Kütahya Dumlupınar University
Institute of Graduate Studies

Journal of Scientific
Reports-A
E-ISSN: 2687-6167
Number 53, June 2023

RESEARCH ARTICLE

THE EFFECTS OF CIRCULAR INSERTS ON THE THERMAL AND FLOW CHARACTERISTICS IN A HORIZONTAL PIPE EXCHANGER: A NUMERICAL INVESTIGATION

Muhammet Kaan YEŞILYURT[1], Nesrin ADIGÜZEL[2*], Orhan YILDIRIM[3], Ömer ÇOMAKLI[4], Faraz AFSHARI[5]

[1]Vocational School of Technical Sciences, Ataturk University, Erzurum,kaan.yesilyurt@atauni.edu.tr
ORCID: 0000-0002-7207-1743
[2]Department of Mechanical Engineering, Ataturk University, Erzurum, nesrin.adiguzel@atauni.edu.tr
ORCID: 0000-0001-7610-2757
[3]Department of Mechanical Engineering, Ataturk University, Erzurum, orhan.yildirim@atauni.edu.tr
ORCID: 0000-0001-8780-1297
[4]Department of Mechanical Engineering, Ataturk University, Erzurum, ocomakli@atauni.edu.tr
ORCID: 0000-0003-4631-7989
[5]Department of Mechanical Engineering, Erzurum Technical University, Erzurum, faraz.afshari@erzurum.edu.tr
ORCID: 0000-0001-9192-5604

Receive Date:11.08.2022 *Accepted Date: 06.03.2023*

ABSTRACT

The aim of the present study was to numerically investigate the effects of circular inserts placed inside a circular tube in order to evaluate the heat transfer characteristics under different operating conditions. Computational Fluid Dynamics methods were used to solve the model, which is a heat pipe with an outer diameter of 21 mm equipped with circular inserts with a distance of 20 cm. Different mass flow rates of the heat transfer fluid, including 25, 50, 75, 100, and 125 g/s were examined, and the thermal behavior of the turbulators and the flow structure were investigated. R19.0 version of ANSYS Fluent software was used as the CFD program to obtain the desired results and contours. From the results, it was found that circular inserts can be used in heat pipes to produce vortices and thus improve the heat transfer.

Keywords: *Vortex generator, Turbulator, Circular Pipe Inserts, Numerical study, ANSYS Fluent, Turbulent flow, Friction factor, Heat Transfer*

1. INTRODUCTION

Whether running on renewables or fossil fuels, i.e., coal, oil, or gas, thermal systems that transport energy from one medium to another are always put into use in numerous applications. Improving the heat exchanger performance and ensuring maximum thermal transmission while also keeping the system as small as possible with high efficiency is of great importance [1]. The efficiency of these systems is primarily determined by their heat transfer characteristics, and several approaches aimed at improving the heat transfer processes have been developed over the last decades. As such, there has been an exponential increase in the number of studies focused on heat transfer enhancement, in parallel with the increasing number of patents and products [2].

Heat transfer enhancement technologies are based on active systems that rely on the use of external power sources and passive systems that do not require external power but rather take advantage of geometrical changes, arrangements, or modifications, rough surfaces, swirling flow devices, etc. [3]. One of the most considered approaches is dealing with the flow regime throughout the system and improving the heat transfer, enabling development of more compact systems, with a smaller room requirement for installation [4], reduced initial cost and lower payback time. In this area, studies involve numerical and experimental methods and mostly employ a circular single tube or pipe as the test section through which lies an insert (which are also called swirl/vortex generator or turbulator) to create turbulent flow [5–13]. Inserted into the pipes or tubes as blades [14], rods [15, 16], rings [17, 18], tapes [8, 19], strips [8, 20] or coils [21, 22], tube inserts enhance heat transfer by providing increased flow mixing and thus creating turbulent flow induced by longitudinal vortices. Among the parameters that affect the enhancement rate induced by the turbulators are their shapes, geometry, placement, spacing or pitch, flow attack angle, material, effective diameter, Reynolds number, and further modifications such as perforations, wings, etc. [1]. Surveying the literature, numerous studies are available on the improvement of heat transfer using inserts, turbulators, and swirl/vortex generators. Some involve only numerical/analytical and some only experimental studies, whereas others involve both numerical and experimental studies in a comparative and confirmative manner. In a study by Tatsumi *et al.* [23], 3D unsteady flow was analyzed in a square channel with rough walls, in the cases where full-span ribs or discrete ribs were attached in sets to the wall of the channel. The flow and associated heat transfer in the channel were reported to be 3D even in the case of full-span ribs when the ratio of rib height to channel height was small. Large-scale downstream eddies that rotate in counter directions were reported to have accompanied the flow through the ribs and thus have improved flow mixing in the case of discrete ribs, resulting in higher wall heat transfer.

In a study aimed at investigating the effects of conical ring inserts, placed as having 10, 20 and 30 mm spacing, on the performance characteristics through entropy generation minimization and efficiency improvements. For this purpose, maximum entropy generation was determined at same Re number for each configuration. In terms of entropy generation, conical ring inserts were thermodynamically advantageous up to Re 8000. The efficiency increased as the pitch decreased from 0.86 to 1.16 [24]. Salman *et al.* [25] carried out a comparative numerical investigation of the effects of V-cut and classical twisted tape inserts with different twist ratios on heat transfer and friction factor. The enhancement factor was reported to have been found to be positively related to the Re number and inversely related to the twist ratio for both the classical and V-cut twisted tape inserts. Furthermore, the V-cut twisted tape provided a better surface for fluid contact and hence a greater heat transfer coefficient compared to right-left helical tape inserts. The overall maximum heat transfer enhancement was reported to be 107%.

