# Heat Transfer Enhancement in a Tube Fitted with a Circular Ring As Vortex Generator: A CFD Analysis

[1]Nilesh R. Singh, [2]Pramod G. Rahate, [3]Avinash A. Datarkar, [4]Nilesh N. Raut
[5]Sunilkumar R. Patil, [6]Kiran R. Jadhav
*[1-5]Mechanical Engineering Department, Bharati Vidyapeeth's College of Engineering Lavale, Pune 412115*
*[6]Engineering Science Department, Bharati Vidyapeeth's College of Engineering Lavale, Pune 412115*

*Abstract*— **The rate of heat transfer, which is dependent on a number of factors like Reynolds number, fluid properties, and surface features, can be increased by using various techniques. In this research, an experimental analysis was conducted on a smooth circular tube and subsequently on a tube fitted with inserts. During the experiment Reynolds number was varied from range of 15,000 to 25,000, while a maintaining heat flux constant along the tube circumference. The tube, constructed from steel, includes a centrally placed rod supporting symmetrically arranged inserts. Computational Fluid Dynamics (CFD) analysis is carried out to evaluate heat transfer enhancement using inserts. The experimental findings on heat transfer enhancement and pressure drop in a tube without inserts are validated against numerical results obtained from a Fluent solver. Nusselt number and friction factor are calculated based on both experimental and simulation data. The CFD model suggests an optimal insert configuration with a pitch of 3 cm and an angle of attack of 30°.**

***Keywords*— CFD, Heat Transfer Enhancement, Inserts, Circular Ring, Ring Inserts.**

NOMENCLATURE:-

*$A_s$–Surface area, $m^2$*
*$C_{pa}$- Specific heat air, kJ/kg K*
*$D_i$ - Internal Diameter of the test tube, m*
*$D_o$ - Extarnal Diameter of the test tube, m*
*L - Length of test tube, m*
*f - Friction factor of tube fitted with modified vortex generator.*
*$f_s$ - Friction factor of plane tube*
*$h_i$ - Average heat transfer coefficient, ($w/m^2K$)*
*K - Thermal conductivity of air, W/m K*
*P - Pitch length of the of inserts, m*
*m - Mass flow rate, kg/ s*
*Nu - Nusselt number*
*$Nu_c$ - Nusselt number on the basis of equal pumping power (Without vortex generator)*
*$Nu_s$ - nusselt number on the basis of equal Reynolds number (Without vortex generator)*
*Q - Volume flow rate of the hot air, m /s*

## 1. Introduction:-

The efficiency of heat exchangers is often constrained by the thermal performance of gases, which generally exhibit lower heat transfer coefficients compared to liquids or multiphase flows. To optimize energy performance while minimizing volume and manufacturing costs, high-performance heat exchangers are in demand. Enhancing heat transfer can be achieved by increasing the heat transfer coefficient, augmenting the heat transfer surface area per unit volume, or both.

Tube inserts are widely used for heat transfer enhancement as they are easy to install, do not require material deformation, and provide improved thermal performance. The use of twisted tape and wire coil inserts is common due to their effectiveness in increasing heat transfer.

### 1.1 HEAT TRANSFER ENHANCEMENT:-

Generally speaking, the hydraulic diameter of the flow passage is decreased by inserts of various types that are inserted to increase the rate of heat transfer. The main reasons why inserts like ribs, dimples, wire coils, and twisted tapes improve heat transmission in a tube flow include secondary flow, flow blocking, and flow partitioning. As a result of a decreased free flow area, flow obstruction raises the pressure drop and intensifies viscous effects. Additionally, blockage speeds up the flow and, in certain cases, creates a sizable secondary flow. Because secondary flow produces swirl and the consequent fluid mixing enhances the temperature gradient, which eventually results in a high heat transfer coefficient, secondary flow also improves the thermal contact between the fluid and the surface.

