TEST Engineering & Management

January - February 2020
ISSN: 0193 - 4120 Page No. 9695 - 9702

CFD and Experimentation on Heat Transfer Enrichment in Receiver Tube for Parabolic Trough Collector

M. A. Kadam, Dr. D. G. Kumbhar, Dr. A. M. Pawar

M. A. Kadam*, Research Scholar, Department of Mechanical Engineering, Bharati Vidyapeeth (Deemed to be University), College of Engineering, Pune Email: makadam@bvucoep.edu.in

Dr. D. G. Kumbhar, Associate Professor, Department of Mechanical Bharati Vidyapeeth (Deemed to be University), College of Engineering, Pune Email: dgkumbhar@bvucoep.edu.in

Dr. A. M. Pawar, Associate Professor, Department of Mechanical Engineering Bharati Vidyapeeth's College of engineering for Women, Pune. Email: avibvp@gmail.com, avinash.m.pawar@bharatividyapeeth.edu

Article Info
Volume 82
Page Number: 9695 - 9702
Publication Issue:
January-February 2020

Article History
***ArticleReceived**: 18 May 2019*
***Revised**: 14 July 2019*
***Accepted**: 22 December 2019*
***Publication**: 13 February 2020*

Abstract

In this work, the CFD simulation of the water flow through the plain circular tube is done, and the constant heat flux is given on the surface of the tube for predicting the Nusselt's number and friction factor which help in determining the enhancement in heat transfer rate. The results obtained from CFD investigation are compared with the Dittus-Boelter correlations. This will validate the numerical model used here. To perform the CFD analysis first a 3-dimensional geometric model is created, then model is meshed according to the boundary conditions and then simulation is performed.

Keywords; *CFD simulation, receiver tube, enhancement factor.*

I. INTRODUCTION

CFD mainly starts with the creation of geometry and then generating the mesh on that model. To analyse the flow through the tube we have used the ANSYS workbench (18.2) (FLUENT), which tells us how the temperature and velocity varies along the length of the tube. The point of maximum temperature and maximum velocity can be easily obtained. It is obvious that the temperature distribution will be parabolic along the length of the tube and maximum temperature is obtained near the walls of tube. Similarly, velocity distribution will be parabolic along the length of tube and maximum velocity is obtained at a centre of the tube. ANSYS FLUENT helps in giving the actual magnitude of velocity variation along the length of the tube when we assign the proper boundary condition. Similarly, we also got the temperature and pressure contours. ANSYS FLUENT gives the value of wall shear stress which helps in determining the value of friction factor, Nusselt's number and the thermal performance factor, rectification is not possible.

II. LITERATURE SURVEY

Literature survey based on numerical analysis:
Yao et al. [1] performed a numerical simulation of heat transfer performances of solar water heaters fitted with twisted tape inserts for different initial temperatures (ranging from 273K to 313K). The author has performed the numerical analysis by using two twist ratios (i.e., 2.5 and 4). The predicted results match well with experimental data. It has been observed that by using the

twisted tape inserts across the fluid flow, velocity magnitude reduces which makes the temperature field more uniform. Results show that the mean Nusselt number of the solar water heaters with y = 2.5 is 1.07% smaller and y = 4 is 9.29% larger than the normal solar water heater.

Bhattacharyya et al. [2] performed a numerical study of thermal performance factor in a tube equipped with a twisted tape swirl generator. The characteristics of twisted tape include the, the entrance angle (α) and the pitch (H). Numerical analysis has been performed by using transition-SST model for an entrance angle α = 180°, 160° and 140° with Reynolds number varying between 100 and 20,000. The computational results are in good agreement with experimental data. The results show that as an entrance angle goes on increasing the heat transfer value also increases. As compared to the conventional twisted tape insert, the use of single twist twisted tape insert supplies considerable increase on heat transfer and pressure.

