NUMERICAL INVESTIGATION OF HEAT TRANSFER ENHANCEMENT IN A CIRCULAR TUBE USING RIBS OF SEPARATED PORTS ASSEMBLY

Farhan Lafta Rashid
Prof. Dr. Arkan Altaie
Ass. Prof. Dr. Moayed R. Hasan
University of Technology-Iraq

Abstract

The paper deals with numerical investigation for the influence of separated ports assembly ribs on heat transfer ina steel tube of 50 cm long , outside diameter of 60 mm and inside diameter of 30 mm with constant outside surface temperature of 1000, 1200 and 1400 K^o.The renormalization group k-ε model is used to simulate turbulence in ANSYS - FLUENT 14.5. The ribs assembly (5x5 mm triangle passage) were fitted in the tube and separated by 8cm pitch. Results of temperature and velocity distribution along the tube center line for the case of tube with internal ribs were compared with that of plain tube , these results show that the use of internal ribs enhance the heat transfer rate and found to possess the highest performance factors for turbulent flow.

Keywords: CFD, heat transfer enhancement, cooling enhancement, internal ribs, turbulators and turbulent flow

Nomenclature

E: Empirical constant in turbulence model (9.793)
g: Gravity acceleration, m/s2
k: Thermal conductivity,W/m K
P: Mean pressure, Pa
T: Temperature,K
ui: Velocity vector, m/s
u´ : Root-mean-square of the turbulent velocity fluctuations
x, y, z: Coordinates

Greek Symbols

μ: Dynamic viscosity, Pa·s
ρ: Density, kg/m3
τ : Shear stress

Introduction

Heat transfer enhancement is a subject of considerable interest to researchers as it leads to saving in energy and cost. Because of the rapid increase in energy demand in all over the world, both reducing energy lost related with ineffective use and enhancement of energy in the meaning of heat have become an increasingly significant task for design and operation engineers for many systems[Veysel, 2008].

Heat transfer enhancement by inserting ribs is commonly used application in tubes. Ribs improve the heat transfer by interrupting the wall sublayer. This yields flow turbulence, separation and reattachment leading to higher heat transfer rates. Due to the existence of ribs effective heat transfer surface increases. Many researches have been carried out on heat transfer enhancement achieved by different ribs [Buchlin-2002, Tanda-2004,Jordan-2003,San-2006,Lu-2006,Tanda-2001].

Yakut et al.[K. Yakut, 2004] investigated the role of conical-ring turbulator on heat transfer enhancement and friction factor by judging fluid flow in tubes when a uniform heatflux was maintained varied pitch ratios were used. It was observed that the turbulator with the smallest pitch ratio offered highest heat transfer enhancement and thermal performance factor.

Durmus[A. Durmus, 2004] employed conical turbulators with four variations in conical angles viz. 5°, 10°, 15°and 20° for heat transfer enhancement. Apparently, the heat transfer rates as well as friction coefficients increased with increasing turbulatorangles.

Kumar and Saini [Kumar, 2009] presents the performance of a solar air heater duct provided withartificial roughness in the form of a thin circular wire in arc shaped geometry has been analyzed using CFD. The effect of arc shaped geometry on heat transfer coefficient, friction factor, and performance enhancement were investigated covering the range of roughness parameter (from 0.0299 to 0.0426) and working parameter (Re from 6000 to 18000). Different turbulent models have been used for the analysis and their results were compared. Renormalization group (RNG) *k-e* model based results have been found in good agreement and accordingly this model was used to predict heat transfer and friction factor in the duct.

Eiamsa-Ard*et al.* [Eiamsa, 2009] presents the applications of a mathematical model for simulation of the swirling flow in a tube induced by loose – fit twisted tape insertion. Zimparov [Zimparov-2004, Zimparov-2004] investigated a simple mathematical model following the suggestions of Smithberg and Landis has been created to predict the friction factors for the case of a fully developed turbulent flow in a spirally corrugated tube combined with a twisted tape insert.

In this paper, the effect of fitting a new design of ribs (separated ports assembly) in a pipe with internal cooling air flow and constant wall surface temperature will be investigated.

CFD modeling

In this investigation a 3-D numerical simulation of the conjugate heat transfer wasconducted using the CFD code FLUENT 14.5. The CFD modeling involves numerical solutions of the conservation equations for mass, momentum and energy. These three equations are used to model the convective heat transfer process with the following assumptions, (a) steady 3-D fluid flow and heat transfer, (b) incompressible fluid and flow, and (c) physical properties of cooling fluid are temperature dependent. These equations for incompressible flows can be written asfollows:

- mass conservation:

$$\frac{\partial(u_i)}{\partial x_i} = 0 \quad (1)$$

- momentum conservation:

$$\frac{\partial(u_i)(u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}[\mu(\frac{\partial(u_i)}{\partial x_j} + \frac{\partial(u_j)}{\partial x_i}) - \rho(u_i^{/} u_j^{/})] + \rho g_i \quad (2)$$

Energy Conservation

$$\frac{\partial}{\partial x_i}((u_i)(\rho E + p)) = \frac{\partial}{\partial x_i}(K_{eff}\frac{\partial T}{\partial x_i} + u_j(\tau_{ij})_{eff}) \quad (3)$$

Boundary conditions

The boundary zone location is specified in the GAMBIT itself; the inlet, outlet and the wall condition location is specified.