### 1.2. Enhancement Technique:-

The enhancement techniques can be broadly classified as

I. Passive technique
II. Active technique

Various enhancing methods are often divided into passive and active methods. Table 1 provides a list of the different techniques or tools that fall under each of these two groups. The main difference is that passive procedures don't need direct external power input like active methods do. Usually, they add an insert, material, or other device, or they alter the flow channel's surface or geometry. By disrupting or changing the current flow behavior, these passive schemes encourage higher heat transfer coefficients.

1.2.1 The various forms Heat Transfer Enhancement devices are:

a) Tube inserts: These augmentation methods have the advantage of being able to be put in an existing smooth tube heat exchanger and preserving the smooth tube's mechanical strength.
b) Mesh inserts:-
Due to the higher per unit volume of mesh inserts, the temperature decreases considerably along the direction of flow, and the porous segments release a greater amount of energy, primarily upstream.

1.3 Optimization:

Heat transfer enhancement improves thermal performance; it is often associated with an increased pressure drop, requiring higher pumping power. The optimization process aims to balance heat transfer augmentation and energy consumption. By adjusting Reynolds number, heat transfer coefficient, and friction factor, an optimal system design can be achieved.

2. Literature:-

Passive, active, or a combination of passive and active heat transfer augmentation techniques are frequently employed in process industries, evaporator heating and cooling, thermal power plants, air conditioning units, refrigerators, space vehicle radiators, automobiles.

Compared to active approaches, passive techniques which use inserts in the flow path to increase the rate of heat transfer are more favourable since they can be readily implemented in an existing heat exchanger and the fabrication of the inserts is straightforward. The most widely utilized passive heat transfer augmentation methods are ribs, fins, dimples, mesh inserts, wire coils, twisted tapes, and more.

Different heat transfer enhancers are:-
(a) Fins and micro fins,
(b) Porous media,
(c) Large particles, suspensions,
(d) phase-change devices,
(e) Flexible seals,
(f) Flexible complex seals, and
(g) Vortex generators.

Most of heat transfer augmentation methods presented in the literature that assists fins and microfins in enhancing heat transfer are reviewed.

2.1 Mechanisms of Augmentation of Heat Transfer:-

The mechanisms of heat transfer enhancement can be at least one of the following

a) Use of a secondary heat transfer surface.
b) Disruption of the laminar sub layer in the turbulent boundary layer.
c) Promoting boundary-layer separation.
d) Promoting flow attachment/reattachment.
e) Delaying the boundary layer development.
f) Thermal dispersion.
g) Redistribution of the flow.
h) Increasing the difference between the surface and fluid temperatures.
i) Increasing fluid flow rate passively

2.2 Experimental Work Done for Enhancement:-

The experimental setup consists of a one-meter-long tube with a 25 mm diameter and 0.5 mm thickness. A constant heat flux of 125 W is applied to the tube’s circumference, and airflow through the pipe is analyzed at different velocities. The system is modeled using Gambit software and analyzed with Fluent software. A central rod supports symmetrically placed inserts, with variations in pitch (3 cm, 5 cm, and 10 cm) and angle of attack (90°, 60°, 45°, and 30°). The CFD model is validated against experimental results by comparing temperature and pressure variations.

3. Modelling Setup and Governing Equations:-

Establishing the governing equations is the first stage in doing a CFD analysis of a system. The energy equation, momentum equation, and continuity equation are the ones at play. The equations for motion and energy are, respectively

$$\rho \frac{D\vec{u}}{Dt} = -\nabla p + \rho\vec{g} + \mu\nabla^2\vec{\mu} \qquad \text{................ (1)}$$

$$\rho \frac{D\vec{u}}{Dt} = \left[\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right)\right] + \left[u\frac{\partial p}{\partial x} + v\frac{\partial p}{\partial y} + w\frac{\partial p}{\partial z}\right] + \mu\varphi \qquad \text{................ (2)}$$

The governing equations for heated flow through a rectangular porous channel are continuity.

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \qquad \text{................. (3)}$$

X-Momentum

$$\rho\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = -\frac{dp}{dx} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) \qquad \text{..................... (4)}$$

Y-Momentum

$$\rho\left(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right)$$

..................... (5)

Z-Momentum

$$\rho\left(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)$$

..................... (6)

Energy:

$$\left(u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z}\right)=\frac{1}{\alpha}\left(\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}\right)$$

..................... (7)

Because of their intricacy, these equations must be solved computationally. Since these equations are all connected, solving them all at once requires the employment of a computational code. These equations were solved for this analysis using the commercial code FLUENT.