Literature survey based on experimental analysis:

Eiamsa-ard et al. [3] done experimental investigation for thermal performance factor in a tube equipped with delta-winglet twisted tape, using water as a working fluid. Two types of arrangements are used, i.e. Oblique delta-winglet twisted tape (O-DWT) and straight delta-winglet twisted tape (S-DWT). The influences of theses twisted tapes on the heat transfer rate are also described. The characteristics of twisted tape involve three twist ratios (y/w = 3, 4 and 5) and three depths of wing cut ratios (d/w = 0.11, 0.21 and 0.32) over a Reynolds number range of 3000–27,000. Bhuiya et al. [4] performed an experimental investigation of heat transfer rate and fluid-friction characteristics inside a heat exchanger tube equipped with double counter type twisted tape. Experiments are performed using air as a working fluid inside a circular tube turbulent flow regime with Reynolds number varies between the range 6950 to 50050. Double-counter type twisted tapes consist of four different twist ratios (y = 1.95, 3.85, 5.92 and 7.75). Results show that heat transfer rate increases by decreasing the twist ratios.Literature survey based on experimental and numerical analysis: Shinde et al. [5] performed CFD & experimental studies on heat transfer, friction factor and thermal performance factor inside a tube equipped with drilled twisted tape inserts having various hole geometries. The twisted tapes are made in three different twist ratio's (y) of (2.5, 3.33 & 5) and pitch value 45mm, 60 mm & 90 mm. CFD simulation is performed on plain tube and classic, almond, triangular & rhombic cut twisted tape inserts. After CFD investigation, it has been observed that Nusselt's number rises for Almond cut, Triangular cut, Rhombic cut twisted tapes when compared to plain tube by 41%, 48.94%, 54% respectively.

Patil et al. [6] performed the experimental and CFD analysis of heat transfer and pressure drop inside a tube equipped with twisted tape having drills of 5 mm diameter & 8 mm diameter. Further twisted tapes are made in three different twist ratio's (y) of 2.5, 3.33, 5. The pitch value for twisted tape is kept 45 mm, 60 mm & 90 mm. After CFD investigation, it is observed that Nusselt's number is increasing by 42.36 %, 52.82% and 48.92% as compared to plain tube.

III. EXPERIMENTAL SET UP AND APPARATUS

Introduction

In this chapter CAD model of an experimental setup was developed in E-Draw CAD software version 3.2. Figure no. 2 shows the CAD model of experimental setup which shows the nomenclature of the components used in the fabrication of experimental setup. This chapter also deals with the experimental study of the apparatus used in setup with its detail specification and the experimental procedure used in conduction of an experiments. This chapter

TEST Engineering & Management

January - February 2020
ISSN: 0193 - 4120 Page No. 9695 - 9702

also consists of procedure used in the manufacturing of a twisted tape inserts and vortex generators. Experimental calculations used in readings are also discussed below.

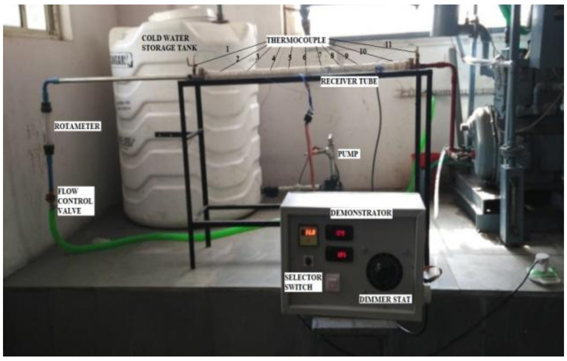


Figure 1: Actual Model of an Experimental Setup

The above figure shows the actual experimental set up, the experiments are carried out at an atmospheric temperature in the IC engine lab. Experimental set up consist of the following equipment, i.e., cold water storage tank, pump, flow control valve, rotameter, receiver tube, thermocouples, demonstrator. Water is used as fluid medium. The water flows from the cold-water storage tank to the receiver tube by means of pipelines. Rotameter is attached between the cold-water storage tank and receiver tube which is used to control the fluid flow rate into the receiver tube. Fluid flow rate is controlled by means of flow control valve provided at the entry section. Heater is mounted on the walls of receiver tube due to which cold water gets heated. Nine thermocouples are mounted on tube to measure the surface temperature of the tube.