Fluid entry boundary condition

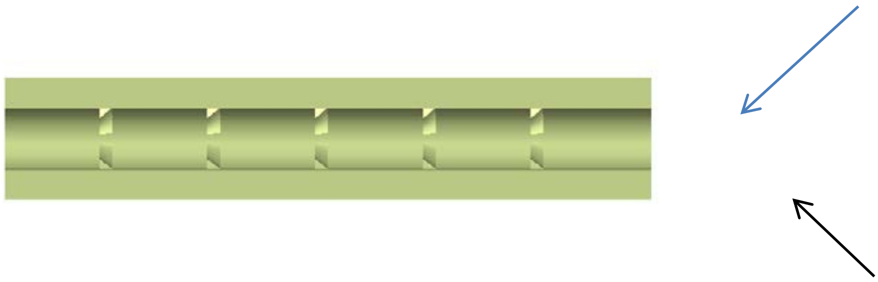
The inlet flow velocity is 10 m/s with constant temperature of 300 K^o.

Wall Boundary Conditions

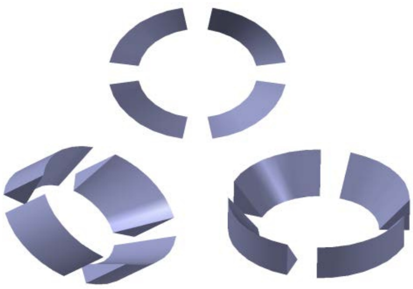
The pipe wall is provided with wall boundary condition, a constant heat flux is provided for plain and ribbed tubes. The outside surface wall temperature is varied from 1000 to 1200 and 1400 K^o.

Geometry of The Test Section

The test section shown in fig. (1) is steel tube with outside diameter of 60 mm and inside diameter of 30 mm at which the coolant air flow in , and having steel separated ports assembly(5x5 mm triangle passage) .The test section was drawn using AUTO CAD 2013.



(A)



(B)

Figure (1) Geometry of the Test Section, A: Tube with ribs , B: Ribs

Results and Discussion

Figures (2) , (3) and (4) show the contours of temperature distribution along the whole test section geometry at constant surface wall temperatures of 1000 , 1200 and 1400 K^O, respectively.

Figures (5), (6) and (7)show the contours of velocity distribution along the whole test section geometry at constant surface wall temperatures of 1000 , 1200 and 1400 K^O, respectively.

Figure (8).shows the temperature distribution along the pipe center line for two cases,one without ribs and the other with ribs at surrounding surface temperature of 1000K^o. It shows that the pipe with ribs has highest outlet air temperature. This means that the pipe with ribs ,has highest surface area resulted in enhancing the heat transfer.

Figure (9).shows the temperature distribution along the pipe center line for two cases,one without ribs and the other with ribs at surrounding surface temperature of 1200K^o.

It shows that the pipe with ribs has highest outlet air temperature. This means that the pipe with ribs ,has highest surface area resulted in enhancing the heat transfer.

Figure (10).shows the temperature distribution along the pipe center line for two cases,one without ribs and the other with ribs at surrounding surface temperature of1400K^o. It shows that the pipe with ribs has highest outlet air temperature. This means that the pipe with ribs ,has highest surface area resulted in enhancing the heat transfer.

Figure (11) shows the velocity distribution along the pipe center line for two cases , one without ribs and the other with ribs at surrounding surface temperature of 1000 K^o. It shows that the pipe with internal ribs having more velocity distribution than the case of plain pipe. This because of the swirls generated from the use of separated ports ribs.

Figure (12) shows the velocity distribution along the pipe center line for two cases , one without ribs and the other with ribs at surrounding surface temperature of 1200 K^o. It shows that the pipe with internal ribs having more velocity distribution than the case of plain pipe. This because of the swirls generated from the use of ribs separated ports ribs.

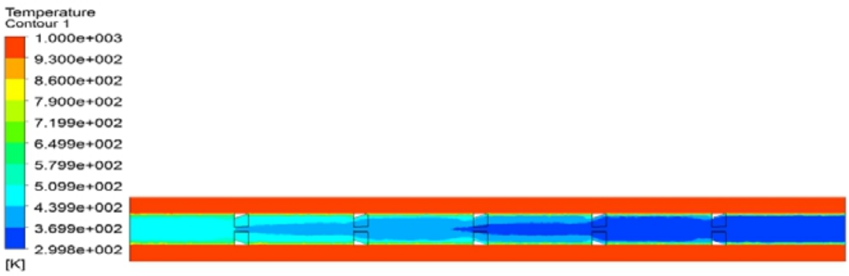
Figure (13) shows the velocity distribution along the pipe center line for two cases , one without ribs and the other with ribs at surrounding surface temperature of 1400 K^o. It shows that the pipe with internal ribs having more velocity distribution than the case of plain pipe. This because of the swirls generated from the use of separated ports ribs.

