International Journal of Recent Trends in Engineering & Research
ISSN (ONLINE) : 2455 – 1457
IMPACT FACTOR : 4.101

Numerical Analysis of Heat Transfer in a Pipe Using Twisted Tape Inserts

R.Santhaseelan[1] , Dr. C. Mathalai Sundaram[2], T. Sudarsanan[3]
[1] *Assistant Professor, Department of mechanical Engineering, Nadar Saraswathi College of Engineering & Technology, Theni*
[2]*Principal, Nadar Saraswathi College of Engineering & Technology, Theni*
[3] *Assistant Professor, Department of mechanical Engineering, Nadar Saraswathi College of Engineering & Technology,Theni*

Abstract - This paper reports numerical examinations of warmth move qualities in swirling stream conditions utilizing CFD simulation. A commercial CFD package, STAR CCM+, was utilized in this examination. 3D models for circular tube fitted with twisted tape inserts were created for the simulation. The swirling stream was presented by utilizing wound tape set inside the circular pipe. The outcomes got from the twisted tape insert are compared with those without turned tape. The information acquired from the CFD reproduction was confirmed with inlet and outlet temperature distinction and warmth exchange attributes. The outcomes show that there was a significant increase in heat transfer coefficient and Reynolds number in the pipe fitted with twisted tape.

Keywords - ***swirling*** flow, twisted tape, circular pipe, heat transfer coefficient

I. INTRODUCTION

1.1 Overview of CFD

Computational fluid elements, typically condensed as CFD, are a part of fluid mechanics that utilizes numerical techniques and calculations to take care of and break down issues that include fluid streams. PCs are utilized to play out the estimations required to recreate the collaboration of fluids and gases with surfaces characterized by limit conditions. With rapid supercomputers, better arrangements can be accomplished. Continuous research yields programming that enhances the exactness and speed of complex reproduction situations, for example, transonic or fierce streams. Starting trial approval of such programming is performed utilizing a breeze burrow with the last approval coming in full-scale testing..

1.2 Finite volume method

The finite volume method (FVM) is a common approach used in CFD codes, as it has an advantage in memory usage and solution speed, especially for large problems, high Reynolds number turbulent flows, and source term dominated flows (like combustion).

In the finite volume method, the governing partial differential equations (typically the Navier - Stokes equations, the mass and energy conservation equations, and the turbulence equations) are recast in a conservative form, and then solved over discrete control volumes. This discretisation guarantees the conservation of fluxes through a particular control volume. The finite volume equation yields governing equations in the form of,

$$\frac{\partial}{\partial t}\iiint Q\,dV + \iint F\,d\mathbf{A} = 0,$$

where
Q is the vector of conserved variables
F is the vector of fluxes (see Euler equations or Navier–Stokes equations)
V is the volume of the control volume element, and
A is the surface area of the control volume element.

1.3 Turbulence models

In computational modeling of turbulent flows, one common objective is to obtain a model that can predict quantities of interest, such as fluid velocity, for use in engineering designs of the system being modeled. For turbulent flows, the range of length scales and complexity of phenomena involved in turbulence make most modeling approaches prohibitively expensive, the resolution required to resolve all scales involved in turbulence is beyond what is computationally possible. The primary approach in such cases is to create numerical models to approximate unresolved phenomena. This section lists some commonly-used computational models for turbulent flows.

Turbulence models can be classified based on computational expense, which corresponds to the range of scales that are modeled versus, resolved (the more turbulent scales that are resolved, the finer the resolution of the simulation, and therefore the higher the computational cost). If a majority or all of the turbulent scales are modeled, the computational cost is very low, but the tradeoff comes in the form of decreased accuracy.

1.4 Methodology

In all of these approaches the same basic procedure is followed during preprocessing.

- The geometry of the problem is defined.
- The volume occupied by the fluid is divided into discrete cells. The mesh may be uniform or non-uniform.
- The physical modeling is defined
- Boundary conditions are defined. This involves specifying the fluid behaviour and properties at the boundaries of the problem.
- The simulation is started and the equations are solved iteratively as a steady-state or transient.
- Finally a postprocessor is used for the analysis and visualization of the resulting solution.

