International Journal of Innovative Technology and Exploring Engineering (IJITEE)
ISSN: 2278-3075 (Online), Volume-9 Issue-4, February 2020

Heat Exchangers Modeling using Industrial Wind Tunnel Simulation for Automobile Industry

Pradnya More, Subhash Gadhave

Abstract*: In the industry 4.0 eras, it is necessary to analyze each parameter precisely. In case of heat transfer studies, the automobile and aerodynamic industry is gaining more and more attention. Many heat exchangers are designed and tested in industry but for any specific product thermal characteristics can be different which depends on the scenario of use of that product. Heat exchangers can test with fluid, air etc. For internal machines like bearing cooling fluid can be used but in case of automobile, aerodynamics air flow is important element. This paper presents the simulation and analytical modeling of various heat exchangers. This can be very useful for lowering the insulation damages or foil damages of vehicle. The wind tunnel test is carried with constant wind flow to assure that vehicle speed and cooling can be moderate without issue of cooling.

Keywords: CFD analysis, heat transfer, heat exchanger, wind tunnel test, aerodynamic analysis.

I. INTRODUCTION

1. Introduction

Heat exchanger [1, 2,3] is an unit through which energy is directed from one material to some other utilizing a solid surface area. Exchanger studies and model accordingly consist of together convection and conduction [4]. Radiative transmission among the exchanger and the destination can quite often be derelict except the exchanger is un-insulated as well as , its outer areas are extremely hot [5]. Couple of significant challenges in heat exchanger evaluation is (1) evaluation of known heat exchangers and (ii) dimensions of heat exchangers for an individual application. Evaluation includes persistence of the level of temperatures balance transfer, the modification in heat of the portions of special materials, and the force drop throughout the heat exchanger [6]. Dimensions includes assortment of a particular heat exchanger right from some of those presently in existence or pinpointing the sizes for the development of an innovative heat exchanger, specified the necessary level of heat transfer and allowed force drop.

Revised Manuscript Received on February 28, 2020.
***** Correspondence Author
Pradnya More*, PhD Scholar, Department of Mechanical Engineering, JJT Name University , Rajasthan, India. Email: trivenimore216@gmail.com
Subhash Gadhave, Associate Professor, Department of Mechanical Engineering, Dr. D. Y. Patil Institute of Technology Pimpri, India.. Email: Subhash.gadhave534@gmail.com

To be able to examine heat exchangers it is very important to present a few types of categorization. Certainly, there are several techniques which usually are considered. The initial views the circulation setup through the heat exchanger, whereas the other is determined by the distinction of devices category principally by structure. For wind tunnel test, proposed research considers air flow through wind tunnel with constant velocity and we can test four types of air flows as counter flow, concurrent flow, cross flow, hybrids such as cross counter flow and multi pass flow. In automobile industries, wind tunnel test is important as aerodynamics and thermodynamics need to be considered. The small change in air flow velocity can impact on heat dissipation hence; this paper presents the circular pipe, multi-pipe and fin type heat exchanger analysis. The analytical model is developed and simulation of wind tunnel is tested.

II. LITERATURE REVIEW

Shell and Tube heat exchangers are established in various power and operation products. The typical benefits in successful heat transfer, however , highly larger force loses around the group of pipes. The consequence is better fluid driving power need which in turn takes off at the rear sections of many of these heat exchangers, as an example of power generation etc. This kind of task suggests an innovative layout of the pipes as a longitudinal sinusoidal wave that decreases the force drop need whilst preserving competent heat transfer elements [7]. A 3d computational fluid dynamics (CFD) analysis of the chaffing and heat transfer elements of a fin pipe heat exchanger keeping elliptical pipes of distinct percentages and direction of these kinds of pipes to the lateral are considered during this unique research. With reference to figure 1, air in the transition strategy is regarded as the operating fluid in this kind of present research.

