

Enhancement of Temperature Distribution and Heat Transfer Coefficient of Ribbed Tube by Simulation

Rahul Kunar

M Tech Scholar

Technocrats Institute of Technology & Science

Bhopal (M.P), India

rahulxp151@gmail.com

Dr Sukul Lomash

Professor

Technocrats Institute of Technology & Science

Bhopal (M.P), India

lomash15@gmail.com

Abstract: The heat transfer from surface may in general be enhanced by increasing the heat transfer coefficient between a surface and its surrounding or by increasing heat transfer area of the surface or by both. The main objective of the study and calculate the total heat transfer coefficient. Improve the heat transfer rate by using ANSYS CFD. During the CFD calculations of the flow in internally ribbed tubes. And calculated the temperature distribution and pressure inside the tube by using ansys. The model was created using CatiaV5 and meshed with Ansys, and the flow analysis is done with Ansys 19.2. The results showing that the heat transfer is increased. The enthalpy and temperature increase with flow is advancing when compare with normal boiler tube. In this study the total heat transfer rate of the pipe increase with the increase the rib height. Total heat transfer rate increase up to 7.7kw. The study show that the improvement in furnace heat transfer can be achieved by changing the internal rib design0.

Keywords: catiaV5, computational fluid dynamic, heat exchanger, Rib tube.

I. INTRODUCTION

The heat transfer from surface may in general be enhanced by increasing the heat transfer coefficient between a surface and its surrounding or by increasing heat transfer area of the surface or by both. Ribs used in cooling channel and heat exchanger channel are most commonly used passive heat transfer techniques. So that the work related to fluid flow and heat transfer in ribbed channel is go so far. This heat transfer augmentation technique are applied to many industrial application such as shell and tube type heat exchanger , electronic cooling devices, thermal regenerators, and internal cooling system of gas turbine. Each rib on downstream separates the flow, recirculate, and impinges on channel wall and these are the main reason for heat transfer enhancement in such channel. The use of rib in heat exchanger not only increase the heat

transfer rate but also substantial the pressure loss [1][3]. The rib arrangement and geometry resulting in different heat transfer distribution by altering the flow field. Therefore by making the modification in rib geometry we can increase the heat transfer rate but at the same time we need to consider the pressure drop also because it increases significantly.

The study of improved heat transfer performance is referred to as heat transfer augmentation, enhancement. The common thermo hydraulic goals are to reduce the size of a heat exchanger required for a specified heat duty and to upgrade the capacity of an existing heat exchanger, also to reduce the pumping power. Heat transfer enhancement techniques are commonly used in areas such as process industries heating and cooling in evaporators, thermal power plants, air conditioning equipment's, refrigerators and radiators for space vehicles, automobiles.

II. LITERATURE REVIEW

Sławomir Grądziel et al. [4] This paper puts forward modelling thermal and flow phenomena in internally rifled tubes. The proposed model is a distributed parameter model based on solving balance equations describing the principles of the mass, momentum and energy conservation. The model enables an analysis of transient-state processes. The aim of the calculations is, among others, to find the distribution of the fluid enthalpy, mass flow and pressure in internally rifled tubes and to determine the heat transfer coefficient.

Alireza Taklifi et al. [5] in this paper presented are the effect of various inclination angles on heat transfer of water at subcritical and supercritical operating pressures is investigated experimentally. These operating conditions covered subcritical, near critical and supercritical water flows and also refers to low

mass flux conditions. The inclination angles were 5, 20, 30, 45 and 90 (vertical) degrees respecting to horizontal plane. The heat flux was kept constant along the test tube by controlling of electric heating. As a result the inner wall temperature and convective heat transfer coefficient variations with respect to heated length and bulk enthalpy of fluid were considered in order to study the heat transfer characteristics of various flows at different inclinations.

Aadel A. R. Al Kumait et al. [6] The study is accomplished by using the finite volume method, and its objective involves finding a low friction factor and high heat transfer enhancement in the presence of TiO₂/water nanofluids. The helical ribbed tubes with a 5.89 mm pitch distance gave higher turbulent kinetic energy due to a stronger swirl intensity, resulting in a thinner thermal boundary layer and a higher Nusselt number with uniform distribution.

Tao Zhang et al. [7] in this paper presented are the turbulent supercritical water flow characteristics within different grooves are obtained using a validated low-Reynolds number $k-\epsilon$ turbulence model. The effects of groove angle, groove depth, groove pitch-to-depth ratio, and thermophysical properties on turbulent flow and heat transfer of supercritical water are discussed. The results show that a groove angle $\gamma = 120^\circ$ presents the best heat transfer performance among the three groove angles. The lower groove depth and higher groove pitch-to-depth ratio suppress the enhancement of heat transfer.