In order to experimentally investigate the effect of coiled wire inserts on the occurrence of dynamic instabilities in a straight tube for forced convection boiling test tube, experiments were carried out in a circular tube with and without the inserts using water as the working fluid at different inlet temperatures and the results were compared. Circularly coiled wire inserts with two different pitch ratios were tested under the conditions that a constant temperature was applied to the outer surface of the test tube and a constant outlet restriction was used at mass flow rates ranging from 110 to 20 g/s. As a result, density waves and pressure drops were reported to have occurred at each configuration and the oscillations occurred at all inlet temperatures [26].

In another study, a comprehensive literature review was presented to investigate heat transfer and flow structure in air ducts. It has been found that perforation in ribs/chambers/blocks and a combined rib and delta fin combination resulted in better thermo-hydraulic performance [27]. Using an uncommon insert material and geometry, Razzaghi *et al.* [28] investigated overall heat transfer along 10 rows of staggered bundle of elliptic tubes with changeable transverse and longitudinal pitches and aluminum foam porous media inserts. Despite increased pressure drop at higher Reynolds numbers, the results suggested that the use of aluminum foams has brought on a significant improvement in the heat transfer and overall efficiency while flow regime remained laminar. An array of conical rings having different pitches were employed in a horizontal tube as tube inserts by Yeşilyurt [29] in a forced-convection boiling two-phase flow system in order to investigate their effects on the flow instabilities as well as the heat transfer characteristics under constant pressure, constant inlet temperature, constant heat flux and output restriction conditions. The characteristic curve of the flow system was reported to have shifted to the right with the smallest increase in the input thermal power, resulting in increased pressure drops at any mass flow rate. The hydraulic and thermal performance of a water-to-air heat exchanger, where air passed through the outer pipes and water flowed through inner pipe, was investigated in the presence of perforated and typical helical fins at different pitch ratios and Re number. Using NSGA II and ANSYS Fluent14, thermal performance, Darcy friction factor and Nu number were obtained using proposed empirical equations [30].

Karagoz *et al.* [14] used cylindrical inserts with different attack angles and pitches (101, 216, 340) to create turbulent flow in the heat exchanger tube at different flow rates and Reynolds numbers (6000, 11,000, 17,000) and investigated the effects of Nusselt number, Reynolds number and friction factor on the heat transfer rate. The inserts in the heat exchanger tube were reported to have led to a significant increase in Nu number and energy efficiency. Zheng *et al.* [15] numerically studied the effect of rod type swirl generators placed in a heat exchanger tube on the thermal and the hydraulic performance of the system and showed that multiple vortices were induced by the introduction of inserts. The slope of the rod, diameter ratio and Re were considered to affect the heat transfer rate and friction factor significantly. Artificial neural networks were also used to supports a multi-objective optimization model in order to ensure maximum heat transfer, minimum pressure drop, and an optimal Pareto front. Applying a thermal-hydraulic performance evaluation criterion, the greatest improvement in the heat transfer were achieved with the 0.058 diameter and 57.057° inclined vortex rod at 426.767 Re number. Li *et al.* [31] investigated the turbulent flow and heat transfer through a tube equipped with drainage inserts. By diverting the fluid running at the central section of the pipe towards the wall, the drainage inserts were reported to improve the mixing of cold and hot fluid as well as create vortices that agitate the fluid. The results obtained from the numerical analyses as well as their confirmation with the experimental data revealed that a 3.3 pitch ratio and a 45° inclination angle of the insert were reported to be the optimum arrangement to yield the greatest performance evaluation criteria, hence better heat transfer and flow performance. The thermal and hydraulic properties of nanofluid flow in a turbulent forced convection regime in a circular tube equipped with cone-shaped inserts were analyzed with numerical studies using the finite volume method.

Karuppasamy *et al.* [32] used Al_2O_3 and CuO whereas Mohammed *et al.* [10] used Al_2O_3, CuO, SiO_2 and ZnO particles of various diameter at different volume fractions in water as the carrier fluid. While Karuppasamy *et al.* [32] achieved the highest improvement with Al_2O_3 Mohammed *et al.* [10] reported highest increase in the Nu number and friction factor with SiO_2. In a study, helical-surface disc inserts were examined for their effects on the pressure drop and heat transfer characteristics in a double-tube heat exchanger (DTHE) where hot water flowed through the inner pipe, and cold air flowed inside the annulus. Having tested different values of spacing, helix angle (ϕ) and diameter ratio (DR), the best improvement in the overall heat transfer, Nu and friction factor was reported with DR = 0.42, which was the smallest of all tested, and helix angle of $\phi = 40°$, which was the greatest of all tested [33].