IV. CFD INVESTIGATION AND VALIDATION

CFD (Computational Fluid Dynamics) is a superior numerical tool available to engineers to solve the complex equation of fluid mechanics and heat transfer which can capture the actual physics of the real-life problems. Fluid flows are mainly iterated by partial Differential Equations (PDE) which represents the conservation laws for the governing equations available i.e. mass, momentum and energy equations. Computational Fluid Dynamics is used to replace such PDE system by the governing equations which can be solved using Computer Systems. The essential chief behind performing CFD operation is that simulated flow region which is divided into number of smaller cells [6].

1 CFD Analysis of Plain Circular Tube

In this chapter, the CFD simulation of the water flow through the plain circular tube is done, and the constant heat flux is given on the surface of the tube for predicting the Nusselt's number and friction factor which help in determining the enhancement in heat transfer rate. The results obtained from CFD investigation are compared with the Dittus-Boelter correlations. This will validate the numerical model used here. To perform the CFD analysis first a 3-dimensional geometric model is created, then model is meshed according to the boundary conditions and then simulation is performed.

2 Defining the Material Properties

Since water is taken as a working fluid during experimentation, therefore it is obvious to choose water as a working fluid for the simulation purpose. Water is passing from the tube. Therefore, under the material section the fluid to be chosen is water-liquid. The properties of water at 30°C are taken for the simulation purpose. Table 1 shows properties of water at 30°C.

Table 1: Properties of water at 30°C

Sr. No.	Properties	Value
1	Density	995.71 Kg/m^3
2	Specific Heat Capacity	4178 J/kgK
3	Thermal Conductivity	0.6 W/mK
4	Viscosity	0.000798kg/ms

3 Boundary Conditions

The boundary conditions for the simulation are given according to the parameters measured

Published by: The Mattingley Publishing Co., Inc.

January - February 2020
ISSN: 0193 - 4120 Page No. 9695 - 9702

during the experiment of circular tube without insert. Water is used as a working fluid therefore the boundary conditions will be set by providing the velocity inlet, inlet temperature, pressure outlet, outlet temperature and the constant heat flux on the wall of the tube. Water at 30°C is flowing from the tube whose flow rate varies from 2 lpm to 10 lpm. Hence the inlet velocity varies from 0.1176 m/s to 0.5878 m/s. The outlet pressure selected is an atmospheric pressure hence a zero-gauge pressure. The backflow outlet temperature is taken same as an inlet temperature. A constant heat flux of4020.7564 W/m^2 is given on the wall of the tube. Turbulent Intensity and Hydraulic diameter are also specified for an initial guess. Hydraulic diameter set here is a tube diameter of 19 mm and the turbulent intensity is 5%.The wall of the tube was considered to have a zero roughness with no slip condition.

Table 2: Readings for a plain circular tube using Standard k-€ model

Q (lpm)	*V* (m/s)	*Re*	τ_w(N/m^2)	*Vout*(m/s)	*f*	*Nu*
2	0.1176	2787.99	0.1005	0.1243	0.05226	32.25926
4	0.2351	5573.6	0.2906	0.2427	0.039638	48.91379
6	0.3527	8361.6	0.5742	0.361	0.0354	65.53565
8	0.4703	11149.58	0.937	0.4795	0.032743	80.82772
10	0.5878	13935.2	1.3696	0.5981	0.030761	94.90712

4Sample CFD Calculations

For Plain Circular Tube

Flow Rate = 2 lit/min Hydraulic Diameter = 0.019 m, Radius of Tube = 0.0095 m Density of Fluid = 995.71 kg/m^3, Viscosity = 0.000798 kg/ms