3.1 Discretization methods:-

(i) The Finite Volume Method (FVM) is one of the discretization techniques being employed. In research codes and commercial software, this is the "classical" or standard method that is most frequently employed. Discrete control volumes are used to solve the governing equations. FVM discretizes the N-S equation by recasting its partial differential equations (PDEs) in the conservative form.

$$\frac{\partial}{\partial t}\iiint QdV+\iint \mathrm{FdA}=0 \qquad \text{..................... (8)}$$

Where "A" is the cell surface area, V is the cell volume, F is the vector of fluxes, and Q is the vector of conserved variables.

(ii) Technique of finite elements (FEM). Although it can also be used for fluids, this approach is widely used for solid structure analysis. However, more caution is needed when using the FEM formulation to guarantee a conservative answer. For the Navier-Stokes equations, the FEM formulation has been modified. FEM is far more stable than the FVM technique, even though conservation needs to be considered.

A weighted residual equation is created using this method:

$$R_i=\iiint W_i Q dV^e \qquad \text{.................(9)}$$

4. Descriptions of the Problem and Geometry:-

A tube of length one meter and diameter one inch with thickness 0.5 mm is the dimension of model under consideration. Constant wall flux 125 watt is maintained at the circumference of hollow tube. Objective of this experiment is to remove heat by the air that flow through this pipe with a certain velocity. For this case firstly it is modeled in gambit software and analyzed in fluent software. Values of pressure drop and temperature difference is validated for experimental resultant. In the next case, an insert of in the form of ring is used. The results are analyzed for three different parameter like angle of insert, spacing between insert and Reynolds number. Objective of experiment is to find out at which angle, pitch spacing and Reynolds number, minimum pressure drop is observed with high heat transfer rate. For the angle variation, trial is made for different angle. Meshing is the most important work in this project. If the meshing is fine, result will be more accurate. For the analysis fluent software is used. At different iteration, convergence of solution takes place.

After smooth tube validation, tube with insert is used for analysis. Inserts are made of aluminum. Length of insert is 10.5mm, inclined part width is 2mm, & pitch of insert is 10 cm. A rod of diameter 2mm is placed axially inside the tube to provide a support for inserts. Now flow rate is changed at each trial. Pressure drop across the tube and Temperature variation is studied throughout the tube.

4.1 Validation of CFD model

An essential part of any modeling study is comparing numerical predictions with experimental data. The main goal is to show that CFD models are accurate enough to be used for simulation with confidence and to have the results taken into account when making design decisions.

4.2 Validation of Smooth Tube:-

The Gambit software is used for bottom-up validation modeling of the tube, and the meshing tool in Gambit is used for tetrahedral meshing.

Tube specifications:

- Length - 1000 mm
- Diameter – 25 mm
- Thickness – 0.5 mm

The simulation is done at different Reynolds number ranging from 15000 to 25500.