1.5 About Software's

In this Numerical analysis of heat transfer in circular tube project two different software's are used. They are,

1. Pro/ENGINEER Wildfire 5.0
2. STAR CCM+

1.5.1 Pro/EWildfire 5.0

Creo Elements/Pro (formerly Pro/ENGINEER), PTC's parametric, integrated 3D CAD/CAM/CAE solution, is used by discrete manufacturers for mechanical engineering, design and manufacturing. Creo Elements/Pro provides a complete set of design, analysis and manufacturing capabilities on one, integral, scalable platform. These required capabilities include Solid Modeling, Surfacing, Rendering, Data Interoperability, Routed Systems Design, Simulation, Tolerance Analysis, and Tooling Design

1.5.2 STAR CCM+

STAR-CCM+ is unrivalled in its ability to tackle problems involving multi-physics and complex geometries. STAR-CCM+ has an established reputation for producing high-quality results in a single code with minimum user effort. Designed to fit easily within your existing engineering process, STAR-CCM+ helps you to entirely automate your simulation workflow and perform iterative design studies with minimal user interaction. The net result of this is that engineers get to spend more time actually analyzing engineering data and less time preparing and setting up simulations.

II - NUMERICAL ANALYSIS OF FLOW CHARACTERISTICS IN PLAIN TUBE

2.1 Flow In Pipes

Fluid flow in circular and noncircular pipes is commonly encountered in practice. The hot and cold water that we use in our homes is pumped through pipes. Water in a city is distributed by extensive piping networks. Oil and natural gas are transported hundreds of miles by large pipelines. Blood is carried throughout our bodies by arteries and veins. The cooling water in an engine is transported by hoses to the pipes in the radiator where it is cooled as it flows. Thermal energy in a hydraulic space heating system is transferred to the circulating water in the boiler, and then it is transported to the desired locations through pipes. Fluid flow is classified as external and internal, depending on whether the fluid is forced to flow over a surface or in a conduit. Internal and external flows exhibit very different characteristics.

2.1.1 Properties of pipe flow

Liquid or gas flow through pipes or ducts is commonly used in heating and cooling applications and fluid distribution networks. The fluid in such applications is usually forced to flow by a fan or pump through a flow section and pay particular attention to friction, which is directly related to the pressure drop and head loss during flow through pipes and ducts. The pressure drop is then used to determine the pumping power requirement. A typical piping system involves pipes of different diameters connected to each other by various fittings or elbows to route the fluid, valves to control the flow rate, and pumps to pressurize the fluid. The terms pipe, duct, and conduit are usually used interchangeably for flow sections. In general, flow sections of circular cross section are referred to as pipes (especially when the fluid is a liquid), and flow sections of non circular cross section as ducts (especially when the fluid is a gas).

2.2 **Technical Details**

The configuration of the twisted tape inserts is illustrated in below table. An aluminium tape of 2 mm thickness and 25mm width is uniformly winding over a length of produce twist ratio of 8 respectively. The twist ratio “y” is defined as the ratio of the length of one full twist (360∘) to the tape width. Steel tube with a diameter (D) of 30mm and length (L) of 2000mm was used as test section and water was selected as the working fluid. The dimensions and the thermo physical properties of fluid and materials are summarized in Table .

Technical details twisted tape

Parameters	Description
Inner tube diameter d1 (mm)	30
Outer tube diameter (mm)	35
Test tube length (mm)	2000
Material of test tube	Steel
Density (Kg/)	1000
Specific Heat (J/Kg K)	502.48
Thermal Conductivity (W/mK)	16.27
Tape pitch length (H,)/mm	200
Twist ratio (Y=H/W)	8
H	Linear distance of the tape for rotation
W	Width of the twisted tape
Tape thickness (mm)	2
Tape width (mm)	25
Material of twisted tape	Aluminium

Technical details test conditions

Parameters	Description
Fluid	Water
Renolds number (Re)	100 – 2100
Type of flow in inner tube	Laminar
Inlet temperature (K)	333