III. OBJECTIVE

- The main objective of the study and calculate the total heat transfer coefficient.
- Improve the heat transfer rate by using ANSYS CFD.
- During the CFD calculations of the flow in internally ribbed tubes.
- Calculated the temperature distribution and pressure inside the tube by using ansys.

IV. METHODOLOGY

A. CFD Analysis on Pipe

This paper presents the calculation results of thermal and flow phenomena occurring in internally rifled tubes for a medium with supercritical parameters. The results were obtained by means of a numerical analysis and CFD. The CFD modelling was carried out using the Ansys Fluent software package

B. Mathematical model

Equations describing the principles of the mass, momentum and energy conservation, respectively, will be solved to determine the distribution of the mass flow (mass flux) and the medium

enthalpy (temperature) along the tube. The base balance equations are expressed in [4]. After appropriate reductions and transformations, mass, momentum and energy equations are written so that space derivatives should be obtained on the left side of the equations, whereas time derivatives occurring on the right side are replaced with backward difference quotients. This system of ordinary differential equations is solved using the Runge-Kutta method. After the changes described above are introduced, the energy balance equation takes the following form:

$$\frac{dh_j^*}{dz} = \frac{q_j^{*-\Delta z}}{\dot{m}_j^{*-\Delta z}} \left(\frac{h_j^* - h_j^{*-\Delta z}}{\Delta z} + \frac{4\alpha_{w,j}^{*-\Delta z} (T_{w,j}^{*-\Delta z} - T_j^{*-\Delta z})}{d_o \rho_j^{*-\Delta z}} \right) \quad (1)$$

The fluid density is found as a function of enthalpy and pressure:

$$\rho_j^* = f(h_j^*, p_j^*) \quad (2)$$

Solving the mass and momentum conservation equations, the following is obtained, respectively:

$$\frac{d\dot{m}_j^*}{dz} = -A \frac{\rho_j^* - \rho_j^{*-\Delta z}}{\Delta z} \quad (3)$$

$$\frac{d}{dz} \left(\frac{(\dot{m}^2)_j^*}{A^2 \rho_j^* + p_j^*} \right) = \frac{1}{A} \frac{(\dot{m}^2)_j^* - (\dot{m}^2)_j^{*-\Delta z}}{\Delta z} - \frac{dp_j^*}{dz} - \rho_j^* g \sin \phi \quad (4)$$

The fluid temperature history is found as a function of enthalpy and pressure:

$$T_j^* = f(h_j^*, p_j^*) \quad (5)$$

Subscript "j" in Equations (1-5) denotes the number of analysed cross-sections and varies from $j=1,2,\dots,M$. All thermo physical properties of the fluid and of the wall, as well as the heat transfer coefficient, are determined on-line. The time- and space-dependent distribution of the tube wall temperature is also calculated using the transient thermal conductivity equation and assuming uniform heating of the tube with a heat flux with density q [5]:

$$c_w \rho_w \frac{\partial \theta}{\partial \tau} = \frac{1}{r} \frac{\partial}{\partial r} \left[r k_w \frac{\partial \theta}{\partial r} \right] \quad (6)$$

under the following boundary conditions:

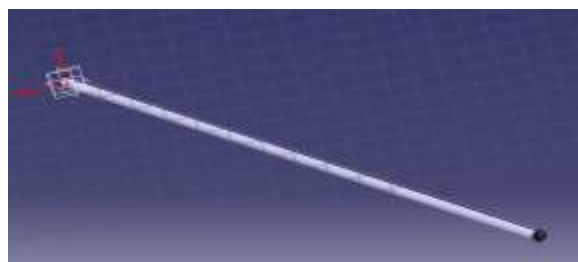
$$k_w \frac{\partial \theta}{\partial r} \bigg|_{r=r_1} = \dot{q} \quad (7)$$

$$k_w \frac{\partial \theta}{\partial r} \bigg|_{r=r_1} = \alpha (\theta_w - t) = \alpha (\theta - t) \quad (8)$$

