ISSN 2395-1621

Cfd analysis of single phasecircular tube by means of groove twisted tape

#1Prakash B Patil,#2 Dr.Rameshrudrapati, #3Dr.K.P.Kolhe

1prakashpatil444@gmail.com
2rameshrudrapati@gmail.com
3skishor75@gmail.com

#13 Department of Mechanical Engineering,
JSPMs Imperial College Of Engineering And Research, Wagholi, Pune.412207, India.

ABSTRACT

In the recent era, due to limited availability of energy resources and ever increasing cost of the same,the different techniques are used to augment the heat transfer in active research areas. The enhanced surfaces in the heat exchangers are often suggested to augment the heat transfer and different active and passive techniques are recommended for this purpose. One of the passive techniques to enhance the heat transfer is the application of insertion devices such as twisted tapes. Here we use twisted tape which is one of the techniques to enhance the heat transfer because of the easy set up experimentation and relatively low cost compared to active type heat transfer augmentation. These researches focused on finding a technique not only increasing heat transfer, but also achieving high efficiency. Achieving higher heat transfer rates through various enhancement techniques can result in substantial energy savings, more compact and less expensive equipment with higher thermal efficiency. Heat transfer enhancement technology has been improved and widely used in heat exchanger applications; such as refrigeration, automotives, process industry, chemical industry, etc. One of the widely-used heat transfer enhancement technique is inserting different shaped elements with different geometries in channel flow .The use of CFD ANALYSIS is to model and predict the experimental results of heat transfer in circular tube equipped with the twisted tape.

ARTICLE INFO

Article History

Received :18th November 2015

Received in revised form :

19th November 2015

Accepted : 21st November , 2015

Published online :

22nd November 2015

Keywords— **CFD analysis, fluid flow, twisted tape, heat transfer, inserts.**

I. INTRODUCTION

The conversion, utilization, and recovery of energy in industrial, commercial, and domestic application usually involve a heat transfer process. Improved heat exchange, over and above that in the usual or standard practice, can significantly improve the thermal efficiency in such applications as well as the economics of their design and operation. The need to increase the thermal performance of heat based equipments (for instance, heat exchangers), thereby effecting energy, material, and cost savings as well as a consequential mitigation of environmental degradation has led to the development and use of many heat transfer enhancement techniques. These methods are referred to as augmentation or intensification techniques. Enhancement techniques essentially reduce, for example, the thermal resistance in a conventional heat exchanger by promoting higher convective heat transfer coefficient with or without surface area increases (as represented by fins or extended surfaces). As a result, the size of a heat exchanger can be reduced, or the heat duty of an existing exchanger can be increased, or the exchanger's operating approach temperature difference can be decreased. The latter is particularly useful in thermal processing of biochemical, food, plastic, and pharmaceutical media, to avoid thermal degradation of the end product. On the other hand, heat exchange systems in spacecraft, electronic devices, and medical applications, for example, may rely primarily on enhanced thermal performance for their successful operation. In the present work, heat transfer enhancement for fluid flowing through a pipe with wire coil inserts is to be analyzed using Computational Fluid Dynamics (CFD). The impressive improvements in computer performance, matched by developments in numerical methods, have resulted in a growing confidence in the ability of CFD to model complex fluid flows. CFD techniques have been applied on a broad scale in the process industry to gain insight into various flow phenomena, examine different equipment designs or compare performance under different operating conditions.

II.AIM & OBJECTIVE

The current literature shows the lack of study in the area of twisted tape insert. Hence results were varying continuously according to different parameters. In this paper the experimental and CFD analysis is compared with and without the groove twisted tape inserts and solve systematically.The aim of the study was to determination of friction factor and Nusselt number for smooth tube and for various groove twisted tape inserts with varying pitches and wave-widths.

Hence the objectives of this experiment are as following:

1. Determination of friction factor and Nusselt number for smooth tube and for various groove twisted tape inserts with varying pitches and wave-widths.

