbance caused by ribs is very clear. Its effect on temperature distribution is also evident. Wakes generated by ribs and developed into vortices are clear in creasing cooling air temperature. Hence, the effect of ribs on cooling of metal temperature is very clear. However, these figures show different effects depending on the type of ribs used. Figures(7-9) shows temperature contour of cooling air flow along the pipe at surrounding air temperature of 573,873,and 1173K,respectively for a pipe with ring ribs. It shows that in the first 1/3 of length of the pipe the cooling air temperature remains constant at the center but increases near the walls. However , at the second segment (1/3 of length), it shows an increase in cooling air temperature even at the center of pipe. While, at the end segment completely reached (366.26 ,439.45,and 513.64K) respectively , it shows a uniform elevated air temperature. This is a good indication that as flow progresses in to the pipe, it temperature increases until eventually became uniform.

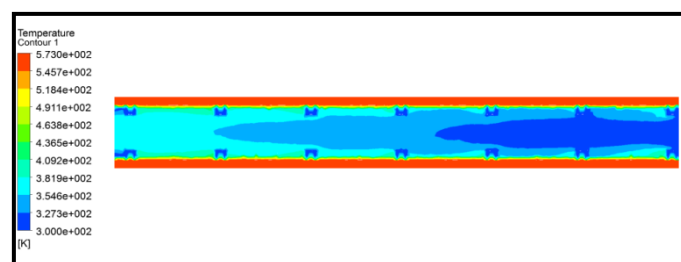


Fig.(7) Contour of temperature for constant surface temperature(573K)

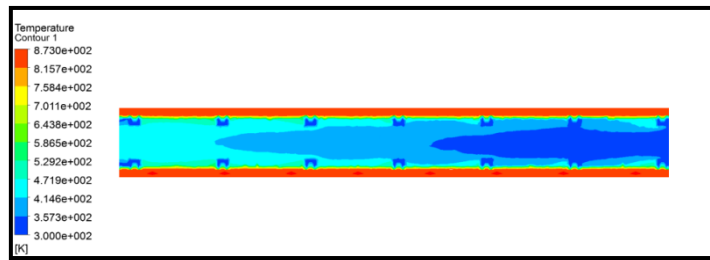


Fig.(8) Contour of temperature for constant surface temperature(873K)

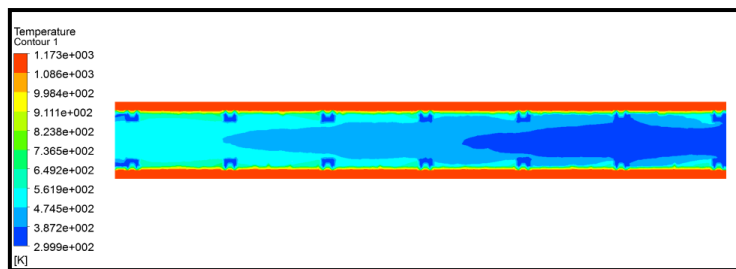


Fig.(9) Contour of temperature for constant surface temperature(1173K)

Figure(10) show a good agreement between the theoretical and numerical results.

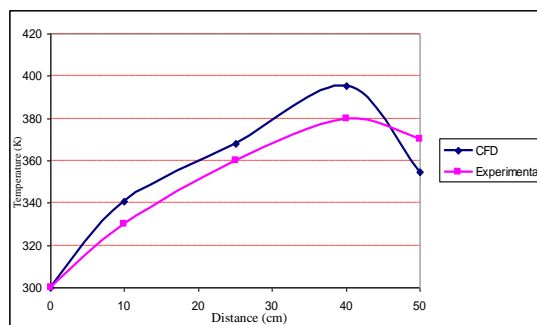


Fig.(10): Temperature distribution

For all case as shown in figures (11), the air velocity is seen to be the same in spite of using different constant temperature at pipe environment over the same Reynolds number.