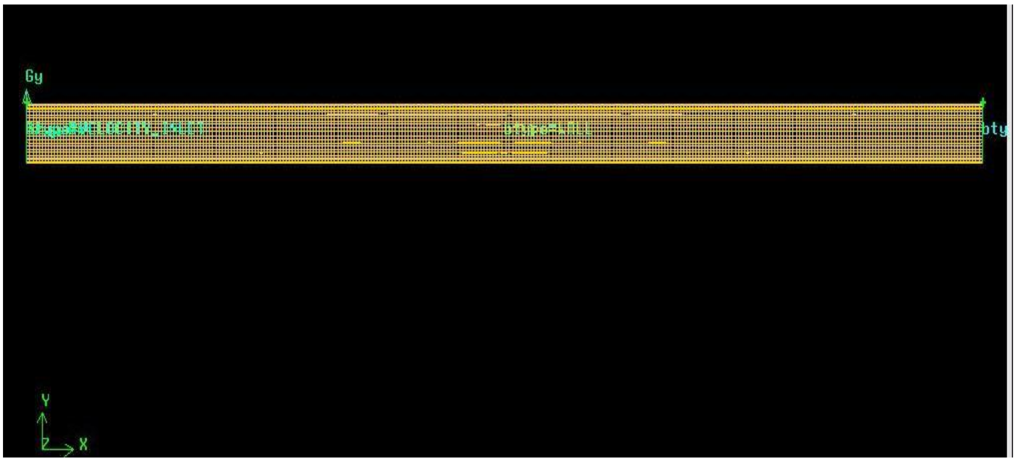

Fig.4.1 Geometry of Tube in Gambit

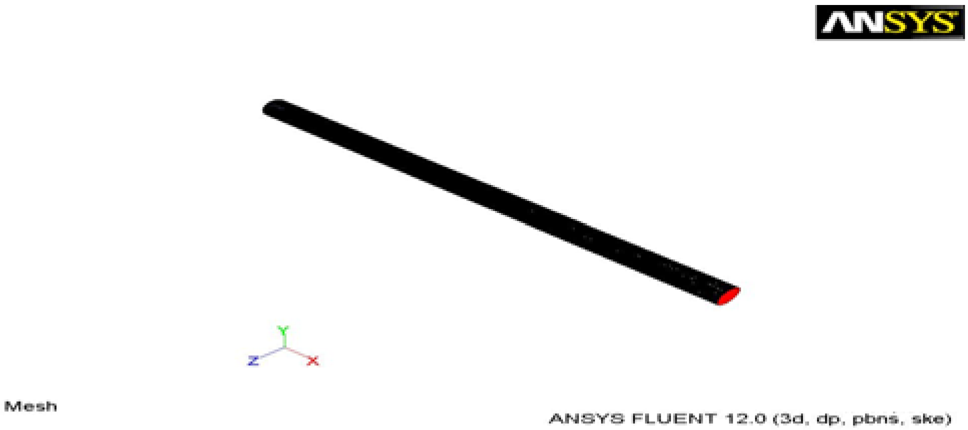


Fig4.2 mesh of tube

| | |
|---|---|
| Cells- | 46020 |
| Faces | 140020 |
| Nodes | 48060 |
| Partitions | 1 |

1 cell zone, 4 face zones.

Software used:-
For modelling and meshing:-GAMBIT
For analysis:-ANSYS FLUENT 12.0

4.3 Validation Results
Validation of the Gambit Model is done at different Reynolds number, while Temperature and Pressure variation throughoutthe tube is monitored to get the temperature and pressure profile.