Conclusion

Numerical simulation has been presented on heat transfer characteristics for the flow of cooling air in heated tube under steady state turbulent flow.The CFD predictions for the case of tube with ribs were compared against the tube without ribs.

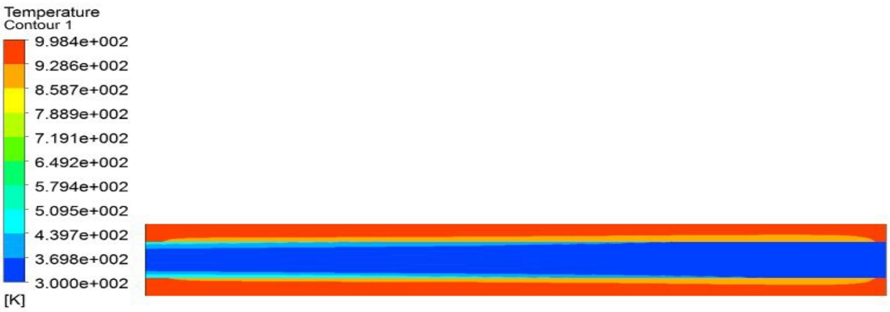
The following conclusions can be drawn from the present study:

1. CFD predictions were shown to reproduce the enhancement in heat transfer for the use of internal ribs, with respect to the plain tube.
2. Based on CFD analysis, higher thermal hydraulic performance were obtained for the tube with ribs than the tube without ribs.
3. tube with ribs gave more velocity distribution than the tube without ribs.
4. The temperature of the plain pipe was found to be approximately un affected for cases of 1000, 1200 and 1400 K^o. While when ribs are used , the effect was to increase the temperature by 370 , 225 , and 75K^o for the cases above, respectively.



Flow Direction ←

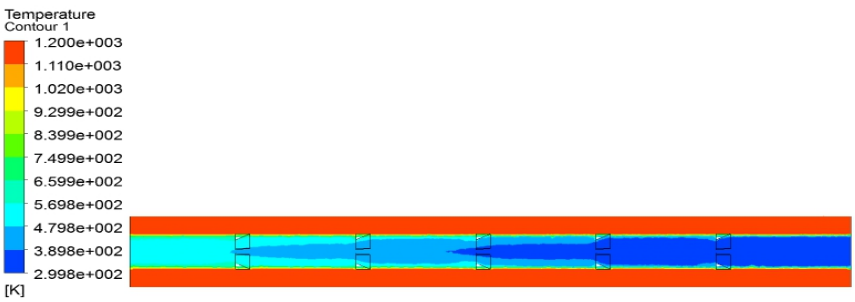
(A)



Flow Direction

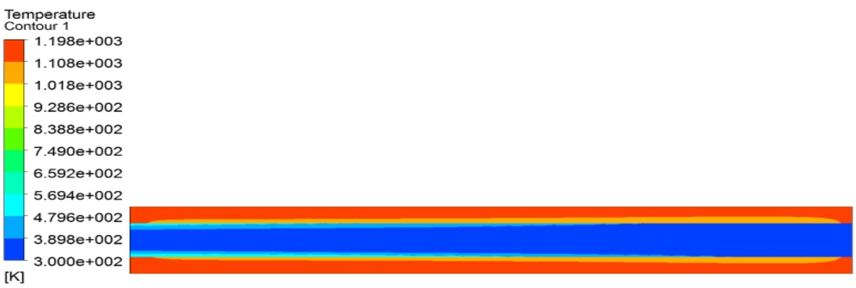
(B)

Fig.(2) Contour of Temperature Distribution at Constant Surface Temperature (1000 k^o)
A: With Ribs
B: Without Ribs



Flow Direction

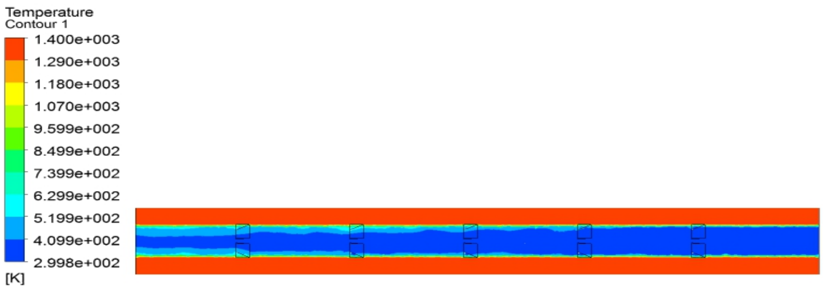
(A)