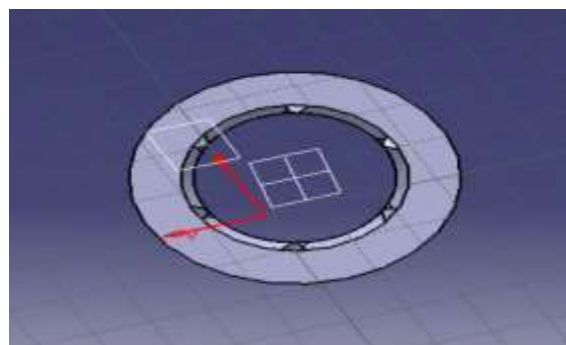


c. Preparation of model

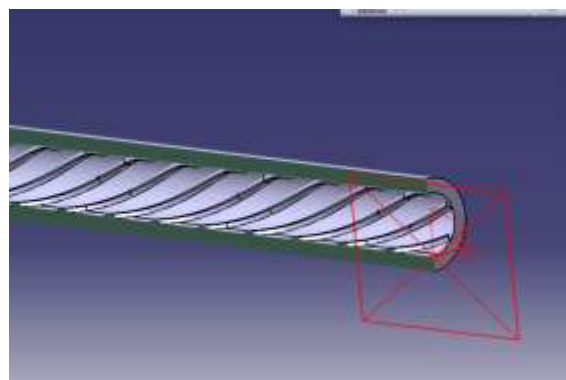
A CAD model is prepared in catia



(a) Aisomatrical pipe



(b) Front veiw of pipe



(c)

Fig. 1 (a)(b)(c) Catia model design of PIPE

D. Steps of Working

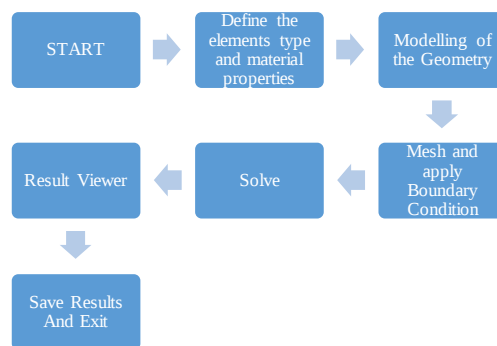


Fig. 2: Setup of working

1. Import Model in ANSYS

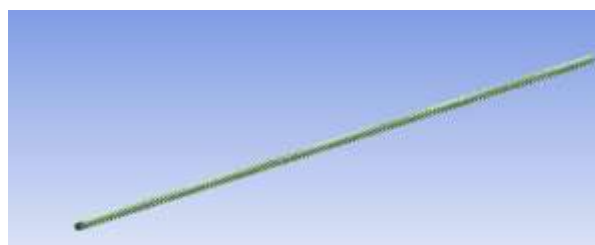


Fig. 3 Import domain Extract model in ANSYS

Case-1:-base model

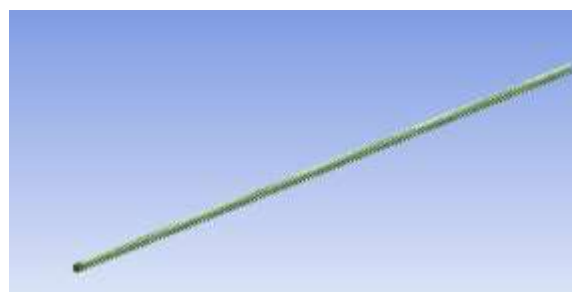


Fig. 4 Case-1 model

Dimension	Value
outer diameter, d_o	50.8 mm
inner diameter (without ribs), d_i	34.9 mm
minimum diameter, d_{min}	32.9 mm
wall thickness, g	7.95 mm
rib height, h	1 mm
pitch, p	30 mm
rib at the base, a	5 mm
rib average width, b	4.5 mm
angle between rib arms, α	45°
rib helical angle, β	30°
number of ribs, N	6

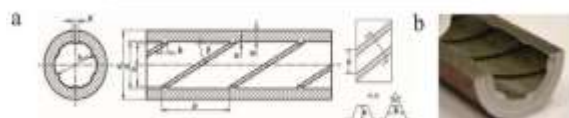


Table 1 Change the value of rib height

CASES	Rib Height(h)(mm)	Rib at average width)(mm)
CASE-1	1	4.5
CASE-2	2	4.5
CASE-3	3	4.5
CASE-4	4	4.5