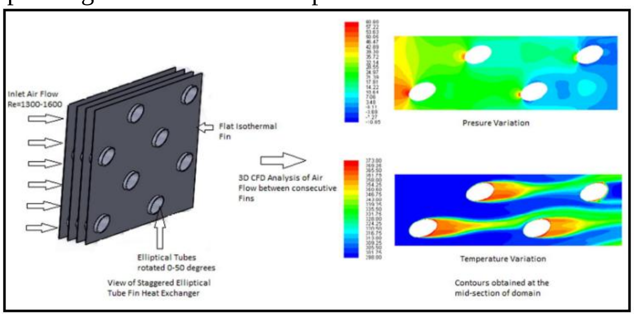

Figure 1: Fin tube heat exchanger (Yogesh, Siddhant Singh, et al, 2018)

The ellipticity percentage of the pipes implemented intended for the investigation traces among 0.5 and 0.7 and so the consequence of rotation of these kind of pipes on Colburn variable, chaffing variable as well as , productivity index is actually analyzed. The Colburn variable boosted primarily hitting a maximum, subsequent to that it takes decreased with boost in pipe pattern. Boost in ellipticity and Reynold's figure lessened the Colburn variable. Chaffing variable deviates in direct ratio to pipe pattern and as well, ellipticity percentage, where it cuts back with the gain in Reynold's count. The productivity index variable for many of the designs is considered [8].

A unique analysis was executed to analyze the airside heat and hydraulic capabilities among a small bare pipe heat exchanger, which comprises of 480 bare pipes by an exterior diameter of 0.5 mm. Its efficiency is examined with a louvered-fin and flat-tube heat exchanger by way of the identical anterior area, using the equivalent evaluation quality matrix. The impact of air flow ranges and fluids flow ranges on final heat transfer capability and force drop are mentioned. From the trial and error outcomes, it was eventually determined that reviewed with the standard heat exchanger, this bare pipe heat exchanger can reach the matching temperatures potential with 70% less swaddle level and 65% less materials quantity [9]. Through this work, trial and error as well as, mathematical investigation was carried out to identify the temperatures change and force drop elements of corrugated pipes. Measuring and simulation are carried out for 3 types of corrugated pipes. The equivalent basic pipes were simulated at the equal time. In the mathematical simulation the circulation is designed as getting symmetric and thoroughly elliptic by utilizing specific size strategy and the Low-Re k-ε rapid unit. The consistent temperature flux boundary predicament is stipulated to mimic the utility temperature utilizing in the studies. The heat transfer capabilities of the corrugated pipes are reviewed with ordinary pipes within 3 limits [10].

This paper signifies a 3-d mathematical simulation to anticipate heat transfer and force drop of turbulent fluid air-flow within, inner internally star multi-finned pipes. The acquired outcomes show that together heat transfer level and chaffing component of star multi-finned pipes are noticeably larger as opposed to the ordinary pipe. Optimum amount of Nu and f for case equipped with curly star fins are 500 and 780% large than the ordinary pipe. Thermal routines of all explored conditions are larger than unity and as well, it accelerates by boost of Reynolds number [11]. Among the most frequently implemented heat exchangers in the chemical sector and energy production is the Shell and Tube Heat Exchanger (STHE), that comprise of a cylindrical enclosure that have multiple cylindrical pipes throughout, in which one particular fluid passes using the intrinsic pipes whereas the additional encompasses it. In addition, the separations transmit the circulation from the outer shell to improve its getaway with the aid of the heat exchanger and strengthen the transmission approach [12].

Geometry and functioning variables of cross-flow air-water heat exchanger with small route coils, hooked up in an opened circuit wind tunnel and employed in treatment plan research, had been carried out in mathematical exploration to be able to confirm the numerical model. 3 dimensional models with air and water liquid flow and heat transfer fields is applied, as it presents further specific outcomes contrasted to models that consider steady heat or steady temperature fluxes on the tube wall surfaces. Formulated model consisted total span of air as well as, water flows in the heat exchanger [13].