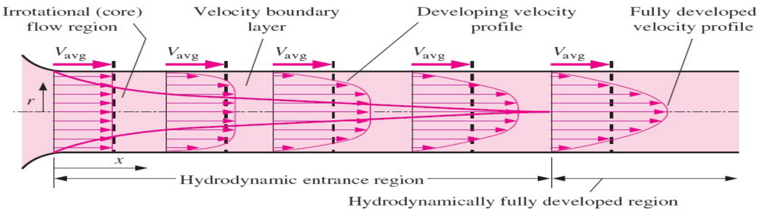
2.3 Drawing and Meshing

Drawing and meshing is the first step of the numerical analysis of flow characteristics in plain tube. Drawing should be done using the Pro-E wild fire software. In the CFD software only draw the fluid area of the system. Only the IGES files should be able to import in the Star CCM+, so drawn Pro E file should save as the IGES file as a surface mesh. The IGES file should mesh using Star CCM+ software and mesh view should shown in fig

Mesh diagram

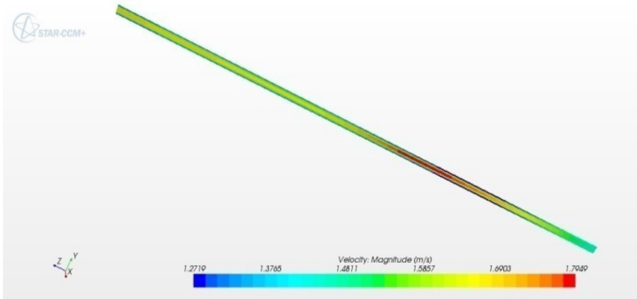
2.4 Velocity Profile

Consider a fluid entering a circular pipe at a uniform velocity. Because of the no-slip condition, the fluid particles in the layer in contact with the surface of the pipe come to a complete stop. This layer also causes the fluid particles in the adjacent layers to slow down gradually as a result of friction. To make up for this velocity reduction, the velocity of the fluid at the midsection of the pipe has to increase to keep the mass flow rate through the pipe constant. As a result, a velocity gradient develops along the pipe. The region of the flow in which the effects of the viscous shearing forces caused by fluid viscosity are felt is called the velocity boundary layer or just the boundary layer. The thickness of this boundary layer increases in the flow direction until the boundary layer reaches the pipe center and thus fills the entire pipe, as shown in Fig



Velocity profile

The region from the pipe inlet to the point at which the boundary layer merges at the centerline is called the hydrodynamic entrance region, and the length of this region is called the hydrodynamic entry length. Flow in the entrance region is called hydro dynamically developing flow since this is the region where the velocity profile develops. The region beyond the entrance region in which the velocity profile is fully developed and remains unchanged is called the hydro dynamically fully developed region. The flow is said to be fully developed when the normalized temperature profile remains unchanged as well. Hydro dynamically developed flow is equivalent to fully developed flow when the fluid in the pipe is not heated or cooled since the fluid temperature in this case remains essentially constant throughout.

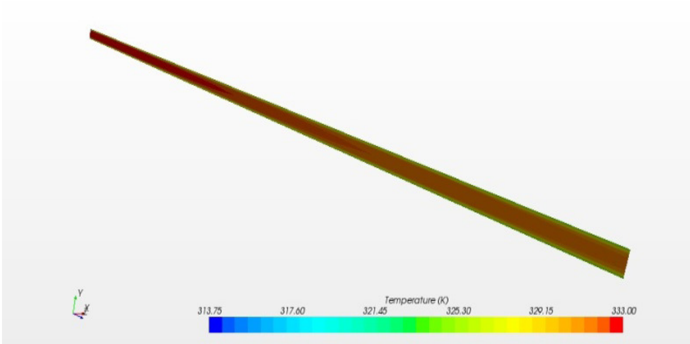


Velocity profile

This derived result clearly shows the velocity creation of inside the pipe. The Velocity is maximum at the center of the pipe.

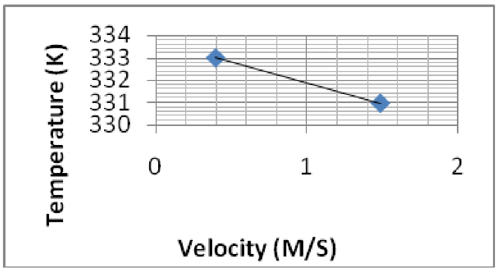
2.5 Temperature Difference

Inlet temperature of the water is 333K and velocity is 0.385 m/s and numerically analyzed results are shown in fig.