2. The results of Nusselt number, friction factor, Performance evaluation criteria for all the copper groove twisted tape inserts are plotted on the graph and compared with the values for the smooth tube.

3. To observe the effect of varying twists and wave-widths, another set of graphs are plotted for copper inserts and smooth tube.

4. Comparing results of groove twisted tape with plane tape.

III. EXPERIMENTAL WORK

The experimental study on passive heat transfer augmentation using groove twisted tape inserts for varying twist ratio and wave-width with copper as a material were carried on in a single phase flow heat exchanger having the specifications as listed below:-

A. *Specifications of set up:*

1 Inner diameter of pipe = (d_o) = 0.026 m.
2 Outer diameter of pipe = (d_i) = 0.03 m.
3 Length of test section = L = 0.7m.
4 Voltmeter = 0-200 V; ammeter 0-2 amp.

Nichrome wire (resistivity = 1.5 x $10^{-6}\Omega$m) heater wound around test pipe

Fig.1 Experimental set up for forced convection

B. *Specifications of inserts-*

- Width of tape, W = 24mm (Constant)
- Thickness of inserts, t = 1mm (Constant)
- Twist ratios, TR = 8.33, 8.83,9.375, 9.79 & 10.42
- Length of insert, L = 700 mm (Constant)
- Wave-width, WW = 13,16 & 24 mm
- length = 10mm

The apparatus consist of a blower unit fitted with the test pipe. The test section is surrounded by nichrome heater. Six thermocouples are embedded on test section and two thermocouples are placed in the air stream at the entrance and exit of test section to measure air temperatures. Test pipe is connected to the delivery side of the blower along with the orifice to measure flow of air through the pipe.

1) T_1 measures inlet air temp & is indicated by Digital Voltmeter (DV) on the front panel. The thermocouple no 8 is used for measuring outlet air temperature. The thermocouples T_2 to T_7 are used to measure surface temperature of test section. The power control unit uses an isolating transformer for stepping down mains voltage to 130 volts AC & get controlled DC output in the range 0-150 volts using uni junction transistor(UJT), pulse transformer &Silicon Controlled Rectifier (SCR) circuit & capacitive filter. The heater power is necessarily DC. The voltage signals (V) for measurement is obtained by means of a potential divider & current signal (I) is obtained by measuring the voltage drop across a small resistance. This current is measured by means of an ammeter on the front panel & for voltage measurement test points are provided below the ammeter. The output can be controlled by means of potentiometer. The ground is internally connected.

Table.1 Observation table without insert

Temperatures °C							Manometer Difference		Inclined Manometer Reading
T_1	T_2	T_3	T_4	T_5	T_6	T_7	T_8	h_w	h_f
32.7	42.6	48.9	52.6	56.8	57.6	54.3	40.3	41	3.25

Table 2. Observation table with copper insert

Temperatures °C								Manometer Difference	Inclined Manometer Reading
T_1	T_2	T_3	T_4	T_5	T_6	T_7	T_8	h_w	h_f
31.9	40.8	45.8	48.1	50.7	50.9	49.2	40.7	41	5

IV. CONCLUSION & GRAPH

A. *Parameters of heat transfer from circular tube without insert*

As per the observation table 1 experimentation was carried out and temperature and manometer difference values obtained. This manometer difference indicates change in mass flow rate operated by the valve next to the test section, this increase in manometric head indicates increase in mass flow rate and this causes increase in Reynolds number. As Reynolds number increases the friction factor has to reduce. As per calculations the values of parameters are obtained as follows.

Q_a = 0.003813 m^3/s, Re = 10963.77, f_{th}= 0.0306417, Nu_{th} = 35.16, f_{exp} = 0.0327, Nu = hD/k Nu_{exp}= 36.34

B. *Sample observation and result table of copper insert with TR-8.83, WW-16 mm*

As shown in observation table no. 2 and graphs, Reynolds number increases then friction has to reduce, then at lower Reynolds number the friction factor is high however as the Reynolds number increases the friction factor drops as compared to the initial value. The Nusselt number increases with the increase in Reynolds number which indicates that heat transfer enhancement takes place at higher Reynolds number.