In another study, heat transfer and flow characteristics of mist/steam refrigerant in a U-shaped gas turbine were investigated and the mass ratio and mist to steam as well as the mist diameter were carefully analyzed. With the increase in the mist-to-steam mass ratio, the heat transfer from the ribbed wall has increased proportionally. When the mist diameter was 10μm, the mass ratio of the mist increased from 2% to 10%, whereas the increase in the average Nu number in the first and second passages were 60.13% and 112.5%, respectively [34]. In another study, the turbulent thermal behavior of CuO-distilled HO-based nanomaterials resulting from the placement of a new turbulator was modeled. Four different cases with different geometries, namely straight pipe, twisted band, barrier twisted band, and perforated barrier twisted band, were considered and a two-phase mixing model was applied. The intensification of turbulence by the addition of barriers led to a remarkable improvement in the heat transfer and an increase in the pressure drop was noted due to placement of barrier on the twisted tapes [35]. Recent technological advances have led to significant advances towards more compact and efficient heating.

The development of novel heat transfer improvement techniques remained a hot topic in the global research. Over several decades, various active and passive heat transfer improvement techniques were developed for convective heat transfer. While passive methods are favorable for their simplicity and not needing external energy consumption, active methods enable the modification and deterioration of the thermal boundary layer. Such a technique is the use of tube inserts that create fluid mixing and secondary flows, diverting and/or rerouting the flow stream [20].

The convective heat transfer in an oval channel exposed to a constant heat flow from the bottom was modeled as two-phase water-Al_2O_3 mixture nanofluid and the effects of Re number, nanofluid volume ratio and different combinations of co-conical inward inserts (CCI-in), counter-conical inserts (CoCI) and co-conical outward inserts (CCI-out) on the flow pattern and heat transfer characteristics were investigated. The highest heat transfer coefficient was observed with the CCI-inward inserts, by approximately 17% higher than the plain tube, owing to the induction of secondary flows [20]. In another study, heat transfer characteristics and flow properties of a mist-steam two-phase flow were numerically investigated and experimentally validated in different channels with different void ratios (ε) and rib designs. With ε greater than 0.264, the thermal recovery performance obtained with column-row-ribs was better than with solid ones. On the other hand, the increase in the mist droplet diameter was reported to worsen thermal enhancement factor and increased instabilities in the heat transfer [36]. In another study, the effect of spherical troughs on exergy efficiency and thermos-hydraulic performance of a MWCNT-Al_2O_3/water hybrid parabolic solar collector was investigated through the Nu number, pressure loss and exergy performance. Results showed that the groove height was associated with higher heat transfer, higher exergy efficiency whereas pressure loss was associated with higher Re number and φ [37]. The effects of conical and fusiform inserts in a DTHE with circular and rectangular tubes on heat transfer and turbulent flow patterns were investigated using 21 different configurations at Re 4000, 7000, 10000 and 13000. The highest convection coefficient was reached when the inner tube of the DTHE was circular [38]. The effects of water-to-water aluminum oxide addition in a six-toothed disc turbulator double-pipe heat exchanger were numerically investigated; Nanoparticles with 1%, 4% and 6% concentrations were added to the hot liquid and flowed in a tube with a Reynolds range of 3000 to 13000. At Reynolds 500, cold water flows through the shell and a six-toothed disc turbulator is located in the shell. The increase in the local Nu number by about 70%, hence the improved heat transfer, induced by the collision of the fluid with the surface of the inserts and due to the breaking of the boundary layer resulted in increased the thermal efficiency [39]. The effects of ring turbulators on the flow instabilities were also investigated in another study on an experimental forced-convection two-phase flow system featuring a straight horizontal tube. The effects of a series of rings in variable fluid inlet flow rate were investigated, keeping the pressure, inlet temperature, and thermal power constant. In addition, the wall temperatures and pressure-flow changes of the pipes were also investigated and it was found that at a given mass flow rate, the pressure increased with thermal power at any mass flow rate [18]. In a study, Shivamallaiah and Fernandes [13] numerically investigated the friction characteristics of air flowing through a circular pipe with semi-elliptical inserts at varying aspect ratios, flow attack angles and longitudinal pitches in the range of Re 8000–26000 turbulent flow regime to simulate the heat transfer along the pipe. The presence of semielliptical inserts was reported to have a remarkable effect on the friction characteristics and the Nu number by generating strong longitudinal eddies that intensify fluid mixing in the vicinity of tube wall. While attach angle, pitch and aspect ratio, all had significant effects on friction, aspect ratio's effect on heat transfer was more significant. The enhancement in Nu was 2.1 folds whereas the greatest friction increase was 6.34 folds at 30 mm pitch.

Analyzing available studies in the literature indicated the importance of turbulators and tube inserts as well as CFD simulation. In the present study, the effects of ring inserts, placed inside a circular tube with an interval of 20cm, on the heat transfer along the tube was investigated. The originality of this work is proposing the CFD methods to solve engineering problems in the field of heat exchangers. In this context, the study by Yeşilyurt [29] on the effects tube inserts consisting of conically wound spring arrays on the enhancement of heat transfer and flow instabilities was considered and model simulation of the problem was performed. Further, the present study is based on CFD numerical calculations and the main results of the current study flow structure and temperature distribution contours were obtained and discussed in detail.

2. PHYSICAL MODEL and GEOMETRIC CONFIGURATION

As a common test section used in several other studies in the literature, a circular pipe was used in the test setup in this study. The dimensions of the pipe are the same as in Yeşilyurt [29] but the length of the test section was limited to 500 mm. The inner diameter, outer diameter and wall thickness of the pipe are 17 mm, 21 mm, and 4 mm, respectively. Ring inserts made of 13 mm diameter aluminum wire attached on two sides to two straight aluminum rods at an interval of 20 cm were designed and modeled as the turbulators to be employed in the pipe (Figure 1).