- Mass Flow Rate Q = 2 x 10^{-3}/60 = 3.3333 x 10^{-5} m^3/sec
- Cross Sectional Area= 2.8352 x 10^{-4} m^2
- Velocity of the Fluid
  V = Q/A = 0.1176 m/sec
- Reynolds Number
  $$\frac{\rho v D_H}{\mu} = \frac{995.71 \times 0.1176 \times 0.019}{0.000798} = 2787.9$$

From CFD Results, wall shear stress is obtained. For 2 lpm,

- Wall Shear Stress = 0.1005 N/m^2
- Friction Factor
  $$f = \frac{8 \times \tau_w}{\rho v^2} = \frac{8 \times 0.1005}{995.71 \times 0.1243^2} = 0.05226$$

- Nusselt's number

$$Nu = 0.125 x_f \times R_e \times P_r \frac{1}{3} = 32.25926$$

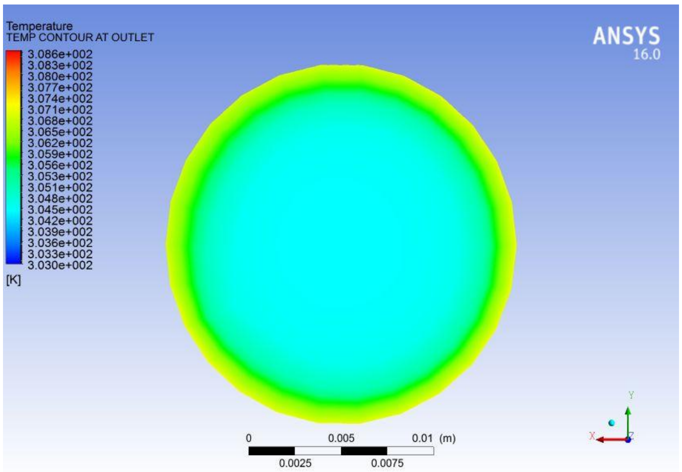


Figure 2.Temperature contour for plain circular tube using standard k-€ model at outlet location for flow rate of 2 lpm

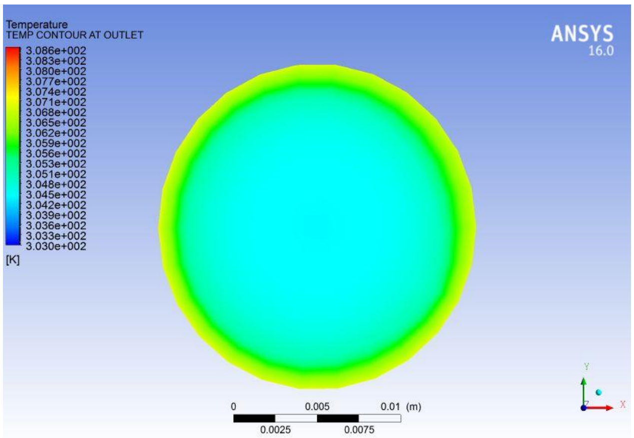
CFD readings for a plain circular tube using SST k-$\boldsymbol{\omega}$ model

Published by: The Mattingley Publishing Co., Inc.

January - February 2020
ISSN: 0193 - 4120 Page No. 9695 - 9702

Table 3: Readings for a plain circular tube using SST k-ω model

Q (lpm)	V (m/s)	Re	τ_w(N/m^2)	Vout(m/s)	f	Nu
2	0.1176	2787.99	0.09272	0.124	0.048449	29.90616
4	0.2351	5573.6	0.2743	0.2431	0.037292	46.01836
6	0.3527	8361.6	0.5387	0.3622	0.032992	61.07717
8	0.4703	11149.58	0.8786	0.4811	0.030498	75.28673
10	0.5878	13935.2	1.2884	0.5999	0.028764	88.74536



Figures 3.Temperature & velocity contour for plain circular tube using SST k-ω model at outlet location for flow rate of 2 lpm