The objective of this kind of task is to analyze the efficiency of a multi-directional wind tower included with vertically specified heat transfer units applying Computational Fluid Dynamics (CFD) and wind tunnel assessment [14,15]. The in scale model is analyzed in a regular flow closed-loop wind tunnel to verify the CFD statistics. Mathematical outcomes of the source circulation are investigated with trial and error statistics. Procedure assessment was practiced utilizing a low-speed closed-loop wind tunnel. The acceleration of the ventilation within the route of the wind tower was deliberated by using a hot wire anemometer, due to its immediate impulse moments and potential of gauging poor air speed, which used to be present at the time of this analysis [16].

III. HEAT TRANSFER ANALYTICS

During the last decade, different versions of heat transfer assessments tend to be applied to check heat exchangers, and so the purpose of offering innovative methodology is to assure the fact that, although the improvement in the heat transfer level, the force of the tunnel outlet continues as at a practical capacity. Research suggest the fact that for most temperatures control models and so the comparable force drop, the alteration proportion of heat transfer variable will be larger compared to the tunnel converter. The motive of the tunnel is to strengthen the air gap and to maximize the time frame of the ventilation. In this assessment, it was tested out to simulate and evaluate the heat transfer of air flow circulation surrounding the pipes. Apart from this, to design heat exchanger proposed study considered three types of designs: circular single pipe, multi-pipe structure and fins structure. The aim is to identify the best suited heat exchanger model for automobile utilization. The specification of material used during experimental simulation is shown in table 1below.

Table 1: Material Properties

Material	Density	Young's Modulus	Poison's ratio
300M Steel	7.88 g/cc	193E+07	0.27

In Figure 2 the wind tunnel inlet and outlets are presented. The pipe transporting heated air is customized in a corresponding length approach. The air space among every pipe is equivalent to examine the heat transfer ranges.

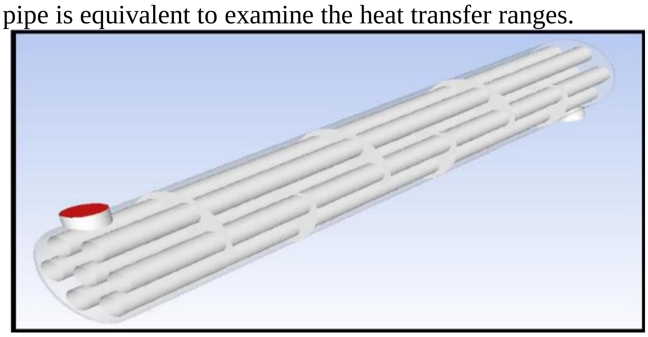

Figure 2: Wind tunnel heat exchanger test

The experimental simulation design consists of wind tunnel with one inlet and one outlet for air circulation, 300M steel material pipes, and the compressor for controlling constant velocity air flow. Heat exchanger is designed with circular pipes and intended to test as single pipe and multiple pipe structure as shown in figure 3. The diameter of pipe kept constant for both tests. The fin type heat exchanger design is also tested for 300M steel material and simulation results are noted.

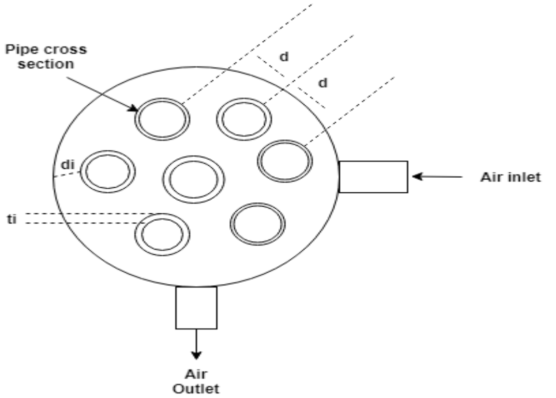

Figure 3: Representation of two-dimensional circular pipes

Due to the thin wall of pipe, the temperature gradient across the thickness is small and the pipe temperature varies only in the x and y-directions. From the pipe tip temperature gradient we can identify the heat transfer rate and in case of multi-pipe structure the heat transfer can be equivalent if material used for all pipes is same. For proposed modeling we used 300 M Steel materials for all pipe structure. Also, the heat transfer coefficient alongside of the pipes used as heat exchangers was with varying temperature. Hence, the heat transfer coefficient (Htc) in the proposed study considered as a variable element. The "Htc" for single pipe heat exchanger region is identified with a step size for heat rise is 50mm (Refer table 2).