JSR A
Journal of Scientific Reports

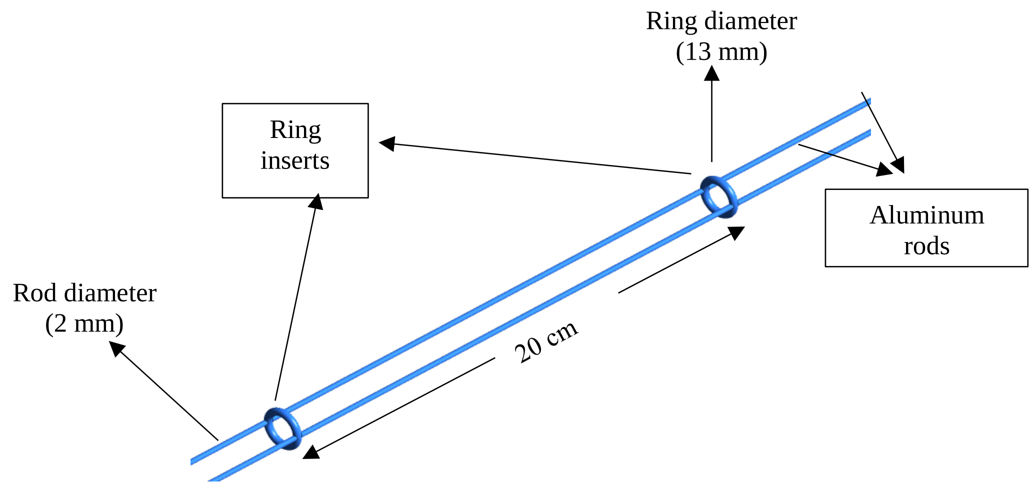

Figure 1. The arrangement and geometric details of the ring turbulators modeled to be employed in the study.

3. COMPUTATIONAL FLUID DYNAMICS ANALYSIS

In the CFD method, several numerical calculations, such as finite elements, finite volumes and finite differences, can be used to solve problems numerically and determine flow characteristics, heat and mass transfer and make predictions about the solutions of real-life problems. CFD solves the conservation equations by making use of equation sets prepared according to control volumes. ANSYS Fluent, a commercial CFD software, that is one of the most popular choices for numerical analyses, was used to carry out the numerical investigation in this study. The analyses were conducted as per the basic steps given below:

- Creating the physical model to be analyzed,
- Designing and meshing the model with a proper mesh quality,
- Constructing the numerical model based on the mathematical set of equations that describe the flow and setting the boundary conditions,
- Running the solution and iterating through all elements,
- Reviewing the results obtained from the solution with the boundary conditions applied,
- Assessing the results in terms of accuracy and precision and starting over the process making any necessary improvements to the model or mesh structure, if necessary.

In this respect, the physical model was first simulated in ANSYS Fluent. The pipe, the insert and the fluid were defined and other necessary definitions were made at first. Afterwards, the mesh was defined and validated. Having ensured that the mesh covers all required zones and is duly sized with all dimensions correct, the solver was chosen and the boundary conditions of the system were defined. After the preparation of the model, solver was established and the equations that govern the flow were

Yeşilyurt, et al., Journal of Scientific Reports-A, Number 53, 190-209, June 2023

solved iteratively to determine the parameters' values in the flow domain. The number of iterations was about 1000 depending on the required precision and the available time allocated for the solution to converge.

The numerical analysis was first carried out for the plain pipe (without the insert placed in) and the values of parameters were saved and analyzed to be referred to as baseline data. Thus, the second numerical analysis will be able to reveal the effect of introducing the insert into the pipe on the values of parameters. The analyses of the water flow through the 500 mm circular heat pipe, with and without the insert, were carried out at a constant wall heat of 6.5 kW, seeking insight into heat transfer characteristics and friction induced by the ring inserts.

3.1. Governing Equations and Assumptions

In the numerical analyses, the exact solution of real life models may be very time-consuming and unnecessarily complex, as such, as a general approach, some assumptions have been made to enable flow and energy equations to be applied in the model.

In the present study, the working fluid was water, and the assumptions made in the analysis are as follows:

i. constant physical properties of the fluid,
ii. incompressible flow,
iii. steady, three-dimensional and fully-developed flow,
iv. gravity acts on the flow.

The two-equation k–ε model, a widely used turbulence model available in most CFD software, such as Fluent, was used in this study to solve the flow field through our circular pipe. The k–ε model can also explicate history effects, such as convection and diffusion of turbulent energy by the virtue that it represents the turbulent properties of the flow by two extra transport equations: the turbulent kinetic energy, k, and the turbulent dissipation, ε, [40].
The first of the transport equations, k, as the name suggests, determines the energy in the turbulence whereas the second, ε, determines the scale of the turbulence, i.e. the rate of dissipation of turbulent kinetic energy.