Flow Direction

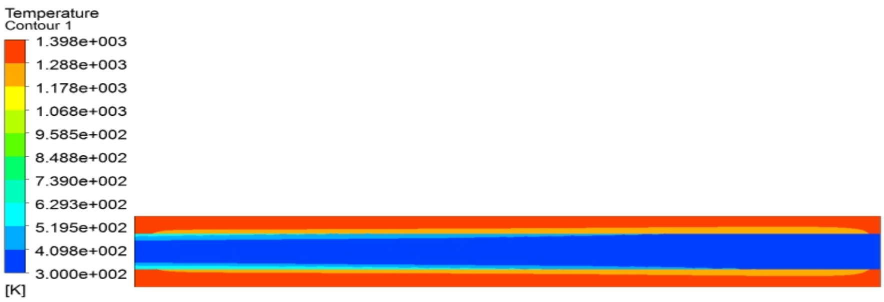
(B)

Fig.(3) Contour of Temperature Distribution at Constant Surface Temperature (1200k^o)
A: With Ribs
B: Without Ribs



Flow Direction

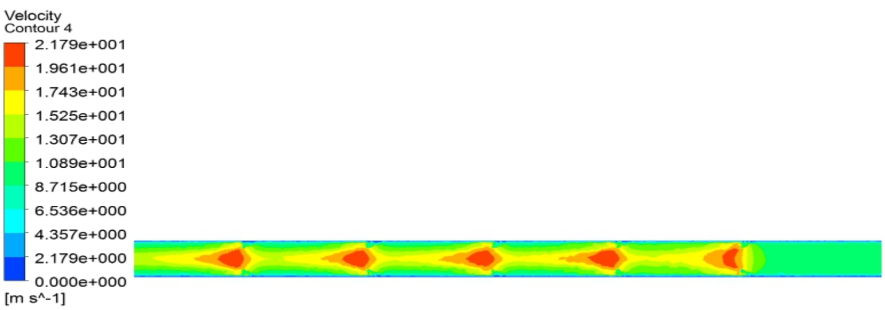
(A)



Flow Direction

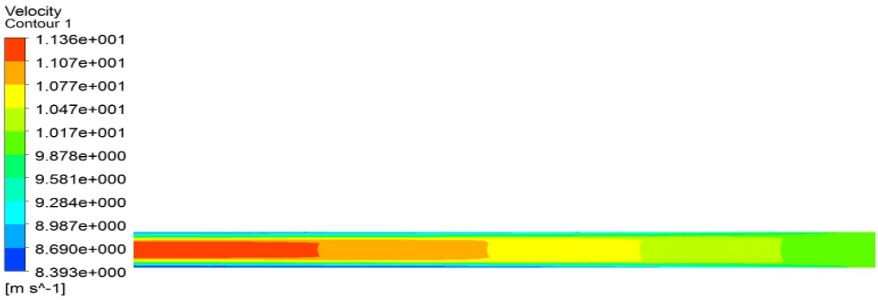
(B)

Fig.(4) Contour of Temperature Distribution at Constant Surface Temperature (1400k^{o})
A: With Ribs
B: Without Ribs



Flow Direction

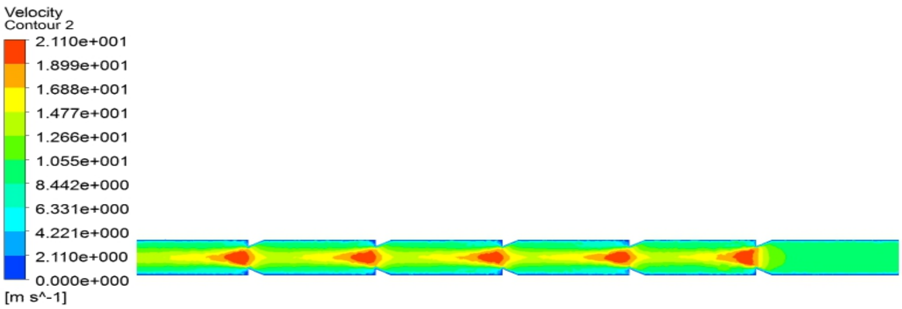
(A)



FlowDirection

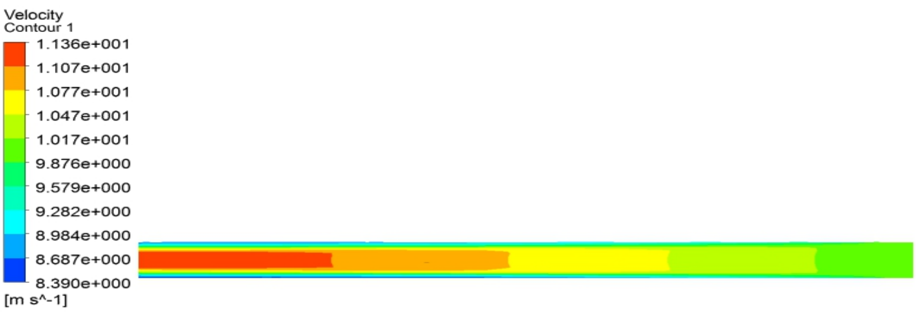
(B)

Fig.(5) Contour of Velocity Distribution at Constant Surface Temperature (1000k^{o})
A: With Ribs
B: Without Ribs