- **At Reynolds no- 15373 for smooth tube:-**

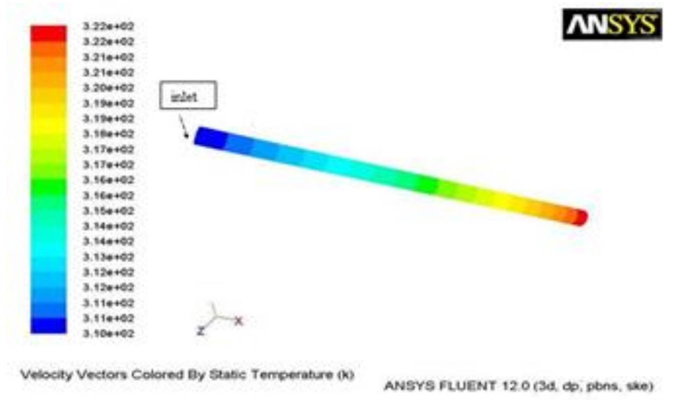


Fig 4.3 Temperature Profile of smooth tube at Reynolds N0.-15373

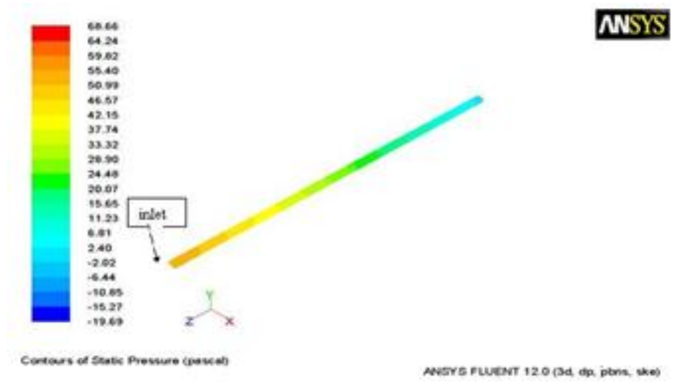


Fig 4.4 Pressure Profile of smooth tube at Reynolds N0.-15373

- **At Reynolds no- 18828 for smooth tube:-**

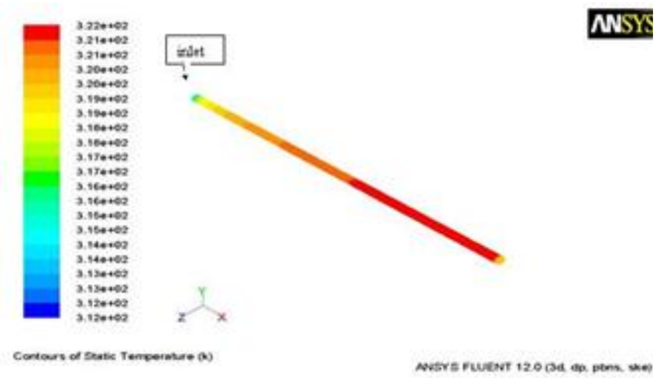


Fig 4.5 Temperature Profile of smooth tube at Reynolds N0.-18828

| Trial | Reynolds no. | ΔT °C (exp) | ΔT °C (CFD) | ΔP (exp) in Pa | ΔP (CFD) in Pa | % error (ΔT °C) | % error (ΔP ) |
|---|---|---|---|---|---|---|---|
| 1 | 15373 | 14 | 12 | 78.31 | 68.48 | 14.28 | 12.82 |
| 2 | 18828 | 13 | 10 | 107.46 | 92.74 | 23.07 | 13.69 |
| 3 | 21741 | 12 | 11 | 186.186 | 162.125 | 8.33 | 12.92 |
| 4 | 24307 | 11 | 10 | 215.584 | 190.243 | 9.91 | 11.75 |