Values given below as per calculation,
Q_a = 0.003292m^3/s, V= 6.2003 m/s, Re = 9466, f_{exp} = 0.06764, Nu =$\frac{hD}{k}$, Nu_{exp}= 51.12

V. VALIDATION USING CFD

There are many devices and systems that are very difficult to prototype. Using CFD analysis, we can predict the performance of a design and test many variations until an optimal result is obtained. To achieve these in physical proto typing and testing would require a huge amount of time and labor. The foresight gained from CFD analysis helps design better and faster. The flow inside tube with different types of inserts is characterized by a complex flow field, which is affected by blockage and recirculation zones enhanced by sharp edges. In this case the actual physical device is replaced by a discrete number of points that represent the entire geometry of the cell where the distributions of pressure, velocity etc. are to be found. The approach requires defining the mathematical equations that govern the physical process. These equations will be solved only at the discrete points representing the geometry. CFD techniques are used to perform the overall performance and optimization analysis of the tubes with/without inserts. The fluid flow and heat transfer of the horizontal tube were simulated using FLUENT. The parameters considered for simulation are same as the physical existing parameters for experimental set up like inner diameter of tube 28 mm, length 50cm, tube and insert materials: copper, heating section 45cm for plain tube etc.
The simulated CFD analysis results like Nusselt number and heat transfer coefficient are validated with the available experimental results for tube with/without mesh inserts.

- Meshing & Grid Generation

After creation of geometry, meshing was done. A section of the mesh generated is shown below:-

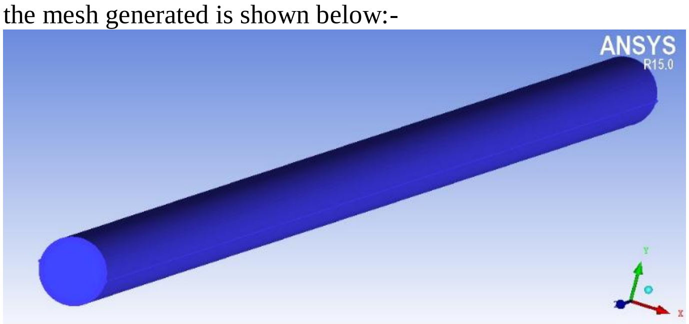

Figure No. 6 Circular Tube Geometry

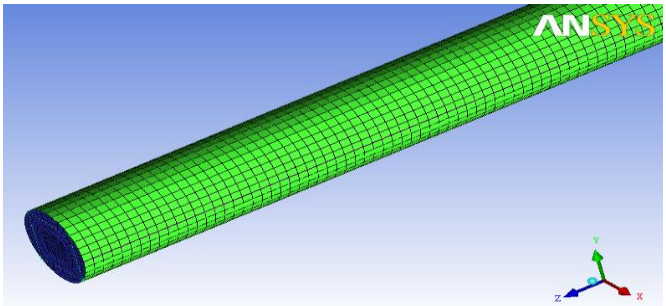

Figure No. 7Plane tube meshing

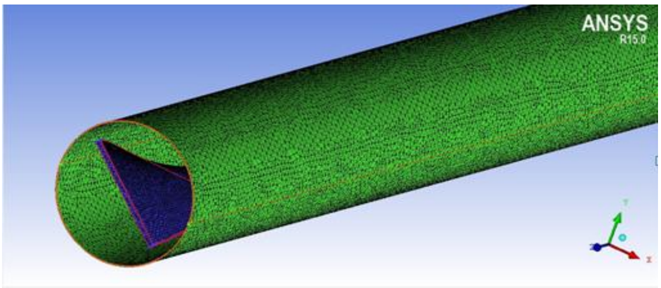

Figure No. 8Tube with inserts mesh

The CFD model was made of 3D- model, since the domain is a pipe. Suitable development lengths were given to nullify the effects of the boundary conditions. The mesh is of good quality tritet meshes with boundary layers at the wall region to predict the wall effects exactly. Mesh was refined in two steps and the results were checked. The results showed no change to the previous mesh step. So, a grid independence test was conducted and the final mesh results were presented in the report. The details for the CFD model and mesh are given below, First mesh 213308, second mesh 2230604, third 1357088 Forth mesh 2137881 Fifth mesh 1888285 &Sixth mesh 1318300. Since a boundary mesh to resolve the near wall region and a good quality quadrilateral mesh is used in the model the results are near reality.