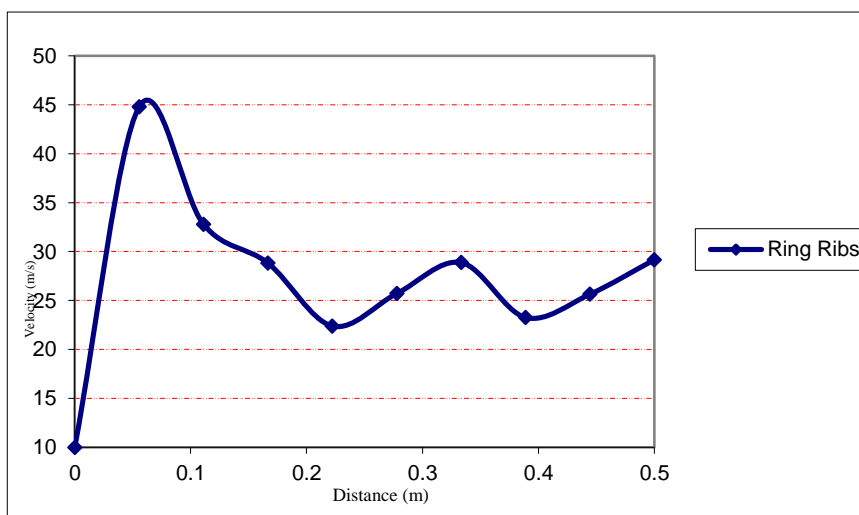


Fig.(11): Velocity distribution

Figures (12) the contours of velocity shown the fluid flow behavior inside the pipe at using ring ribs, the flow accelerated and decelerated along the pipe, the velocity increase in moment the fluid at entrance of rings rib. Hence the effect of ring ribs causes different velocity variation due to the shape of rib.

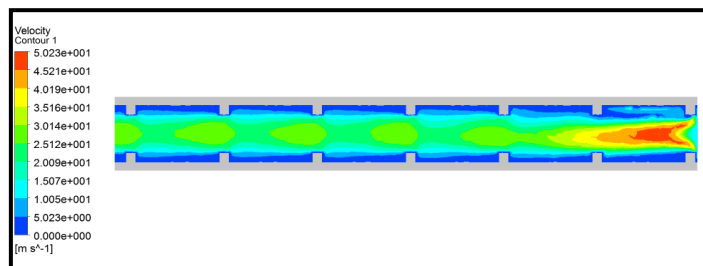


Fig.(12) Contour of velocity

Figure (13) show the cooling air velocity vector for Reynolds number of 31170. It can be noted for figure as the flow approaches the first row of ribs, it accelerates around it and boundary layer flow with separation is observed downstream of ribs. A pair of counter rotating vortices are formed downstream face of each rib[9]. The main stream flow accelerates in the gap region before impinging on the leading edge on the second row ribs which are staggered to the first row ribs. The effect of wake behind the rib decreases due to the increase of velocity between ribs. the velocity from the top and the bottom of the rib increases before the separation point. At the stagnation point in leading edge, the velocity is decreased [10].

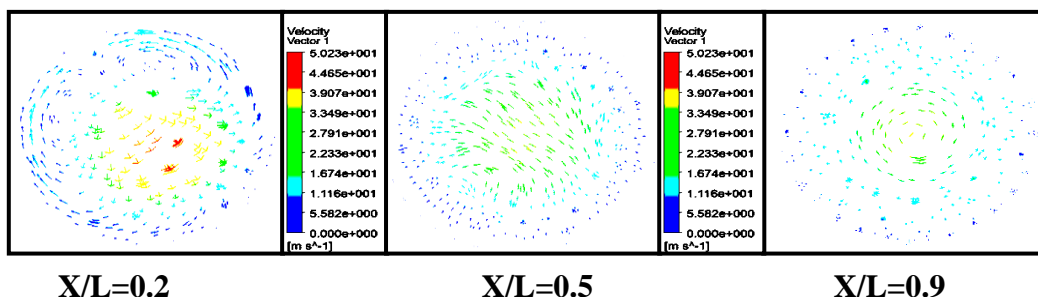
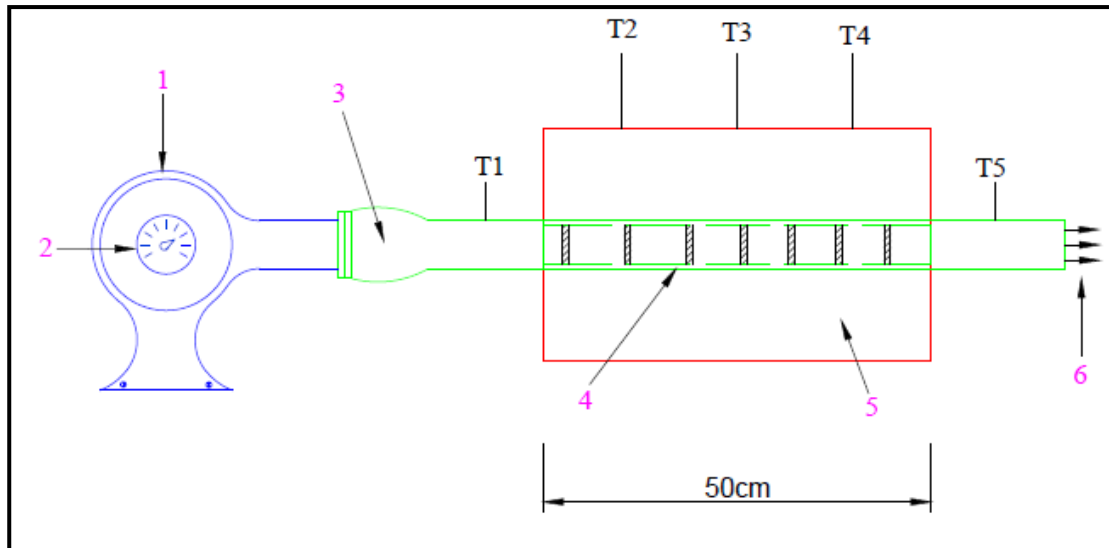


Fig. (13): Velocity vector around the rib.