5 CFD plots for a plain circular tube

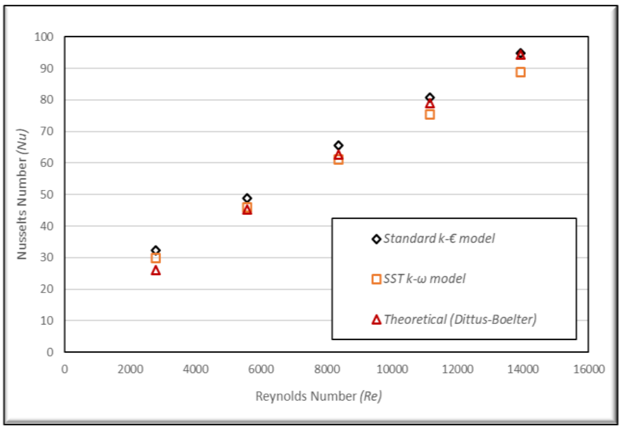


Figure 4. Plot of Nu v/s Re for plain circular tube using different turbulence models

The above figure 4 shows the variation of Nusselt's number v/s Reynolds number for a plain circular tube using the SST k-ω, Standard k-€ model and Dittus-Boelter correlation (Theoretical). The graph shows that as Reynolds number goes on increasing the value of Nusselt's number also increases. The reason for this maybe the increase in eddies because of high velocity which causes development of swirl flow.

V EXPERIMENTAL RESULTS AND DISCUSSIONS

The heat transfer and fluid flow characteristics can be analyzed by determining Reynolds number, heat transfer between tube and fluid, Convective heat transfer coefficient, Nusselt number, friction factor and enhancement efficiency. Input parameters like the mass flow rate and heat flux are based on the literature review. The calculated parameters at different flow rates are shown below in tables. The result tables are as follows:

Table no.4. Experimental result of plain circular tube

Q (lpm)	Qw(J/sec)	Hw (W/m^2°C)	Dh mm of Hg	Re	Nu	f
2	340.9248	734.3473	2	2787.99	23.254	0.05391
4	397.7456	1366.098	7	5573.6	43.259	0.047211
6	454.5664	1789.532	12	8361.6	56.668	0.03596

Published by: The Mattingley Publishing Co., Inc.

January - February 2020
ISSN: 0193 - 4120 Page No. 9695 - 9702

8	568.208	2373.237	19	11149.5	75.152	0.032023
10	625.0288	2935.864	25	13935.2	92.969	0.026973

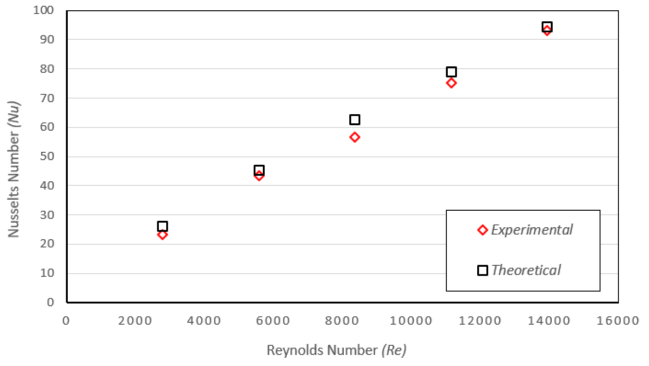


Figure 5. Comparison of plain circular tube readings with Dittus-Boelter Correlation

The above graph shows the variation of Nusselt's number v/s Reynolds number for the plain tube using Experimental and Dittus-Boelter equation, the readings obtained by experiment and Dittus-Boelter equation is ±12. 04% for the Nusselt's number. The result tells the accuracy of experimental model and techniques used for measurements, thus validating the present experimental setup.

Validation of plain circular tube

Validation is performed for experimental results of plain receiver tube with the CFD results. CFD & Experimental readings for a plain circular tube.