Flow direction

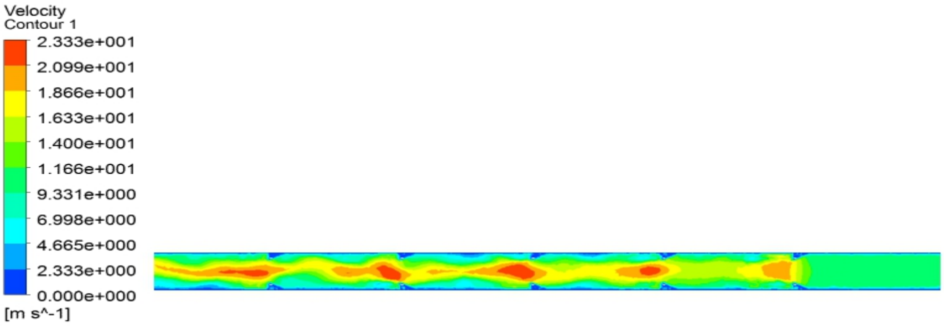
(A)



Flow Direction

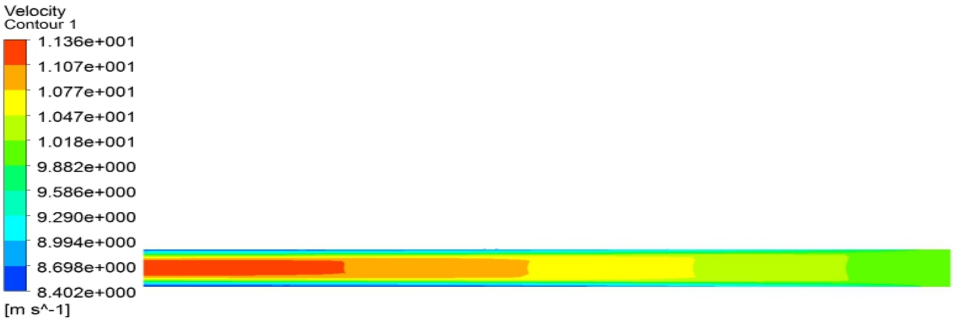
(B)

Fig.(6) Contour of Velocity Distribution at Constant Surface Temperature (1200k^{o})
A: With Ribs
B: Without Ribs



Flow Direction

(A)



Flow Direction

(B)

Fig.(7) Contour of Velocity Distribution at Constant Surface Temperature (1400k^{o})
A: With Ribs
B: Without Ribs

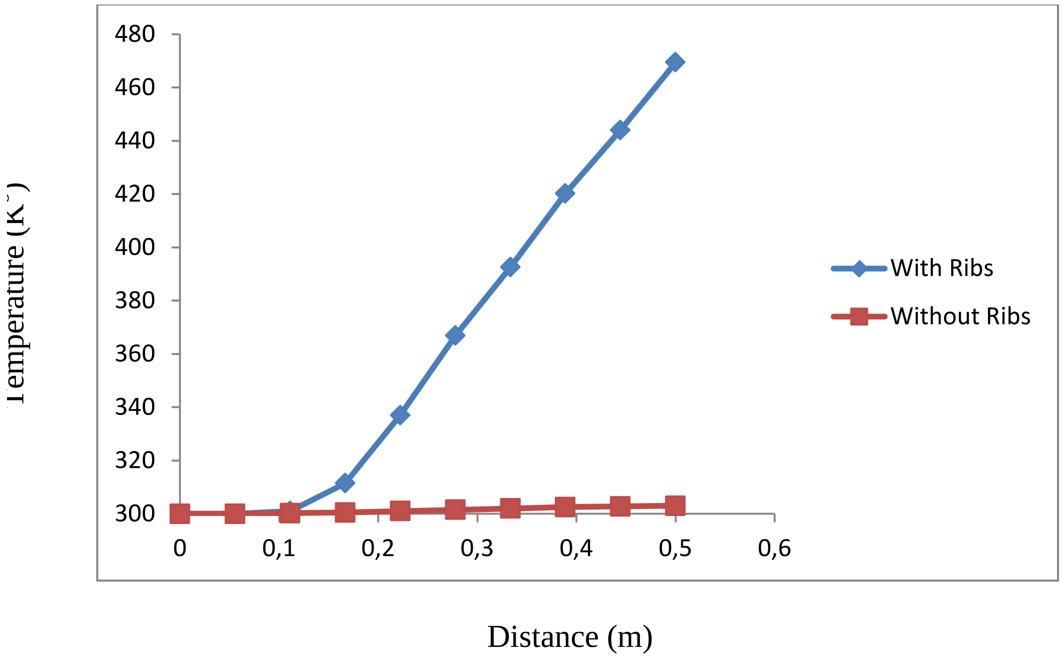


Fig. (8)Variation of Temperature Along the Center line of Tube at Constant Surface Temperature (1000 K^{o})