To identify the steady state variables, further we developed the heat conduction equation. From heat conduction analysis we identified variables for circular single pipe heat exchanger, circular multi-pipe heat exchanger and fin type heat exchanger with wind tunnel test. The velocity of air through wind tunnel is kept constant. The analytical model for circular single pipe heat exchanger, circular multi-pipe heat exchanger and fin type heat exchanger are developed as following:

P_{conv} is convection temperature gradient for pipe with constant diameter, $W_{velocity}$ is constant wind tunnel air velocity, $T_{initial}$ is initial temperature of pipe and T resultant temperature of hot spot. So, we developed the generalized equation as shown in Eq. (1) below.

$$Htc\frac{\partial T}{\partial t} = P_{conv}W_{velocity}(T(t) - T_{initial}) - \phi P_{conv}W_{velocity}(T(t) - T_{initial}) \tag{1}$$

$\phi = 0$ is temperature gradient for constant heat dissipation

$\phi = 1$ is temperature gradient for heat transfer with step size 50

$$W_{initial} = \lambda_{initial} \times P_d \times d_x$$

Where $\lambda_{initial}$ is yield factor, P_d is diameter of pipe and d_x is hot spot location in mm

Hence, the convection heat gradient can be calculated as per Eq. (2) for temperature step size 0 to 50 centigrade

$$P_{conv} = (1 - \textstyle\sum_0^{50} \lambda_i) P_d\, d_x \tag{2}$$

Now, to calculate exact hot spot formation temperature, consider Young's Modulus "Y" as 193E+07 and Poison's ratio "Pr" 7.88 g/cc for 300M steel.

$$T(hotspot) = P_d d_x P_r \left(\frac{P_{conv(x_1 ..., x_5)}}{Y} - \frac{P_{conv}\phi}{Y}\right) Htc\frac{\partial T}{\partial t}(T(t) - T_{initial}) \tag{3}$$

Where

Functions $P_{conv}(x_1 ...., . x_5)$ and $Htc\frac{\partial T}{\partial t}(x_1 ...., . x_5)$ depend on dimensions of pipe/fins

In case of fins as a heat exchanger we need to use rectangular surface area instead of circular pipe dimensions.

For rectangular fin 'F' heat transfer coefficient,

$$Htc\frac{\partial T}{\partial t} = F_{conv}\left(1 - \textstyle\sum_{i=1}^{50} P_r Y\right) + \textstyle\sum_{i=1}^{50} W_{conv}\lambda_{initial} \tag{4}$$

IV. RESULT AND ANALYSIS

Inlet air flow for cooling is shown in figure 2 below along with hot air impression near the pipes and outlet. The sequence of heat element exchange can be seen as a progressive transfer of heat. Accordingly, the heat exchanger design is analyzed in this section. (See figure 4)

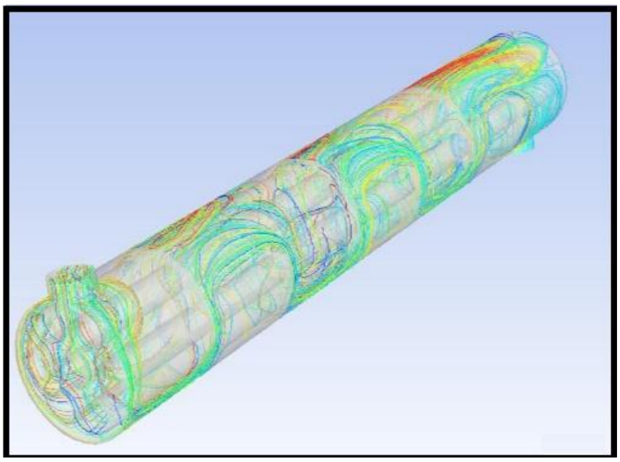

Figure 4: Air flow cooling with wind tunnel test with heat element impression

In case of wind tunnel test, the inner layer of wind tunnel with jets and outer body of subject (pipes to be tested) are crucial elements. The type of material used for pipes is also important. For proposed study, we used 300M Steel with 7.88 g/cc density. It is noted that the heat transfer rate and dissipation factor both are different in terms of occupied area of heating. Hear transfer element is a point of action where heat transfer begins and heat dissipation factor is the time taken by heat element to reach at further heat transfer element or spot.