Temperature at whole length of the pipe

Fig shows only small amount of heat should be transferred to the pipe to the surface. Thus the heat transfer rate is minimum in the ordinary circular tube and outlet temperature of the ordinary circular tube is 330.975K.

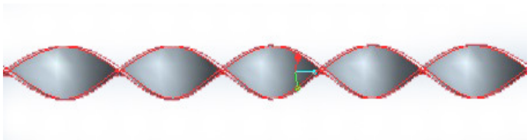


Temperature Vs Velocity

Thus the graph shows the temperature difference and velocity difference in inlet to outlet the straight line shows the heat transfer in the pipe. The inlet velocity of the pipe is 0.385m/s and exit velocity of the pipe is 1.485 m/s. The inlet temperature is 333K and exit temperature of plain tube is 330.975 K.

2.6 Geometry and Mesh

Geometry and Mesh Building Pro-E wild fire software is used for generation of geometry and grids whereas Star CCM+ is used in the module preprocessing. This geometry consists of a cylindrical tube of diameter 30 mm and length of2000mm. The geometry of plain twisted tape inserts with different twist ratio is generated winding uniformly strip of 25 mm width using the twist option in the sweeping of faces at a twist angle of 360º and a length of200 for various twist ratio namely . The volume required for simulation is created by the subtraction of twisted tape geometry from the plain tube geometry. The edge meshing is applied to each edge by using of the particular interval count, whereas the front circular face is meshed by using tetrahedral and pave type meshing. The meshed face was swept over the entire volume using tetrahedral/hybrid elements and T Grid type. Boundary conditions for the meshed volume; inlet, outlet, wall and type of fluid (water) are defined. Subsequently, the mesh file was exported to Star CCM+ for simulation.



Twisted tape

2.7 Mesh view

The derived Mesh of the Circular tube with inserted twisted tape inserts is shown in fig

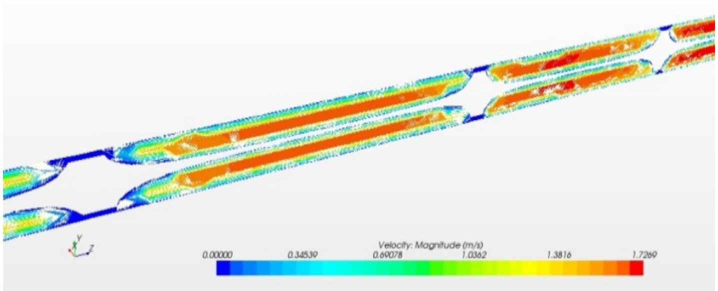
Mesh view of pipe with twisted tape inserts

The orange color should indicate the inlet of the pipe. And the pipe with twisted tape is Meshed as a surface meshing. After the mesh is created the geometrical assumptions are given. STAR CCM+ is used for solver execution and post-processing of the CFD simulation. Solution sequential algorithm (segregated solver algorithm) has been used with settings implicit formulation, steady (time-independent) calculation, viscous laminar model and energy equation. Coupling method and first -

Order Up wind scheme for energy and momentum equations as well as default values for under-relaxation factors and convergence criterion.

2.8 Velocity Profiles

In this section, the results obtained by CFD simulation are presented and discussed. Fig. shows the velocity obtained by simulation and correlations developed by STAR CCM.