Considering the abovementioned assumptions, the continuity equation, conservation of momentum and conservation of energy equations, turbulent kinetic energy equation, and turbulent dissipation energy equations, which are given below in sequential order, have been involved in the computational domain.
The continuity equation is expressed as:

$$\frac{\partial \rho}{\partial t}+\frac{\partial \rho u}{\partial x}+\frac{\partial \rho v}{\partial y}+\frac{\partial \rho w}{\partial z}=0 \tag{1}$$

Momentum equation:

JSR A
Journal of Scientific Reports

$$\frac{\partial \rho}{\partial t} + \vec{\nabla}.(\rho \vec{V}) = \frac{\partial \rho}{\partial t} + \vec{V}.\vec{\nabla}\rho + \rho\vec{\nabla}.\vec{V} = 0 \qquad (2)$$

Energy equation:

$$\rho c_v \frac{dT}{dt} = k\nabla^2 T + \Phi \qquad (3)$$

The turbulence kinetic energy equation, k, is expressed as follows:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left((\mu + \frac{\mu}{\sigma_k})\frac{\partial k}{\partial x_f}\right) + G_k + G_b + \rho\varepsilon - Y_M + S_k \qquad (4)$$

and the dissipation rate of turbulent kinetic energy, ε, is given as:

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left((\mu + \frac{\mu_t}{\sigma_e})\frac{\partial e}{\partial x_j}\right) + C_{1\epsilon}\frac{\varepsilon}{k}(G_k + C_{3\epsilon}G_b) + C_{2\epsilon}\rho\frac{\varepsilon^2}{k} + S_e \qquad (5)$$

where G_k is the turbulent kinetic energy generation due to average velocity magnitudes; G_b is the turbulent kinetic energy generation from buoyancy; Y_M is the effect of increasing or decreasing compressible turbulence on the overall dissipation rate; $C_{1\varepsilon}$, $C_{2\varepsilon}$, $C_{3\varepsilon}$ and C_μ are constants; σ_k and σ_ε are the Prandtl numbers for k and ε, respectively; and S_k and S_e user-defined source terms.
The turbulent viscosity (μ_t) is calculated with Equation 6;

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \qquad (6)$$

JSR A
Journal of Scientific Reports

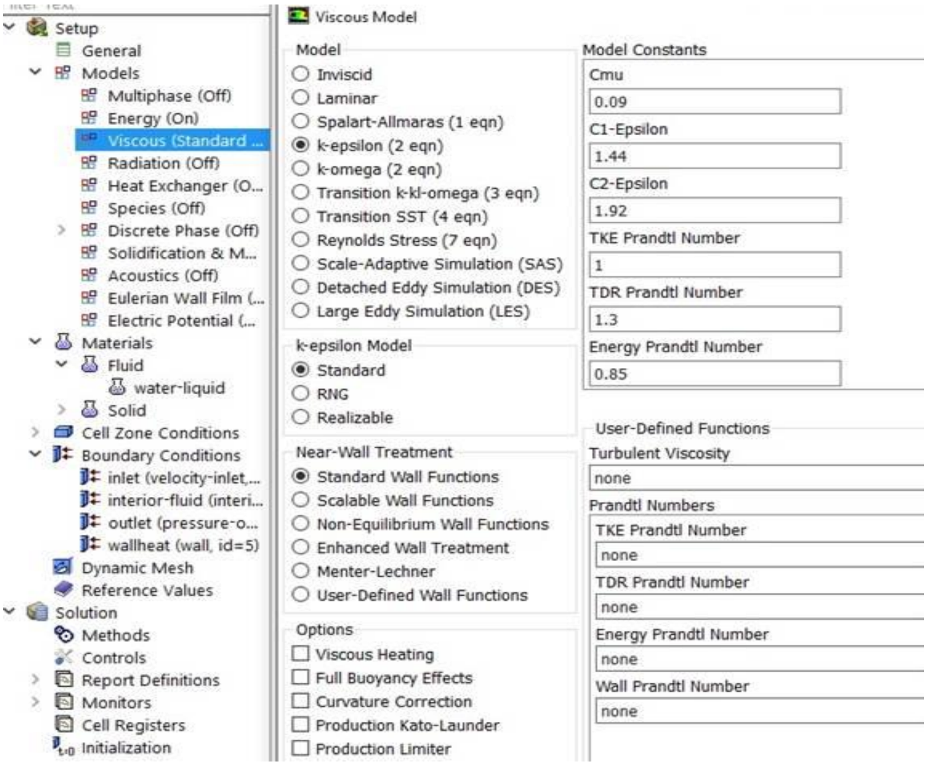

Figure 2. Model selection.

The constants in the model are set as shown in Figure 2. The energy equation used in convection heat and mass transfer in standard k-ε models of turbulent flow is calculated as given in Equation 7;

$$\frac{\partial}{\partial t}(\rho E)+\frac{\partial}{\partial x_i}(u_i(\rho E+\rho))=\frac{\partial}{\partial x_j}\left(k_{eff}\frac{\partial T}{\partial x_j}+u_i(\tau_{ij})_{eff}\right)+S_h \tag{7}$$

where E is the total energy, k_{eff} is the effective heat transfer coefficient, and τ_{ij} is the deviatoric stress tensor.

3.2. Boundary Conditions

The model was designed based on the experimental system used in Adiguzel and Göcücü [18] and Yeşilyurt [29]. The boundary conditions for the numerical analysis to be carried out in ANSYS Fluent were, therefore, determined with reference to the data obtained from the experimental system. As such, the temperature, velocity and pressure values determined during the experiments were used as the boundary conditions in the model. The temperature of water at the inlet was uniform at 292 K and the inlet velocity varied based on the preset mass flow rates. The relative average pressure at the outlet has been defined as zero since it discharges to open air at atmospheric pressure. The Reynolds number was determined by the velocity of water at the inlet, as well as the hydraulic diameter. A constant heat flow rate of 6.5 kW was defined at the pipe wall surface, and a no-slip velocity condition was considered on the pipe wall.