- Boundary conditions

For setting up any CFD problem, the geometry has to be modeled with required details, mesh has to be generated optimally to obtain the results correctly and flow parameters and boundary conditions are to be set up for solving the problem. The discretized domain is solved using solver and results are analyzed in post processor. In the presentinvestigation, CAD model and ANSYS-ICEM 14.0 version software is used for the geometry modelling and mesh generation. ANSYS-FLUENT 14.0 version, software is used for defining boundary conditions, solving and post processing.

At inlet,
X = 0, mass flow rate-= 0.004328 kg/s

At outlet,
X= L, mass flow rate = 0.003737 kg/s

V.CONCLUSION

A The parameters mentioned in the above objectives are achieved by carrying out the experimentation and then making the comparisons of them for varying twist ratios and wave widths in the graphs drawn below.TheNusselt number, friction factor are the important parameters which decides the success of any experimentation work as both parameters are opposite to each other. The Nusselt number shows the percentage increase in heat transfer enhancement when inserts are placed inside a test pipe due to increase in heat transfer coefficient by comparing it, without inserts. Contradictory to it is when inserts are placed inside the test pipe the friction gets produced inside the test pipe due to which there is drop in pressure hence the desire increase in heat transfer coefficient is offset by pressure drop. Hence, the inserts should be designed in such a way that there pumping cost should get offset by heat enhancement.
When inserts are placed it is been observed that friction factor for Copper increases by only upto 2 times than smooth tube for TR-9.79 & WW-16mm. Copper inserts shows better result in terms of friction factor than smooth tube, hence insert with Copper is preferred.

A CFD analysis has been conducted to investigate heat transfer enhancement by means of groove twisted tape inserts in a single circular tube pipe heat exchanger using cold water and hot air as the test fluids. From the experimental results, it can be concluded as follows: It is found that enhancing heat transfer with passive method using different types of groove twisted tape construction in the inner tube of a single circular tube can improve the heat transfer rate efficiently.

ACKNOWLEDGEMENT

It's our great pleasure to present our project WORK entitled CFD analysis of single phase circular tube by means of groove twisted tape inserts. This experiment work is an outcome of various efforts by us in collating and identifying the sources of information and knowledge.We use this occasion to thank our guides with whose guidance this effort would not have borne fruits. We find no words to express our gratitude to Dr. KOLHE K.P who not only advised and guided us during the paper work but also answered all our queries concerning collection of data, proper structuring of the report, and its improvement. I would also thankful of our friends which gave us a proper guide line about our project

REFRENCES

[1] Bergles, A.E. Techniques to augment heat transfer, Handbook of Heat Transfer Applications (Ed.W.M. Rosenhow), 1985, Ch.3 (McGraw-Hill, New York).
[2] Bergles, A.E., Bunn R.L and Junkhan G.H, Extended performance evaluation criteria for enhanced heat transfer surfaces, Letters in Heat and Mass Transfer, Volume 1, Issue 2, November–December 1974, Pages 113-120.
[3] S. Eiamsa-ard, C. Thianpong, P. Promvonge, Experimental investigation of heat transfer and flow friction in a circular tube fitted with regularly spaced twisted tape

elements, Int. Commun. Heat Mass Transfer 33 (2006) 1225–1233.

[4] Rahimi M, Shabanian S.R. and Alsairafi A.A, Experimental and CFD studies on heat transfer and friction factor characteristics of a tube equipped with modified twisted tape inserts, Chemical Engineering and Processing 48 (2009) 762–770.