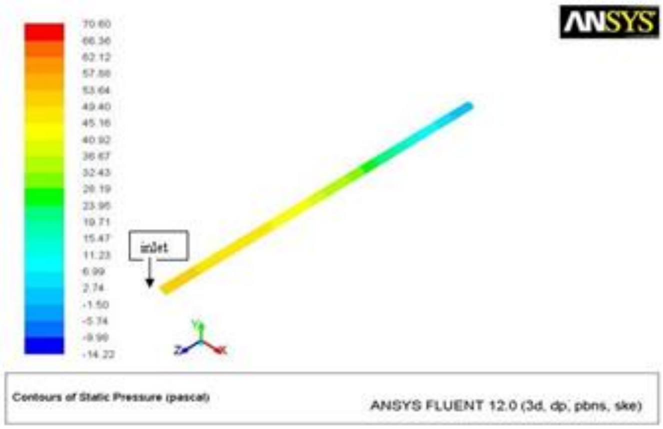


Fig 4.6Pressure Profile of smooth tube at Reynolds N0.-18828

- **At Reynolds no- 18828 for smooth tube:-**

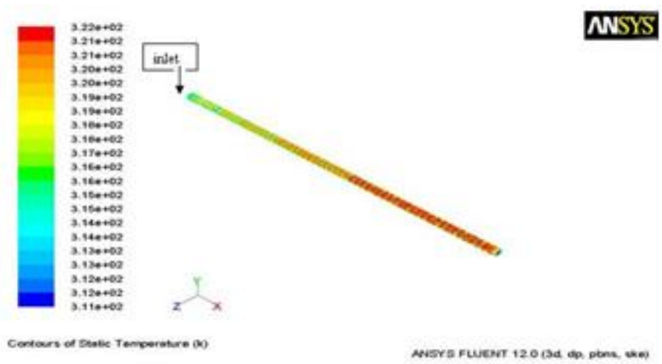


Fig 4.7Temperature Profile of smooth tube at Reynolds N0.-21741

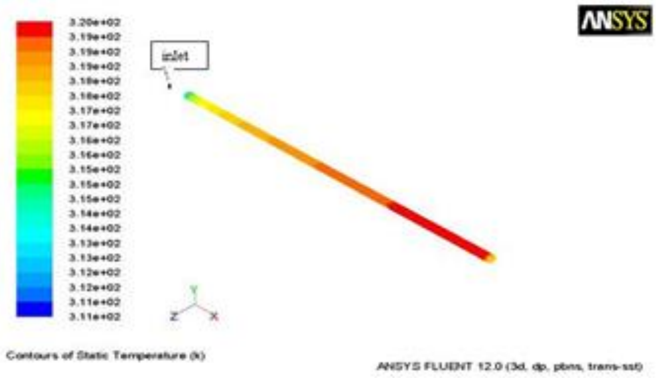


Fig 4.8Pressure Profile of smooth tube at Reynolds N0.-21741

**Table –1 Comparison of pressure drop and temperature change over smooth tube**

**Table – 2 Comparison of Nusselt No.**

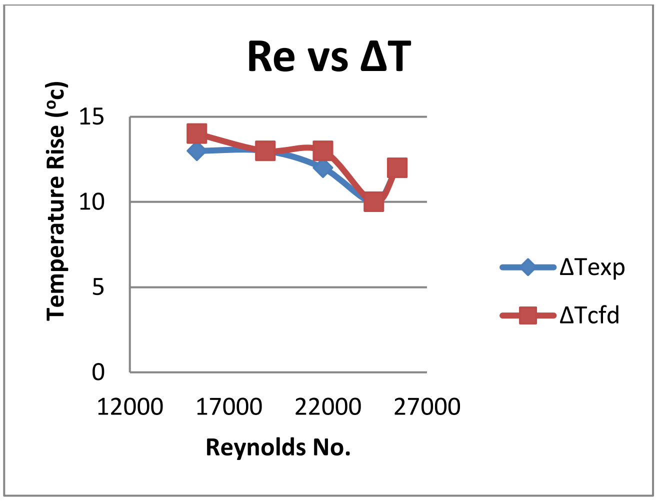


Fig 4.9 Validation by Temperature

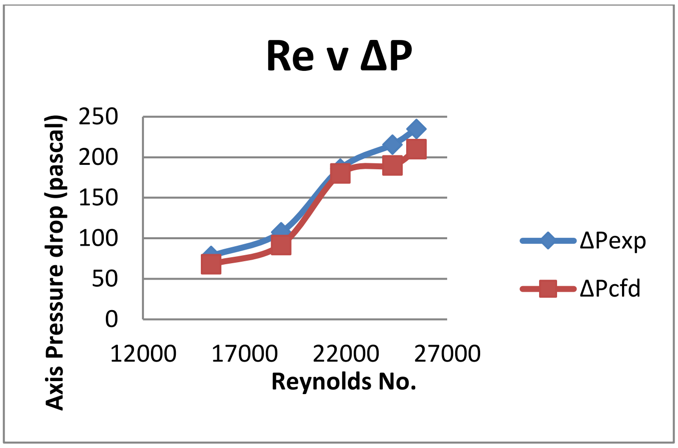


Fig 4.10 Validation by Pressure

Since the above CFD results for smooth tube is satisfactory with respect to the experimental results. Hence the further modification can be done in the model to get the heat transfer enhancement by adding circular rings to it and varying other

| Trial | Reynolds no. | Nusselt No (CFD) | Nusselt No (exp) | % error |
|---|---|---|---|---|
| 1 | 15373 | 118.786 | 123.871 | 4.11 |
| 2 | 18828 | 123.953 | 131.482 | 5.73 |
| 3 | 21741 | 135.784 | 152.125 | 10.72 |
| 4 | 24307 | 157.139 | 183.537 | 14.38 |

parameters

### 5. CFD Results and Analysis

The analysis is carried out for 3 different pitches (distance between two consecutive inserts)

1) 10 cm
2) 5cm
3) 3cm

At different angle of rings i.e. 90°, 60°, 45°, and 30°

CFD analysis for the above arrangement is done at different Reynolds number by varying the flow rate of air at the inlet of tube.