Table 5. Readings for a plain circular tube using CFD & Experimental analysis

Q (lpm)	*Re*	*Nu (Theo)*	*f (Theo)*	*Nu (CFD)*	*f (CFD)*	*Nu (Expt)*	*f (Expt)*
2	2787.99	26.0548	0.04670	29.906	0.04844	23.25	0.0539
4	5573.6	45.3485	0.03734	46.018	0.03729	43.25	0.0472
6	8361.6	62.7316	0.03312	61.077	0.03299	56.66	0.0359
8	11149.58	78.9699	0.03054	75.286	0.03049	75.15	0.0320
10	13935.2	94.3942	0.02874	88.745	0.02876	92.96	0.0269

Table 5, shows the readings for the plain circular tube using CFD and Experimental analysis. The value of friction factor is calculated based on wall shear-stress and the value of Nusselt's number is determined from the Chilton-Colburn analogy used for the turbulent flow inside the tube.

CFD & Experimental plots for a plain circular tube, The figure 6 shows the variation of Nusselt's number v/s Reynolds number for a plain circular tube using the CFD & Experimental analysis, and Dittus-Boelter correlation (Theoretical). The graph shows that the value of Nusselt's number goes on increasing as the Reynolds number increases. The reason for this maybe the increase in eddies because of high velocity which causes development of swirl flow.

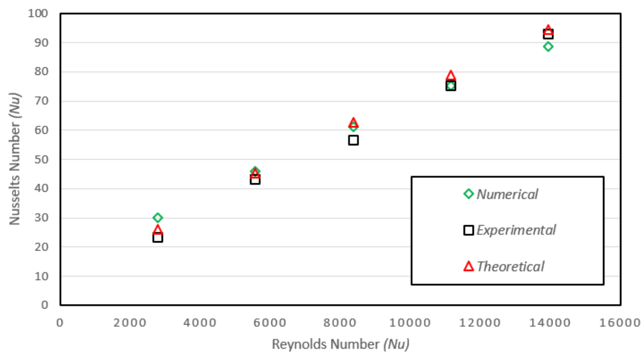


Figure 6. Plot of Nu v/s Re of plain circular tube using CFD & Experimental analysis, and Dittus-Boelter correlation (Theoretical)

Published by: The Mattingley Publishing Co., Inc.

January - February 2020
ISSN: 0193 - 4120 Page No. 9695 - 9702

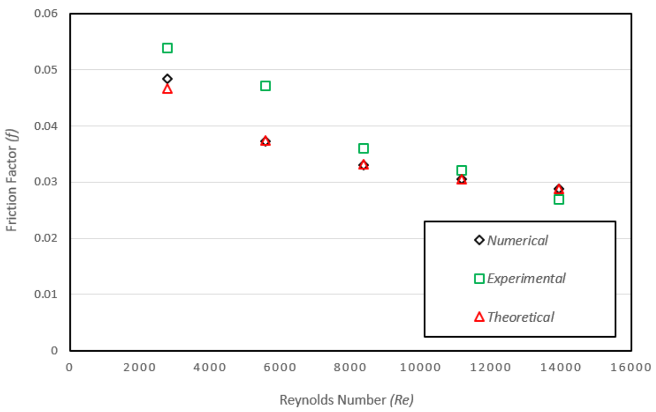


Figure 7. Plot of f v/s Re of plain circular tube using CFD & Experimental analysis, and Dittus-Boelter correlation (Theoretical)

The above figure 7 shows the variation of Friction factor v/s Reynolds number for a plain circular tube using the CFD & Experimental analysis, and Dittus-Boelter correlation (Theoretical). The graph shows that at lower Reynolds number the value of friction factor is higher at, i.e. for the lesser flow rate.

VI CONCLUSION

In this manner, the results obtained from CFD & Experimental analysis for a plain circular tube are compared with Dittus-Boelter equation & it was found that Nusselt's number varies ±14.78% with the numerical readings and Nusselt's number varies ±12.04% with the experimental readings. Hence the numerical model used here is validated.