4. MESHING

Case-1

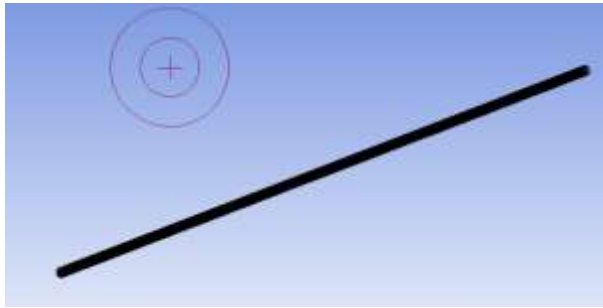


Fig. 5: Meshing: Total No. of Nodes: 39398 & Total No. elements: 166207

CASES	Nodes	Elements
CASE-1	53331	221273
CASE-2	42410	184012
CASE-3	41304	177337
CASE-4	39398	166207

5. DEFINING MATERIAL PROPERTIES

➤ Water properties

Table 4.2 Water Properties

Properties	Values
Density (kg/m ³)	998
Specific heat(j/kg-k)	4182
Thermal conductivity(w/m-k)	0.6

➤ copper properties

Table 4.3 copper properties

Properties	Values
------------	--------

Density (kg/m ³)	8978
Specific heat(j/kg-k)	381
Thermal conductivity(w/m-k)	387.6

Boundary Condition

1. Define Name selection

Define inlet and outlet for CFD analysis.



Fig. 6 Define boundary conditions

Boundary conditions	Values
Inlet velocity	313 °C
Outlet	Pressure outlet

Results Case-1 1mm rib height Result

Velocity contours:- The blue color shows minimum velocity area. And the yellow color show maximum velocity area.

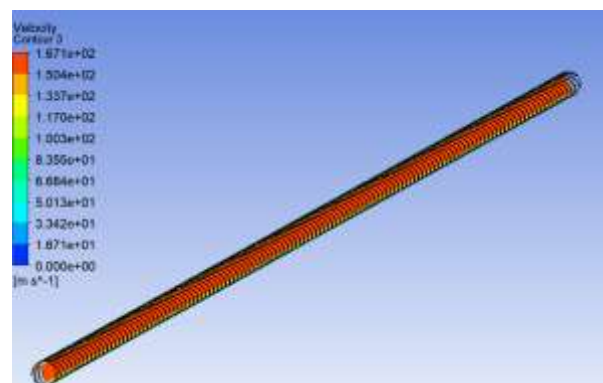


Fig. 7 Velocity contours case 1



Fig. 8 Temperature contours pipe along longitudinal for case-1

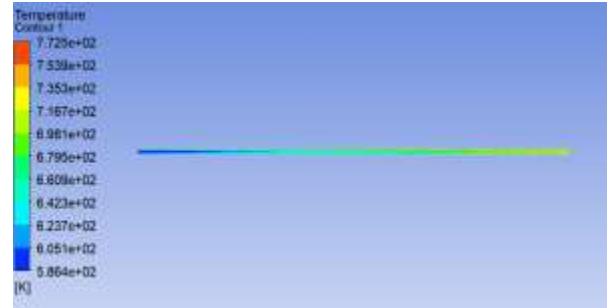


Fig. 11 Temperature contours pipe along longitudinal for case-2

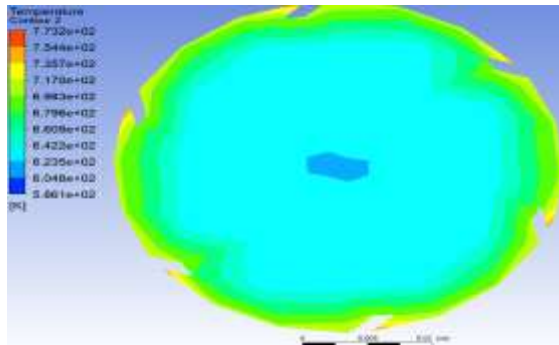


Fig. 9 Temperature contours of pipe front view case-1

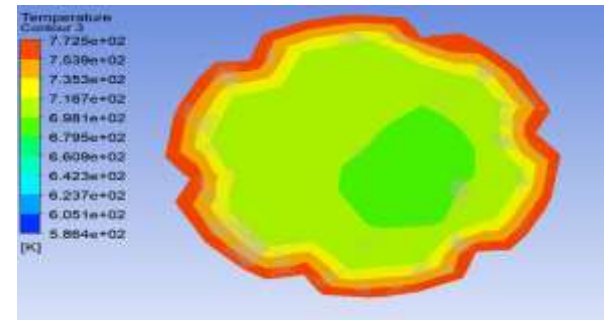


Fig. 12 Temperature contours of pipe from front view case-2

Temperature contours:-maximum temperature inside the tube was 773K and minimum temperature 586k.the light blue color show the minimum temperature region and red color show maximum temperature region.