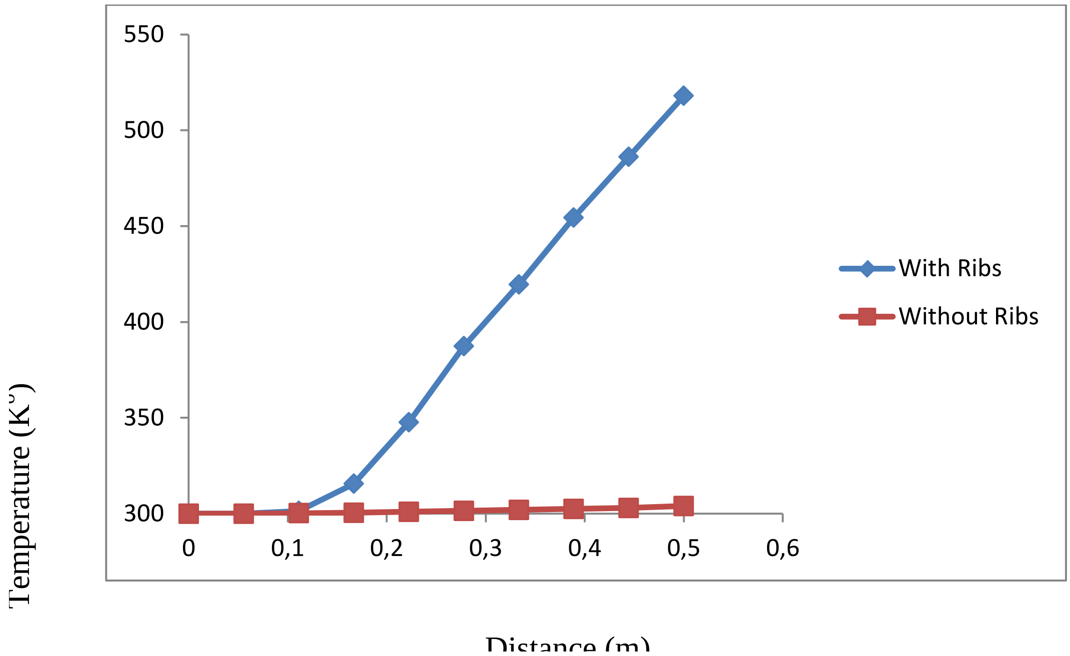


Fig. (9) Variation of Temperature Along the Center line of Tube at Constant Surface Temperature (1200 K^{o})

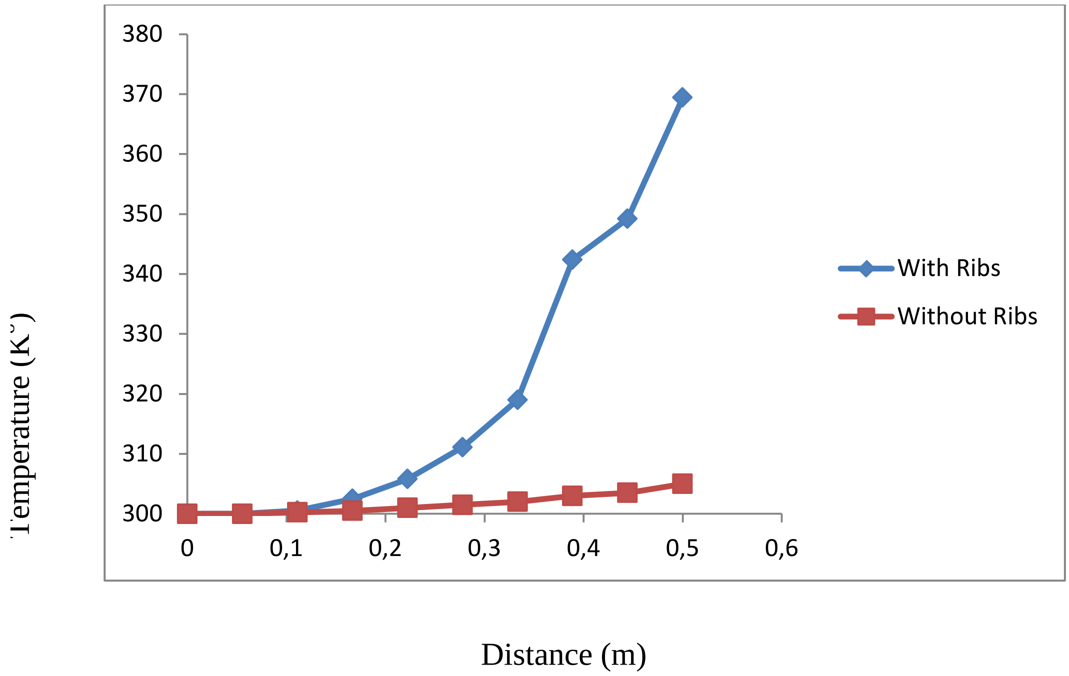


Fig. (10) Variation of Temperature Along the Center line of Tube at Constant Surface Temperature (1400 K^{o})