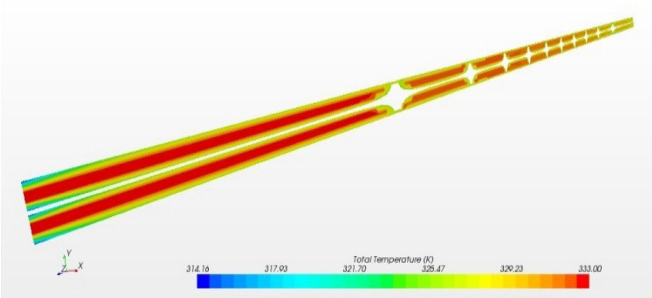


Velocity profile of pipe with twisted tape inserts

Fig shows the velocity profile of the circular tube after the twisted tape inserts. Compare to the ordinary circular tube, the velocity is maximum at the center of the pipe on the ordinary circular tube but after the twisted tape inserts we got two maximum velocities. This figure shows the longitudinal and radial vortices generated in the quadrant-cut twisted tape inserts. These vortices play a critical role for disturbing the boundary layer and uniform the temperature in the core flow. Consequently, enhanced mixing by extension heat transfer rate between the fluid at the core and the heated surface can be achieved. The velocity is directly proportional to the heat transfer, so if velocity is increase the heat transfer is also increased. Thus figure shows that the velocity at wall of the pipe. In the circular pipe the velocity at wall is zero and we got two maximum velocity points.

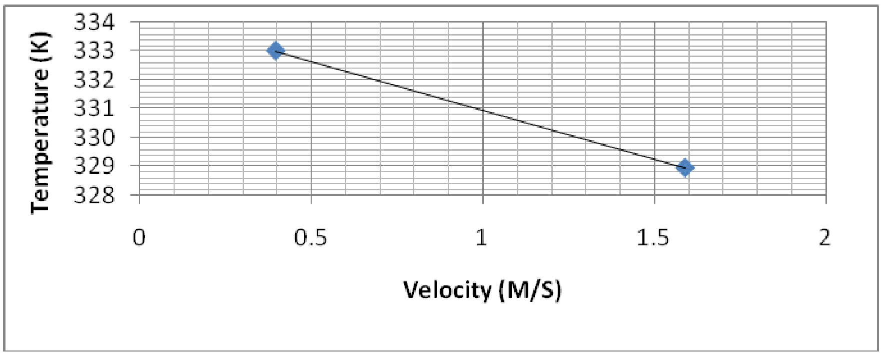
2.9 Temperature Difference

Figure shows the inlet temperature of the plain tube is 333K and surface temperature is room temperature 306K and the exit region of the plain tube twisted tape inserts. The heat transfer rate is increased after the twisted tape is inserted. The outlet temperature of the pipe is 328.95K, which is more efficient than ordinary circular tube. The temperature distribution is shown in figure



Temperature distribution of pipe with twisted tape inserts

Fig shows the contour of temperature field for plain tube with twisted tape inserts with twisted ratio (y = 2.93) produced better temperature distribution on tube wall than plain tube. From this maximum heat transfer rate compare to plain tube. For all simulated data, it is found that the Nusselt number in the tube fitted with twisted is higher than the plain tube. The twisted tape provides an additional turbulence to the fluid it is main factor for increasing heat transfer. The temperature difference and velocity difference is showed in fig.



Temperature Vs Velocity of pipe with twisted tape inserts

The graph shows the Temperature difference between the inlet and outlet temperature for the circular tube with twisted tape inserts. Inlet temperature of the pipe is 333K and outlet temperature is 328.95K. Velocity at inlet is 0.397m/s and velocity at exit is 1.589m/s.

III - RESULTS AND DISCUSSIONS

3.1 Effect of Twist in Heat Transfer

Heat transfer rate with the application of the twist tape is much higher than that of the plain tube. The higher heat transfer rate is most likely due to enhancement of mixing as a result of swirl flow generation. In addition, the smaller twist ratio produced higher turbulent intensity with longest flow path than those of larger twist ratios (y).Variation of friction factor with Reynolds number for different twist ratios. It seems that the friction factor generated by twisted tape insert with smaller twist ratio is significantly higher than the plain tube and other twist ratios. Subsequently, the smallest twist ratio leads to higher tangential contact between the swirling flow and the tubes surface. This phenomenon is due to the combined effects of the swirling flow and turbulence that occurred at smallest alternative cuts along the edge of the twisted tape resulting in higher destruction of the thermal boundary layer which leads to enhanced mixing between the fluid at the core and the heated surface.

3.2 Velocity Field

. The velocity vectors predicted for the plain tube and the twisted tape inserts are presented which shows the longitudinal land radial vortices generated in the twisted tape inserts. These vortices play a critical role for disturbing the boundary layer and uniform the temperature in the core flow. Consequently, enhanced mixing by extension heat transfer rate between the fluid at the core and the heated surface can be achieved.