Velocity stream line:- velocity streamline shows the flow of air inside the tube

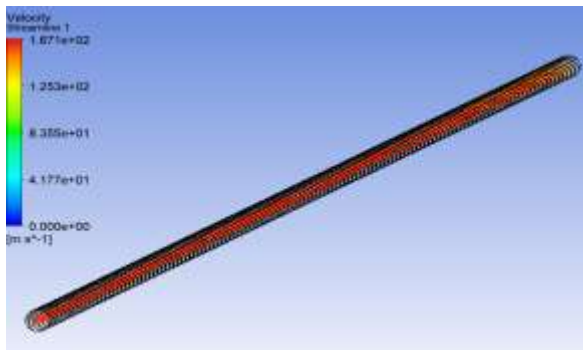


Fig. 10 Velocity stream line for case-1

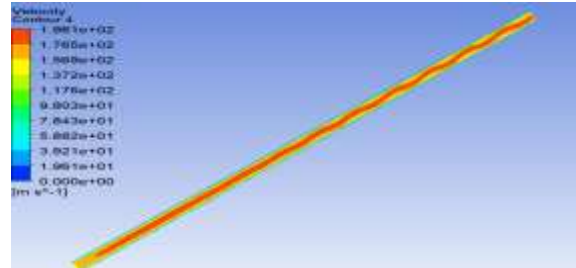


Fig. 13 Velocity contours for case-2

Streamline: - velocity streamline shows the flow of air inside the tube. The flow of air inside the tube shown by stream line.

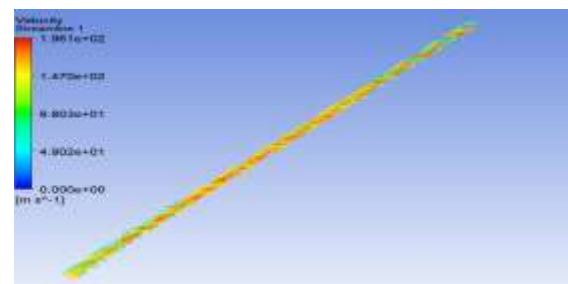


Fig. 14 Velocity stream line for case-2

Case-2 2mm rib height Result

Temperature contours:-maximum temperature inside the collector was 772K and minimum temperature 586k.the light blue color show the minimum temperature region and red color show maximum temperature region.

Pressure contours:- The pressure inside the tube depended on heat transfer rate and air flow. The red color in fig show the maximum pressure zone and blue color minimum pressure zone.

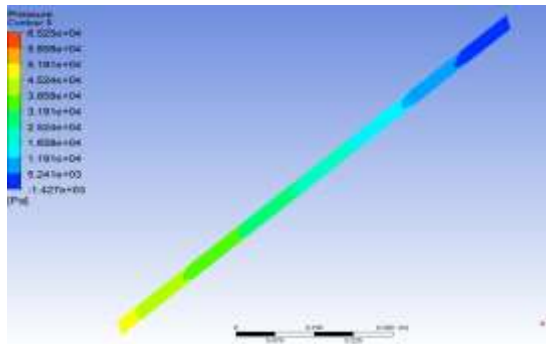


Fig. 15 pressure contours for case-2

Case-3 3mm rib height Result

Temperature contours:-maximum temperature inside the collector was 772K and minimum temperature 579k.the light blue color show the minimum temperature region and red color show maximum temperature region.