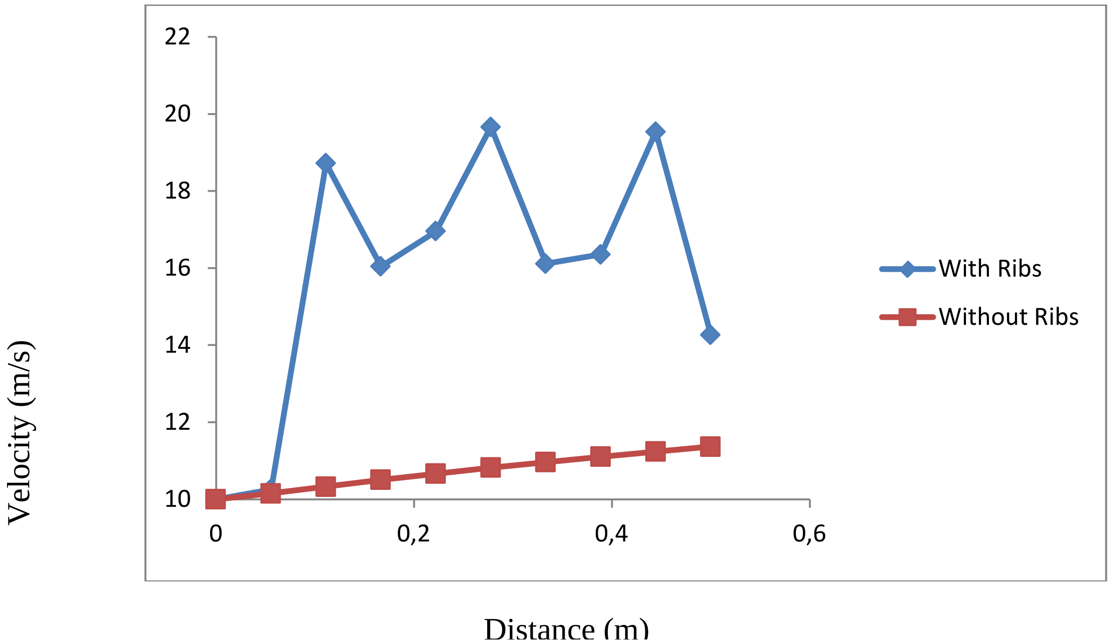


Fig. (11) Variation of Velocity Along the Center line of Tube at Constant Surface Temperature (1000 K^{o})

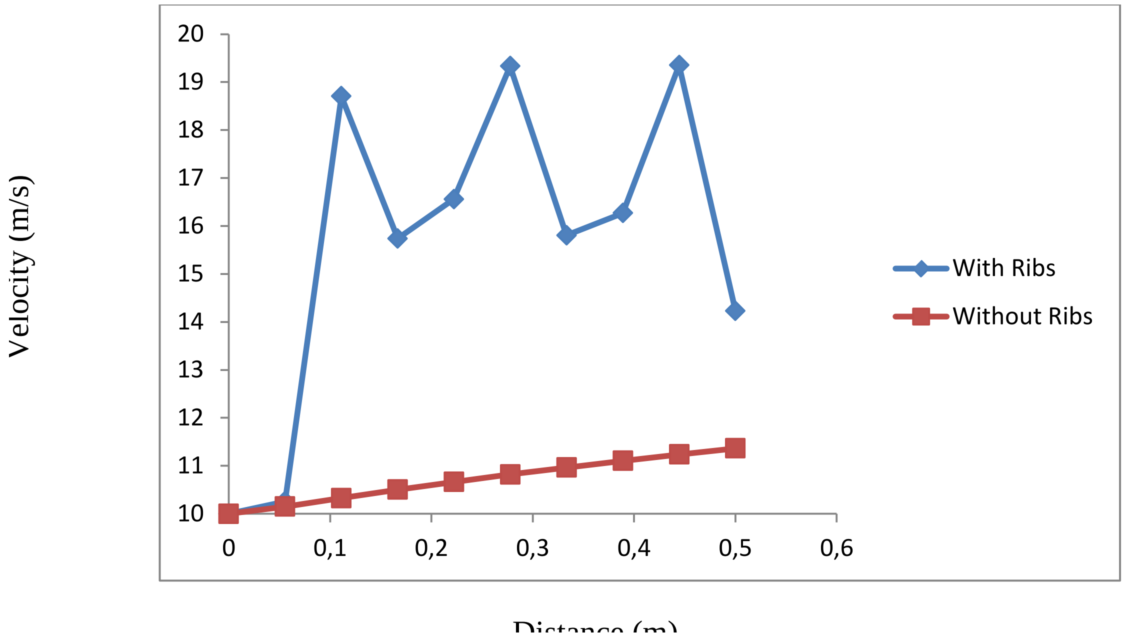


Fig. (12) Variation of Velocity Along the Center line of Tube at Constant Surface Temperature (1200 K°)