The CFD simulations reveal that reducing the insert angle leads to a decrease in pressure drop, while temperature variations exhibit no significant trend. The pressure drop is found to be minimal at an insert angle of 30° and maximum at 90°. As spacing between inserts increases, pressure drop initially rises and then declines. The optimal configuration for heat transfer enhancement is determined based on these findings.By analysis of the above models following results are obtained and they were plotted in a graph for different pitch of rings. And on the basic of the results optimum combination for heat transfer enhancement is suggested.

The entire simulation depends on two parameters: the angle of the inserts and the distance between them. It was discovered that while the tube's pressure drop diminishes with decreasing insert angle, the air temperature chance does not follow a consistent pattern. The pressure drop is greatest at 90 degrees and lowest at 30 degrees. Pressure drop first rises and then falls with increasing insert spacing. However, the temperature differential first rises and then falls as the separation grows. Since the temperature difference varies very little, pressure drop is now the primary criterion.The maximum temperature rise for each pitch is examines as well as the corresponding pressure drop.

**Table 3 Optimum Result**

| Pitch (cm) | Angle (degree) | ΔT ( $^o$c) | ΔP (Pascal) |
|---|---|---|---|
| 10 | $90^o$ | 19 | 1408 |
| 5 | $45^o$ | 22 | 1362 |
| 3 | $30^o$ | 21 | 1001 |

**6. CONCLUSIONS:**

This experimental investigates heat transfer enhancement in tubes fitted with inserts through experimental and CFD analysis. The results indicate that insert angle and pitch significantly influence pressure drop and temperature variation. The optimal configuration consists of an insert pitch of 3 cm and an angle of 30°. Future research can explore the effects of asymmetric insert arrangements and variations in spacing between inserts.

There is some future work regarding this project. Angle between insert at upper side of support and lower side can be varied. Similarly spacing between inserts at upper side can be varied with lower side.

**Refrences –**

[1] *Chi-Chuan Wang , Jerry Lo , Yur-Tsai Lin , Min-Sheng Liu* “Flow visualization of wave-type vortex generators having inline fin-tube arrangement”

[2] *S.Ray, A.W. Date* “Friction and heat transfer characteristics of flow through square duct with twisted tape insert”

[3] *J.W. Rose , A. Briggs* “Performance Comparison Of Some Tube Inserts”

[4] Y Chen, M. Fiebig, N. K. Mitra, “Conjugate heat transfer of a _nned oval tube with a punched longitudinal vortex generator in form of a delta winglet parametric investigations of the winglet”.

[5] Martin Feibig, “Embedded vertices in internal flow: heat transfer and pressure loss enhancement”.

[6] A.m. Jacobi, R.K. Shah, “Heat transfer surface enhancement through the use of longitudinal vortices’.

[7] Stefan tigglebeck, Nimal mitra, Martin febig, “Flow structure and heat transfer in a channel with multiple longitudinal vortex generators”.

[8] 3D numerical and experimental analysis for thermal–hydraulic characteristics of air flow inside a circular tube with different tube inserts, *Yu-Wei Chiu, Jiin-Yuh Jang.*

[9]Energy dissipation analysis of transient heat transfer for turbulent flow in a circular tube with baffe inserts. Ahmet Tandiroglu *, Teoman Ayhan b

[10] CFD analysis of single-phase flows inside helically coiled tubes.J.S. Jayakumara,b,∗, S.M. Mahajania, J.C. Mandala, Kannan N. Iyer a, P.K. Vijayan. (2009)

[11] Experimental and CFD studies on heat transfer and friction factor characteristics of a tube equipped with modified twisted tape inserts. Masoud Rahimi* , Sayed Reza Shabanian, Ammar Abdulaziz Alsairafi