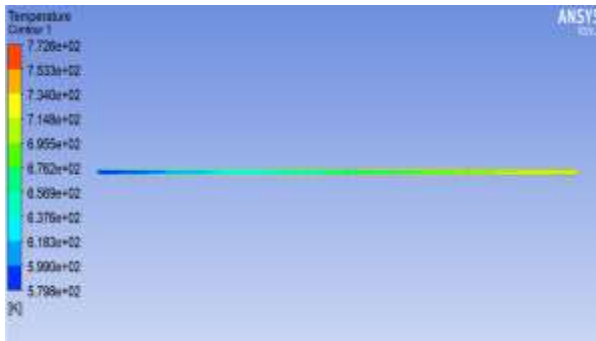


Fig. 16 Temperature contours pipe along longitudinal for case -3

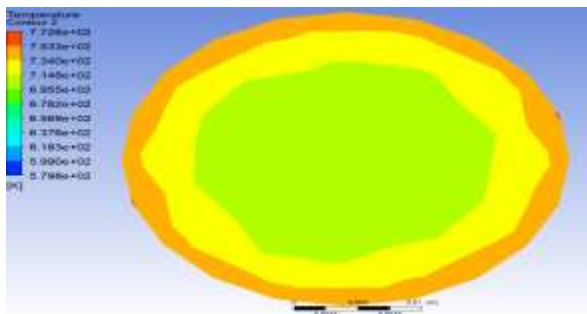


Fig. 17 Temperature contours of pipe from front view case 3

Velocity contours:- The light blue color show the minimum velocity region and red color show maximum velocity region.

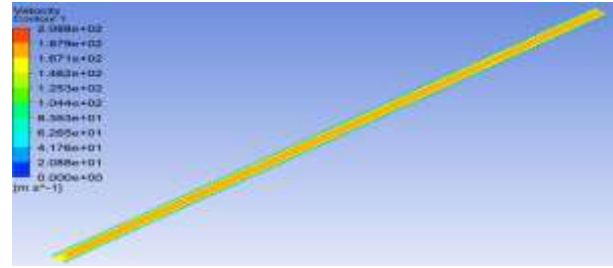


Fig. 18 Velocity contours for case-3

Streamline: - velocity streamline shows the flow of air inside the Tube line. The flow pattern of the air dependent of the air velocity and air domain.



Fig. 19 Velocity stream line for case-3

Pressure contours:-

The pressure inside the tube depended on heat transfer rate and air flow. The red color in fig show the maximum pressure zone and blue color minimum pressure zone.

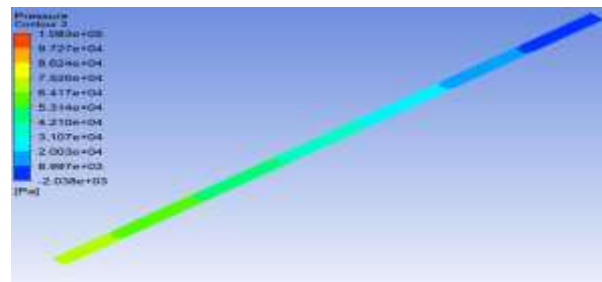


Fig. 20 Pressure contours for case-3

CASE-4 4mm rib height Result



Fig. 21 temperature contours case 4

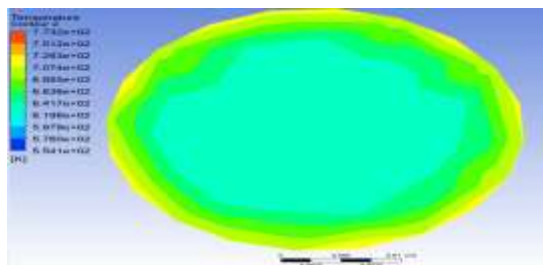


Fig. 22 Outlet temperature contours

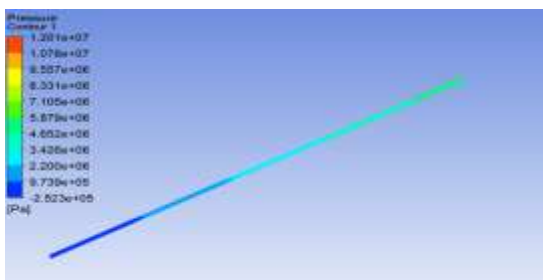


Fig. 23 Pressure contours for case-4

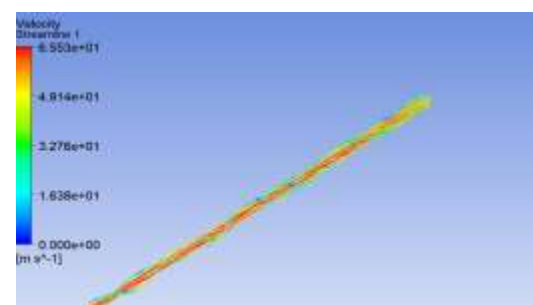


Fig. 24 Velocity stream lines for case-4

V. RESULTS

1. LMTD

The logarithmic mean temperature difference can be calculated simply using its definition:

$$LMTD = \frac{\Delta T_1 - \Delta T_2}{\ln \left(\frac{\Delta T_1}{\Delta T_2} \right)}$$

$$Q = \dot{m}_{oil} \cdot c_{oil} \cdot \Delta T$$

Where

M:-mass flow rate 0.03kg/s

Cp:- specific heat of water

CASES	Inlet temperature	Outlet temperature	LMTD	Q(kw)
-------	-------------------	--------------------	------	-------