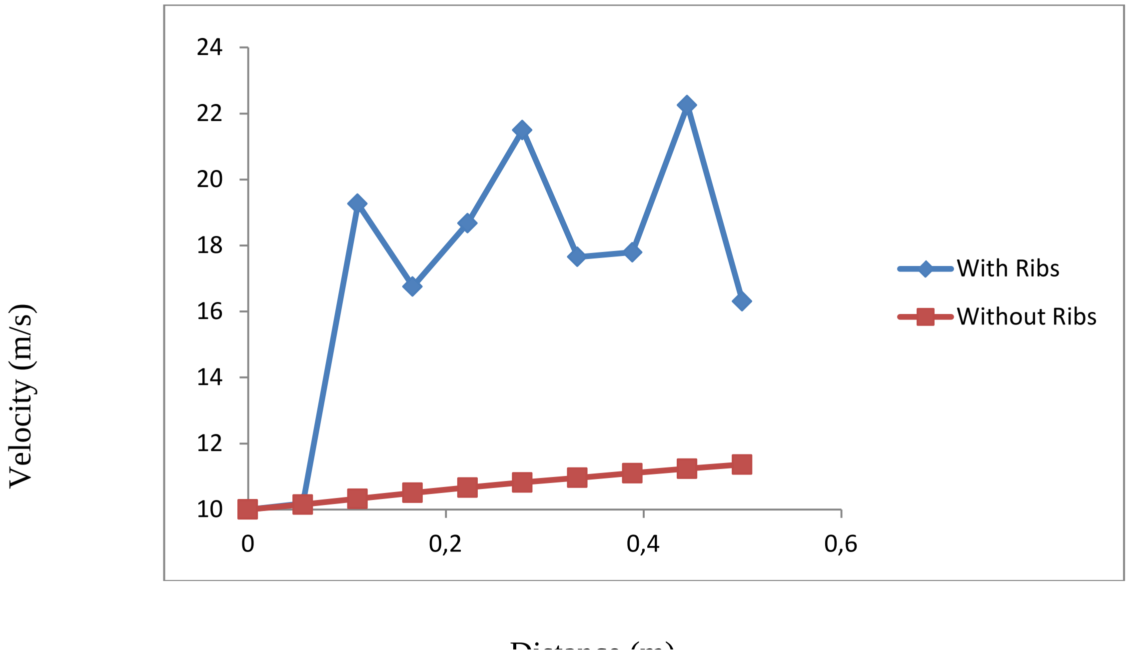


Fig. (13) Variation of Velocity Along the Center line of Tube at Constant Surface Temperature (1400 K°)

References:

A. Durmus,Heat transfer and exergy loss in cut out conical turbulators, Energy Conversion and Management 45 (2004) 785–796.

Buchlin JM. ,Convective heat transfer in a channel with perforated ribs. Int J ThermSci 2002;41:332–40.

Eiamsa-Ard, S., *et al.*, 3D Numerical Simulation of Swirling Flow and Convective Heat Transfer in a CircularTube Induced by Means of Loose Fit Twisted Tapes, Int. J. Communications in Heat Mass Transfer,36 (2009), 9, pp. 947-955.

Jordan SA. ,The turbulent character and pressure loss produced by symmetric ribs in a circular duct. Int J Heat Fluid Flow2003;24:795–806.
K. Yakut, B. Sahin, S. Canbazoglu, Performance and flow-induced vibration characteristics for conical-ring turbulators, Applied Energy 79(2004) 65–76.
Kumar, S., Saini, R. P., CFD Based Performance Analysis of a Solar Air Heater Duct Provided with ArtificialRoughness, Int.J.Renewable Energy, 34 (2009), 5, pp. 1285-1291.
Lu B, Jiang PX. ,Experimental and numerical investigation of convection heat transfer in a rectangular channel with angled ribs. Exp. Therm. Fluid Sci. 2006;30:513–21.
San JY, Huang WC. ,Heat transfer enhancement of transverse ribs in circular tubes with consideration of entrance effect. Int J HeatMass Transfer 2006;49:2965–71.
Tanda G. ,Heat transfer in rectangular channels with transverse and V-shaped broken ribs. Int J Heat Mass Transfer 2004;47:229–43.
Tanda G. ,Heat transfer and pressure drop in a rectangular channel with diamond-shaped elements. Int J Heat Mass Transfer2001;49:3529–41.
VeyselOzceyhan et al.,Heat transfer enhancement in a tube using circular cross sectional rings separated from wall,Science Direct, Applied Energy 85 (2008) 988–1001.
Zimparov, V., Prediction of Friction Factors and Heat Transfer Coefficient for Turbulent Flow of Corrugated Tubes Combined with Twisted Tape Inserts, Part 1 – Friction Factors, Int. J. Heat and Mass Transfer, 47 (2004), 3, pp. 589-599.
Zimparov, V., Prediction of Friction Factors and Heat Transfer Coefficient for Turbulent Flow of Corrugated Tubes Combined with Twisted Tape Inserts, Part 2 – Heat Transfer Coefficient, Int. J. of Heat and Mass Transfer, 47 (2004), 2, pp. 385-393.