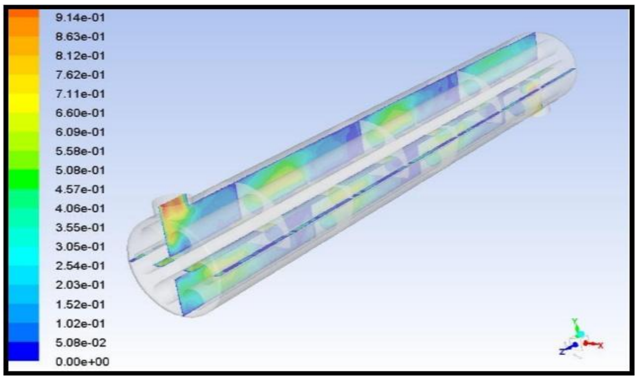

Figure 5: Heat mapping cross-section view

In short, heat dissipation is the travel of heat wave from one heat transfer element to other. The heat wave directions are modeled in four sections shown in Figure 5 above. Each section shows the level of heat element and heat dissipation ratio. The heat exchanger is shown as a slice around the pipe. Equivalent distance is maintained between each section of wind tunnel. The pilot test is conducted with and without circular pipe heat exchanger using wind tunnel test. The results are represented in figure 6 below.

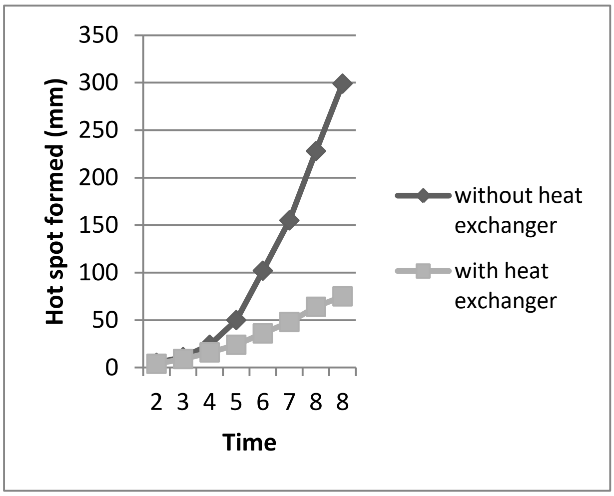

Figure 6: Pilot heat exchanger test result

The result with heat exchanger shows that, the hot spot development is delayed whereas in case of test without heat exchanger hot spot development recorded at early time frame. Further proposed research carried the wind tunnel test for circular pipe shape heat exchanger and the recorded results shown in table 2 below.

Table 2: Hot spot formation without and with single pipe heat exchanger using wind tunnel test

Hot spot Location from pipe end (mm)	Hot spot (mm) without heat exchanger (centigrade)	Average hotspot formation without heat exchanger	hotspot formation (mm) with heat exchanger	Average hotspot formation (mm) with heat exchanger
100	1.3	1.5	0.2	0.28
	1.3		0.2	
	1.5		0.3	
	1.6		0.3	
	1.8		0.4	
150	2.1	2.45	0.4	0.64
	2.3		0.4	
	2.5		0.5	
	2.6		0.6	
	2.6		0.7	
200	2.6	2.82	0.9	1.02
	2.6		1.0	
	2.8		1.0	
	3.0		1.1	
	3.1		1.1	
250	3.4	4.18	1.3	1.42
	3.8		1.3	
	4.1		1.5	
	4.7		1.5	
	4.9		1.5	
300	5.0	5.88	1.7	1.94
	5.5		1.7	
	5.8		1.8	
	6.2		2.1	
	6.9		2.4	

Based on reading depicted in table 2 above, the chart is represented for performance analysis of wind tunnel single circular pipe heat exchanger test with and without heat exchanger as shown in figure 7 below.