CASE-1	313	331	322	2.25
CASE-2	313	345	329	4.01
CASE-3	313	360	336	5.9
CASE-4	313	375	344	7.7

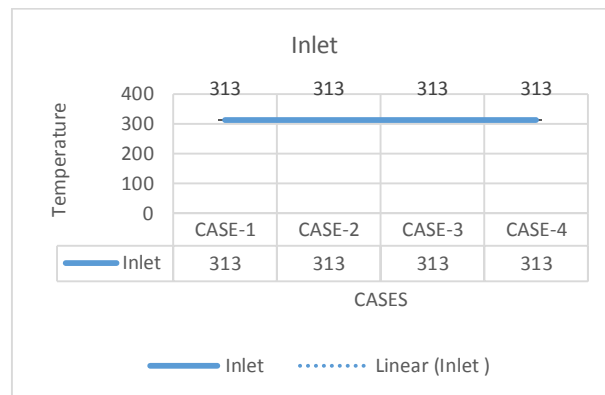


Fig. 25 Inlet Temperature graph

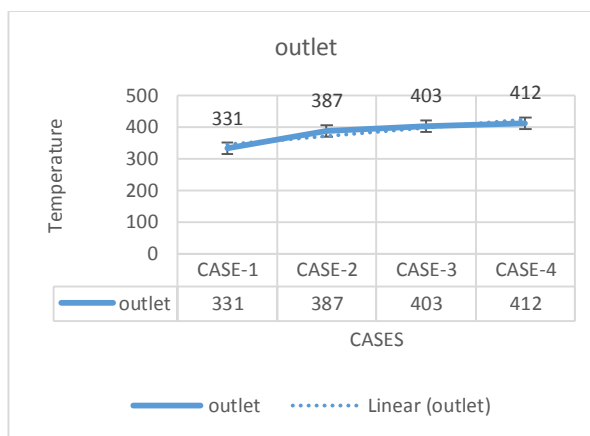


Fig. 26 Outlet Temperature graph

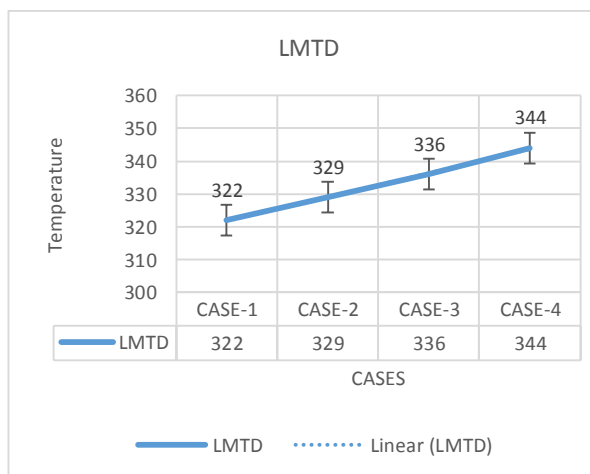


Fig. 27 Logarithmic mean temperature difference graph

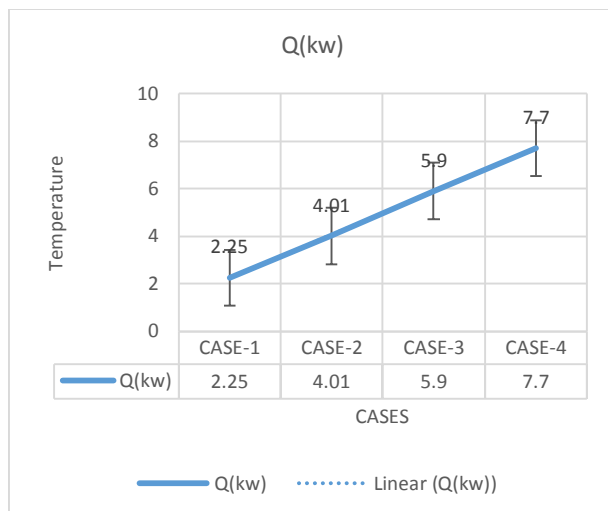


Fig. 28 General heat transfer graph

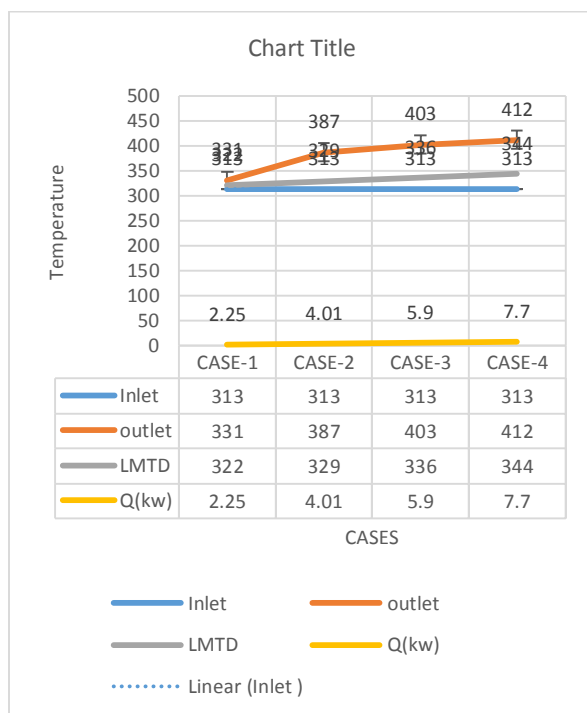


Fig. 29 Temperature and LMTD graph

VI. CONCLUSION

This paper presents the calculation results of thermal and flow phenomena occurring in internally rifled tubes for a medium with supercritical parameters. The results were obtained by means of a numerical analysis and CFD.

The CFD modelling was carried out using the Ansys Fluent software package. The model was created using CatiaV5 and meshed with Ansys, and the flow analysis is done with Ansys 19.2. The results showing that the heat transfer is increased. The

enthalpy and temperature increase with flow is advancing when compare with normal boiler tube. In this study the total heat transfer rate of the pipe increase with the increase the rib height. Total heat transfer rate increases up to 7.7kw. The study shows that the improvement in furnace heat transfer can be achieved by changing the internal rib design.

REFERENCES

- [1] Miss. Ashwini Vasant Thakare, Dr. J. A. Hole "Experimental Validation of Heat Transfer Enhancement in plate Heat Exchanger with Non-Conventional Shapes of Rib" International Journal of Science, Engineering and Technology Research (IJSETR), Volume 5, Issue 3, March 2016.
- [2] Miss. Ashwini Vasant Thakare, Dr. Hole J. A. "Review of Heat Transfer Enhancement in Plate Heat Exchanger with Non-Conventional Shapes of Rib" International Journal of Engineering Research & Technology (IJERT) ISSN: 2278-0181 Vol. 4 Issue 02, February-2015