3.3. Meshing and Mesh Validation (Grid Independence Test)

In the 3D analysis, "tetrahexonal" type mesh was used and boundary layer theory was applied by choosing the "infiltration" option on the pipe wall (Figure 3).

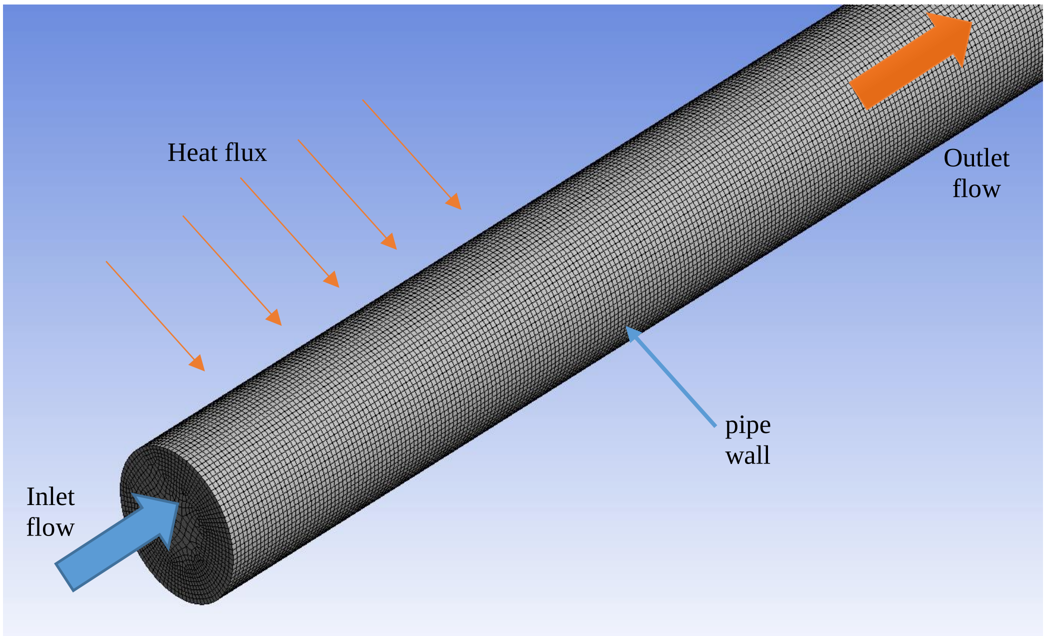

Figure 3. Pipe and mesh structure analyzed.

The mesh structure was created on the determined model and the mesh quality was adjusted. The number of these elements is important to converge to the correct solution. In order to determine the mesh quality, the "skewness" criterion was taken as a reference among the relevant options. Values below the threshold of 0.94 are considered applicable values, and it is generally aimed to further decrease this value.

Excellent	Very good	Good	Acceptable	Bad	Unacceptable
0-0.25	0.25-0.50	0.50-0.80	0.80-0.94	0.95-0.97	0.98-1.00

Figure 4. Mesh skewness spectrum and the scales for the Skewness criterion.

Errors due to mesh structure are a problem that results in an unsuccessful or erroneous analysis. Consideration must be given to cell type, cell count, and mesh quality to ensure the correct solution and obtain reliable results. Therefore, mesh independence tests are necessary to ensure accurate results while also avoiding an excessive mesh number, hence a longer analysis time. First of all, independence from mesh number analyses were performed. After the mesh structure was arranged, the solution part was passed. k-ε turbulence model was selected to solve the problem. Figure 2 shows the operations performed in the solution part. While performing flow and heat analysis, water was chosen as the heat transfer fluid and only the flow region was defined. At this stage, the results of the analyzed model were examined.

4. RESULTS and DISCUSSION

In Figure 5, the geometry of the experimental setup, the inlet and outlet and the heat transfer application have been illustrated in three dimensions in the form of a schematic view of the pipe length. The circular turbulators inside the pipe can be seen in the picture. Circular turbulators have been located at the same distances inside the pipe as shown in the figure.