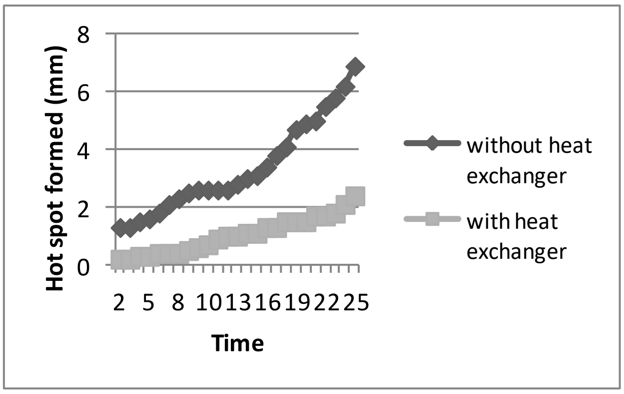

Figure 7: Single circular pipe heat exchanger test result

Similarly, the test is carried out for circular multi-pipe structure and graphical structure is shown in figure 8. The main subject pipe is kept in all three tests to identify the impact of hot spot formation.

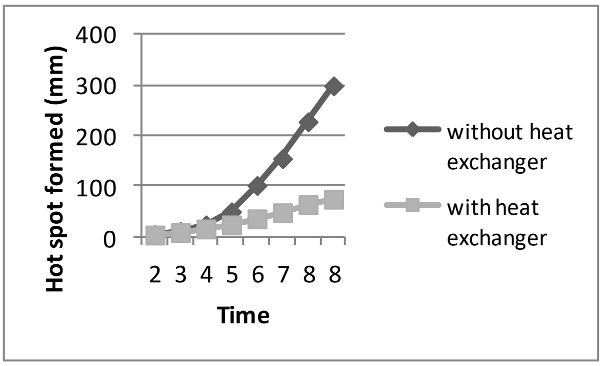

Figure 8: Circular multi-pipe heat exchanger test result

Further, in case of ‘fin’ as a heat exchanger for subject pipe, the analysis is carried out keeping wind velocity constant. The results of comparison are shown in figure 9 below.

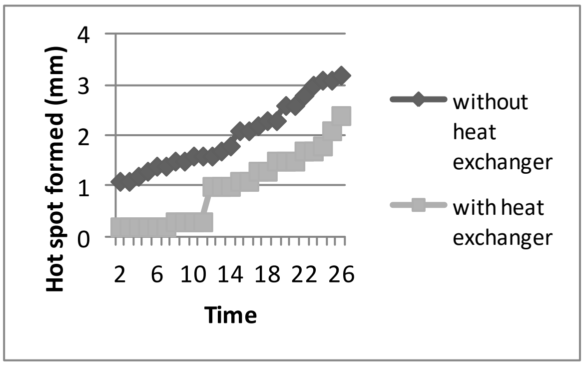

Figure 9: Fin type heat exchanger test result

From the analysis for subject i.e. 300M steep circular pipe, it is concluded that the single pipe, multi-pipe or fin type heat exchangers are effective to delay or avoid hot spot formation.

V. Conclusion

This research proposed simulation and analytical modeling of heat transfer coefficient with different shape heat exchangers. The analysis is carried out for testing of subject material 300M steel with and without heat exchanger and constant wind tunnel air velocity. The wind tunnel test proved that the constant velocity can control temperature gradient by using appropriate heat exchanger material and shape. This will ultimately beneficial for automobile industry. For automobile speed of vehicle can be considered as a wind tunnel velocity. But, instead on wind velocity it is advisable to use efficient heat exchanger.