REFERENCES

1. Keguang Yao, Tong Li, Hanzhong Tao, et al. "Performance evaluation of all-glass evacuated tube solar water heater with twist tape inserts using CFD." Energy Procedia 70 (2015) 332 – 339
2. Oni, T. O., and Paul, M. C. "Numerical investigation of heat transfer and fluid flow of water through a circular tube induced with divers' tape inserts."Applied Thermal Engineering, 98(2016), pp. 157-168.
3. S. Eiamsa-erd, K. Wongcharee, P. Eiamsa-erd, C Thianpong, "Heat transfer enhancement in a tube using delta-winglet twisted tape inserts."Applied Thermal Engineering 30 (2010) 310-318.
4. M.M.K. Bhuiya, A.S.M. Sayem, M. Islam, M.S.U. Chowdhury, M. Shahabuddin, "Performance assessment in a heat exchanger tube fitted with double counter twisted tape inserts, International Communications in Heat and Mass Transfer", Volume 50, 2014, Pages 2533, ISSN 0735-1933.
5. Tushar R. Shinde, Prof. D.G. Kumbhar, "A Review on Experimental and CFD Analysis for Heat Transfer Enhancement in Heat Exchanger Tube using Drilled Twisted Tape Inserts of various Geometries" International Research Journal of Engineering and Technology (IRJET) Volume: 04, Issue 06 June 2017.
6. Sagar V. Patil, Prof. D.G. Kumbhar, "CFD and Experimental Studies of Heat Transfer and Pressure Drop in a Tube Equipped with Drilled Twisted Tape Inserts",International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE) Volume 1, Issue 2 June2016.
7. M.M.K. Bhuiya, M.S.U. Chowdhury, M. Saha, M.T. Islam, Heat transfer and friction factor characteristics in turbulent flow through a tube fitted with perforated twisted tape inserts, International Communications in Heat and Mass Transfer, Volume 46, 2013, Pages 49-57, ISSN 0735-1933.
8. M.M.K. Bhuiya, A.S.M. Sayem, M. Islam, M.S.U. Chowdhury, M. Shahabuddin, "Performance assessment in a heat exchanger tube fitted with double counter twisted tape inserts, International Communications in Heat and Mass Transfer", Volume 50, 2014, Pages 2533, ISSN 0735-1933.
9. Kumbhar D. G, Sane N. K, Exploring Heat Transfer and Friction Factor Performance of a Dimpled Tube Equipped with Regularly Spaced Twisted Tape Inserts, Procedia Engineering, Volume127, 2015, ISSN 1142 – 1149.
10. Yong-zhang CUI, Mao-cheng TIAN, Three-dimensional numerical simulation of thermal-hydraulic performance of a circular tube with edgefold-twisted-tape inserts, Journal of Hydrodynamics, Ser. B, Volume 22, Issue 5, 2010, Pages 662-670, ISSN 1001-6058.

Published by: The Mattingley Publishing Co., Inc.

TEST Engineering & Management

January - February 2020
ISSN: 0193 - 4120 Page No. 9695 - 9702

AUTHORS PROFILE

M.A.Kadam is Research Scholar is in MechanicalEngineering Department, Bharati Vidyapeeth (Deemedto be University), Pune, India, He completedMaster ofEngineering in Mechanical Engineering in BharatiVidyapeeth (Deemed to be University), Pune, India

Dr. D.G.Kumbhar is currently working as a Associate professor in Mechanical Engineering Department, Bharati Vidyapeeth (Deemed to be University), Pune, India, He earned his Ph.Ddegree in Mechanical Engineering from Bharati Vidyapeeth (Deemed to be University),Pune, India.

Dr. A. M. Pawar is currently working as a Associate professor in Mechanical Engineering Department, Bharati Vidyapeeth's College of engineering for Women, Pune, India, He has completed his Ph.D. in Mechanical Engineering fromBharati Vidyapeeth (Deemed to be University),Pune, India.