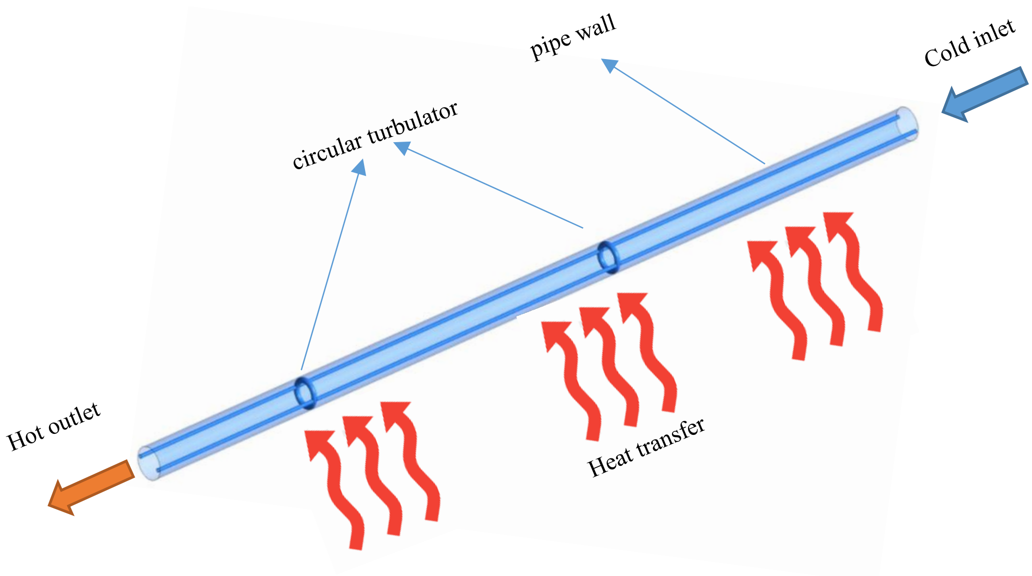

Figure 5. 3-D boundary condition and geometry of the pipe and circular pipe inserts.

In this study, after the mesh independence was ensured for the defined domain, solutions were performed and the results were compared. Various cell configurations and sizes were also performed and no remarkable change was observed. As a result of mesh analysis, the defined geometry was meshed using a maximum cell number of 4,000,000 cells. It should be noted in the meshing process that the defined meshes are well concentrated in the regions close to the wall and circular turbulators in order to get results with the desired accuracy. In the meshing procedure, the skewness value was 0.84 with a growth rate of 1.2. In Figure 6, two views of the mesh configuration have been presented to show the quality of the applied mesh inside the pipe and circular turbulator.

JSR A
Journal of Scientific Reports

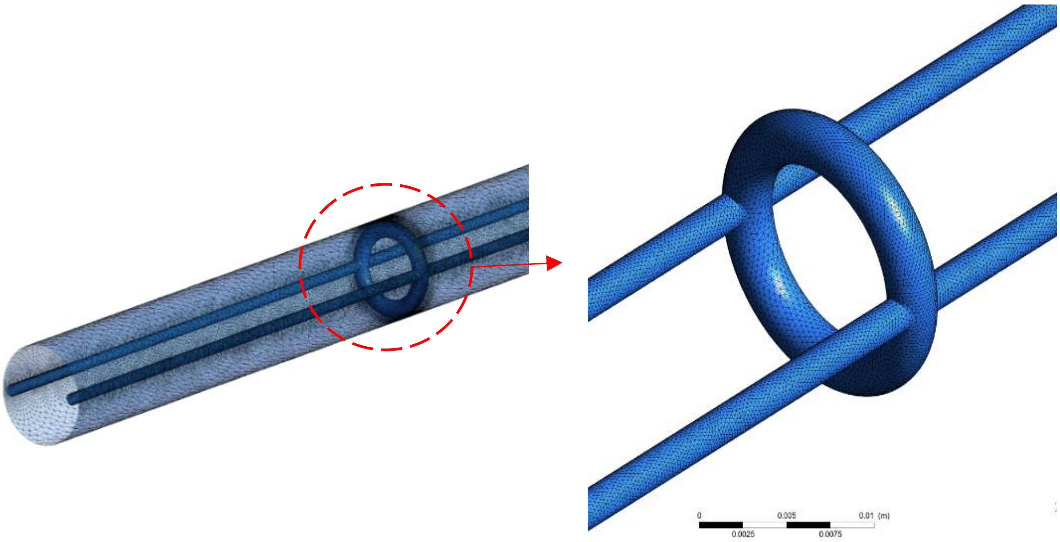

Figure 6. Mesh quality of pipe and circular turbulator.

To ensure the accuracy of the solutions, a simulation was performed for plain pipe (with no turbulators placed in) and the necessary contours were obtained. The velocity and temperature distributions obtained for plain pipe are in good agreement with the existing studies in the literature as shown in the following figures. In Figure 7, the velocity contours for plain pipe have been presented at different mass flow rates from 0.25 g/s to 125 g/s. With increasing mass flow rates, velocity increases gradually. In Figure 8, temperature contours for the plain pipe have also been presented. The effect of mass flow rate on the temperature distribution can be clearly seen in the figure. Moreover, the higher temperature of the regions near the boundary wall can be s in the obtained contours.

JSR A
Journal of Scientific Reports

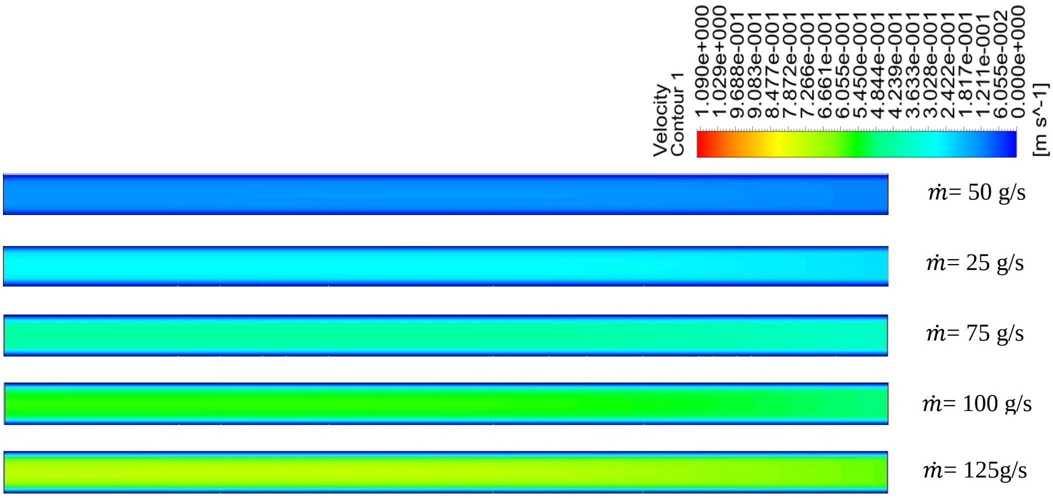

Figure 7. Velocity contours for plain pipe at different mass flow rates.