- [3] Mi-Ae Moon, Min-Jung Park, Kwang-Yong Kim, Evaluation of heat transfer performances of various rib shapes, International Journal of Heat and Mass Transfer, pp. 275–284, Jan 2014.
- [4] Sławomir Grądziel, Karol Majewski "Simulation of Temperature Distribution and Heat Transfer Coefficient in Internally Ribbed Tubes" Procedia Engineering, vol. 157, pp. 44-49 · August 2016.
- [5] Alireza Taklifi, Mohammad Ali Akhavan-Behabadi "Experimental investigation of inclination effect on subcritical and supercritical water flows heat transfer in an internally ribbed tube" Heat and Mass Transfer volume 53, pages635–647, 2017.
- [6] Aadel A. R. Al Kumait, Thamer K. Ibrahim "Experimental and numerical study of forced convection heat transfer in different internally ribbed tubes configuration using TiO₂ nanofluid" Volume48, Issue5 05 April 2019.
- [7] Tao Zhang, Defu Che "Numerical investigation on heat transfer of supercritical water in a roughened tube" Journal Numerical Heat Transfer, Part A: Applications An International Journal of Computation and Methodology Volume 69, Issue 6, 2016.
- [8] Zhanwei Liu, Yanwei Yue "Numerical Analysis of Turbulent Flow and Heat Transfer in Internally Finned Tubes" Energy Res., 18 July 2019.
- [9] Shivasheesh Kaushik, Vimal Singh Chamyal "Study and Analysis on Boiler Tubes for Performance Enhancement with Varying Corrugated Tube Shapes" International Journal on Emerging Technologies (Special Issue NCETST-2017) 8(1): 727-732, 2017.
- [10] Piotr Duda, Dariusz Rzaśa "A Method for Optimum Heating and Cooling Boiler Components of a Complex Shape" Journal of Thermal Science 24(4):364-369 · June 2015.