3.3 Temperature Field

Contour of temperature field for plain tube and twisted tape inserts are to be compared and the twisted tube inserts will be produced better temperature distribution on tube wall than plain tube

IV - CONCLUSION

In the present study, CFD simulation for the heat transfer augmentation in a circular tube fitted with plain twisted tape in laminar flow conditions has been reported using STAR CCM+. The following conclusions can be drawn from the results of the present study.

➢ The presence of the tube with twisted tape inserts yields a higher heat transfer rate (Nu) of the plain tube due to the formation of vortices that disturbed the boundary layer, and, thus, enhanced mixing by extension heat transfer rate between the fluid at the core and the heated surface can be achieved.

- For all simulated data, it is found that the Nusselt number in the tube fitted with twisted is higher than the plain tube. The twisted tape provides an additional turbulence to the fluid it is main factor for increasing heat transfer

REFERENCES

[1] S.R. Shabanian, M. Rahimi, M. Shahhosseini and A.A. Alsairafi, "CFD and experimental studies on heat transfer enhancement in an air cooler equipped with different tube inserts," International Communications in Heat and Mass Transfer, vol. 38, No. 3, pp. 383–390, 2011.

[2] N Saran Kumar, N Kaleeswaran, B Radha Krishnan, "Review on optimization parametrs in Abrasive Jet Machining process" , International Journal of Recent Trends in Engineering and Research, Volume 04, Issue 10; October- 2018 [ISSN: 2455-1457]. DOI:10.23883/IJRTER.2018.4388.HTNOQ

[3] P. Sivashanmugam, P. K.Nagarajan, and S. Suresh, "Experimental studies on heat transfer and friction factor characteristics of turbulent flow through a circular tube fitted with right and left helical screw-tape insert," Chemical Engineering Communications, vol. 195, No. 8, pp. 977–987, 2008.

[4] J.S. Jayakumar, S. M.Mahajani, J.C.Mandal, P.K. Vijayan and R. Bhoi, "Experimental and CFD estimation of heat transfer in helically coiled heat exchangers," Chemical Engineering Research and Design, vol. 86, No. 3, pp. 221–232, 2008.

[5] Mala, D., S. Sendhilnathan, and A. Vembathu Rajesh. "Heat transfer analysis of al2o3 nanofluid in circular tube with two different twist inserts." Journal of Chemical and Pharmaceutical Sciences www. jchps. com ISSN 974 (2015): 2115.

[6] R. Kharat, N. Bhardwaj, and R.S. Jha, "Development of heat transfer coefficient correlation for concentric helical coil heat exchanger," International Journal of Thermal Sciences, vol. 48, No. 12, pp. 2300–2308, 2009.

[7] S.Sundar, T.Sudarsanan, B.Radha Krishnan, "Review of Design and Fabrication of four wheel Steering system", International Journal of Recent Trends in Engineering and Research, Volume 04, Issue 10; October- 2018 [ISSN: 2455-1457]. DOI:10.23883/IJRTER.2018.4386.3THDB

[8] P. K. Nagarajan and P. Sivashanmugam, "CFD Simulation of heat transfer augmentation in circular tube filled with right left helical inserts with spacer," International Journal of Food Engineering, vol. 6, No. 1, pp. 1–11, 2009.

[9] B. Radha Krishnan, et al. "Design and Analysis of Modified Idler in Drag Chain Conveyor." in International Journal of Mechanical Engineering and Technology, Volume 9, Issue 1, pp. 378–387.

[10] V. Kumar, S. Saini, M. Sharma, and K. D. P. Nigam, "Pressure drop and heat transfer study in tube-in-tube helical heat exchanger," Chemical Engineering Science, vol. 61, no. 13, pp. 4403 - 4416, 2006.

[11] A.VembathuRajesh, et.al "Heat Transfer Analysis of AL_2O_3 Nanofluid in Circular Tube with Copper Plate Winding" Inernational Journal of emerging Technology and innovative Engineering, Volume 03, Issue 1; October- 2017