REFERENCES:

1. Bahiraei, Mehdi, et al. "Recent research contributions concerning use of nanofluids in heat exchangers: a critical review." *Applied Thermal Engineering* 133 (2018): 137-159.
2. Sadeghianjahromi, Ali, Saeid Kheradmand, and Hossain Nemati. "Developed correlations for heat transfer and flow friction characteristics of louvered finned tube heat exchangers." *International Journal of Thermal Sciences* 129 (2018): 135-144.
3. Shirvan, Kamel Milani, Mojtaba Mamourian, and Javad Abolfazli Esfahani. "Experimental investigation on thermal performance and economic analysis of cosine wave tube structure in a shell and tube heat exchanger." *Energy conversion and management* 175 (2018): 86-98.
4. Mirgolbabaei, Hessam. "Numerical investigation of vertical helically coiled tube heat exchangers thermal performance." *Applied Thermal Engineering* 136 (2018): 252-259.
5. Chimres, Nares, Chi-Chuan Wang, and Somchai Wongwises. "Effect of elliptical winglet on the air-side performance offin-and-tube heat exchanger." *International Journal of Heat and Mass Transfer* 123 (2018): 583-599.
6. Jabbour, Joseph, et al. "High performance finned-tube heat exchangers based on filled polymer." *Applied Thermal Engineering* 155 (2019): 620-630.
7. Hussein, Afnan Abdulkareem. An Optimum Sinusoidal Arrangement of Tubes in Shelland-Tube Heat Exchangers with Good Heat Transfer Characteristics Combined with Improved Pressure Drop Requirements. Kuwait university-college of graduate studies, 2018.
8. Yogesh, Siddhant Singh, et al. "Heat transfer and pressure drop characteristics of inclined elliptical fin tube heat exchanger of varying ellipticity ratio using CFD code." International Journal of Heat and Mass Transfer 119 (2018): 26-39.
9. Huang, Zhiwei, et al. "Airside Heat Transfer and Friction Characteristics of a 0.8 mm Diameter Bare Tube Heat Exchanger." Heat Transfer Engineering (2019): 1-11.
10. Sun, Ming, and Min Zeng. "Investigation on turbulent flow and heat transfer characteristics and technical economy of corrugated tube." Applied Thermal Engineering 129 (2018): 1-11.
11. Khanmohammadi, Fahimeh, Mousa Farhadi, and A. Ali Rabienataj Darzi. "Numerical investigation of heat transfer and fluid flow characteristics inside tube with internally star fins." Heat and Mass Transfer 55.7 (2019): 1901-1911.
12. Ruiz, Leonardo Delgado, Carlos Acevedo Peñaloza, and Guillermo Valencia Ochoa. "Effect of the Mass Flow Rate on the Heat Transfer Phenomena in a Shell and Tube Heat Exchanger." *International Journal of Applied Engineering Research* 13.14 (2018): 11387-11391.
13. Glazar, Vladimir, et al. "Numerical analysis of heat transfer in air-water heat exchanger with microchannel coil." *E3S Web of Conferences*. Vol. 95. EDP Sciences, 2019.
14. Calautit, John Kaiser, et al. "Numerical and experimental analysis of a multi-directional wind tower integrated with vertically-arranged heat transfer devices (VHTD)." *Applied energy* 185 (2017): 1120-1135.
15. Sharifi, Khashayar, et al. "Computational fluid dynamics (CFD) technique to study the effects of helical wire inserts on heat transfer and pressure drop in a double pipe heat exchanger." *Applied Thermal Engineering* 128 (2018): 898-910.
16. Singh, Hardeep, et al. *Influence of Secondary Flow Generation on Heat Transfer inside the Fin Type Spiral Sub-Cooled Condenser by Experimental and CFD Analysis*. No. 2018-32-0054. SAE Technical Paper, 2018.

AUTHORS PROFILE

Ms. Pradnya More, PhD Scholar at JJT University, Rajasthan.Mechanical Engineering Department BE , M Tech

Dr. Subhash L. Gadhave, associate professor, mechanical engg. department Lmiste, sae, theired, iaeng, iedrc, Be, me mech (mechatronics, ph.d