Conclusions

In the present work, a numerical and experimental study was performed, with the aim to assess the effect of using ribs on temperature distributions along a horizontal pipe . The main conclusions are:

1. The theoretical results were compared with experimental and showed a good agreement by factor 3.6%.
2. With out ribs the temperature was found to increase by 2.04,4.35and 7.35K for cases 1,2,and 3, respectively. While when ribs used , the effect was to increase the temperature by66.26, 139.45, and 213.64 K for cases 1,2,and 3, respectively.
3. The heat transfer in the test section without ribs was found to increase by45.1,96.39 and163.2W for cases 1,2,and 3, respectively. While when ribe used the heat transfer increase by1468.36,3098.49,and4732.15W for cases 1,2,and 3, respectively . This could be due to the fact that the flow has a high recirculation zone between the ribs, as shown in figure (13) .



A

- 1 Blower
- 2 Velocity Control
- 3 Nozzle
- 4 Test Section
- 5 Box With Constant Temperature
- 6 Outlet
- T₁-T₅ Thermo Couple



B

Fig. (14)Show (A) Schematic diagram of Experiment Setup.
(B)Actual diagram of Experiment Setup.

Nomenclature

A	Heat transfer area, m ²
C _p	Specific heat of air, J/kg.K
D _h	Hydraulic diameter, m
e	Height of rib, m
F	Frictional factor
g	Acceleration due to gravity ,m/s ²
h	Heat transfer coefficient, W/m ² k
p	Pitch of rib, m
L _t	Length of test pipe, m
m	Mass flow rate, kg/s
R _e	Reynolds number
Q	Heat transfer rate, W
U	Overall heat transfer coefficient ,W/m ² K
u	Velocity of flow , m/s
μ	Viscosity of air , N s/ m ²
ρ	Density of air, kg/m ³
T _{w,x}	Local wall temperature, K
T _{b,x}	Local bulk temperature of air ,K
q	Heat flux, w/m ²

References

- [1] J.C. Han, Y.M. Zhang, C.P. Lee, Augmented heat transfer in square channels with parallel, crossed and V-shaped angled ribs, ASME J. Heat Transfer 113 (1991) 590–596.
- [2] J.C. Han, Y.M. Zhang, C.P. Lee, Influence of surface heat flux ratio on heat transfer augmentation in square channels with parallel, crossed, and V-shaped angled ribs, ASME J. Turbomachinery 114 (1992) 872–880.
- [3] J.C. Han, Y.M. Zhang, High performance heat transfer ducts with parallel broken and V-shaped broken ribs, Int. J. Heat Mass Transfer 35 (1992) 513–523.
- [4] T.M. Liou, J.J. Hwang, Turbulent heat transfers augmentation and friction in periodic fully developed channel flows, ASME J. Heat Transfer 114 (1992) 56–64.
- [5] T.M. Liou, J.J. Hwang, Effect of ridge shapes on turbulent heat transfer and friction in a rectangular channel, Int. J. Heat Mass Transfer 36 (1993) 931–940.
- [6] M.E. Taslim, T. Li, D.M. Kercher, Experimental heat transfer and friction in channels roughened with angled, V-shaped, and discrete ribs on two opposite walls, ASME J. Turbomachinery 118 (1996) 20–28.
- [7] A. Saidi, B. Sunden, Numerical simulation of turbulent convection heat transfer in square ribbed ducts, Numer. Heat Transf. A Appl. 38 (1) (2000) 67–88.
- [8] K. Tatsumi, H. Iwai, K. Inaoka, K. Suzuki, Numerical analysis for heat transfer characteristics of an oblique discrete rib mounted in a square duct, Numer. Heat Transf. A Appl. 53 (11) (2008) 1156–1175.
- [9] Sundberg, “Heat Transfer Correlations for Gas Turbine Cooling”, M.Sc. Thesis, Department of Mechanical Engineering Linköping University, Linköping, Sweden, (2006).
- [10] Guoguang, S.u.; Hamn, Ching, Chen and Je-Chin, Han "Computational of flow and Heat Transfer in rotating Rectangular Channels (AR=4:1) With Pin-Fins by a Reynolds stress Turbulence Model", Journal of Heat Transfer, Vol.129 June 2007, pp.685-696.