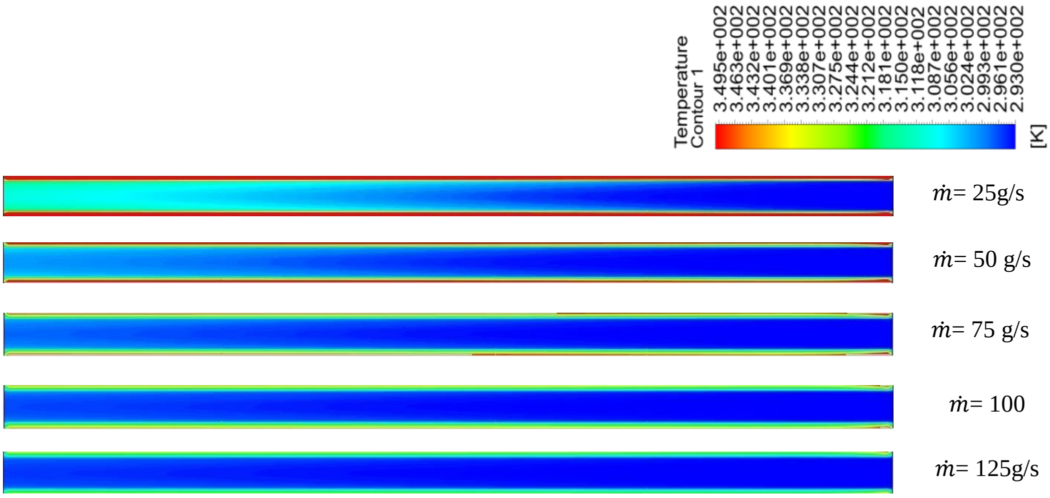

Figure 8. Temperature contours for plain pipe at different mass flow rates.

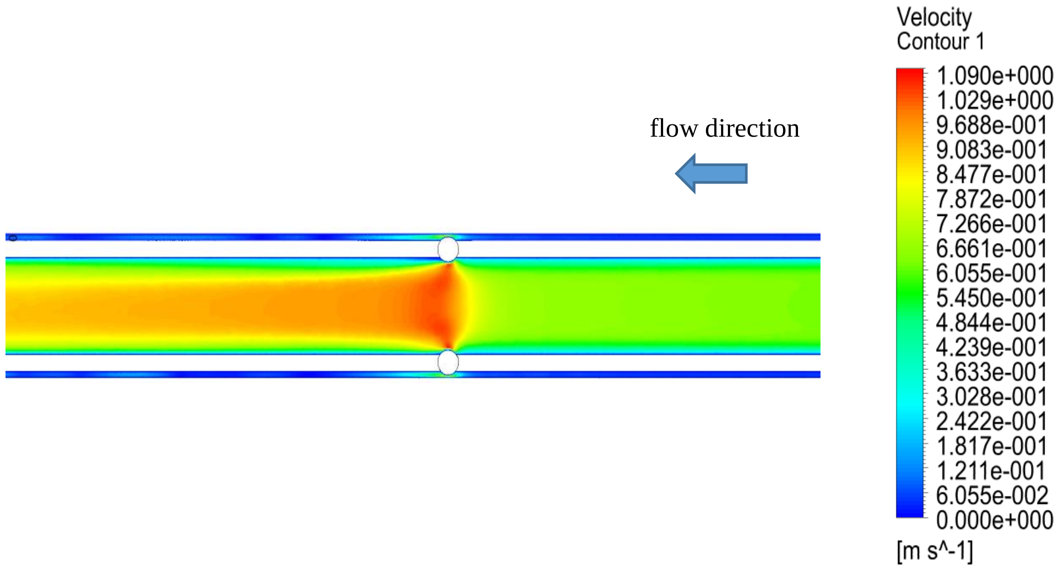

Figure 9. Velocity contour inside pipe at flow rate of 125 g/s.

In Figure 9, the velocity contour of the model has been shown at the fluid flow rate of 125 g/s. The circular turbulators have a favorable effect on disturbing and fluid mixing, and it can be seen that the maximum velocity is in the center of the pipe and the area inside the circular turbulators. The direction of fluid flow has been indicated by an arrow from right to left.

Additionally, velocity contours for pipe inserts (circular turbulators) have been presented in Figure 10 for different fluid flow rates of 25, 50, 75, 100, and 125 g/s. The numerical study was carried out to reveal the influence of installing circular turbulators on flow structure and heat transfer characteristics. It can be observed that Figure 10 demonstrates the fluid flow behavior inside the used pipe from the side view. By increasing the mass flow rate, the velocity variations for all experiments can be compared with those of the presented results.

JSR A
Journal of Scientific Reports

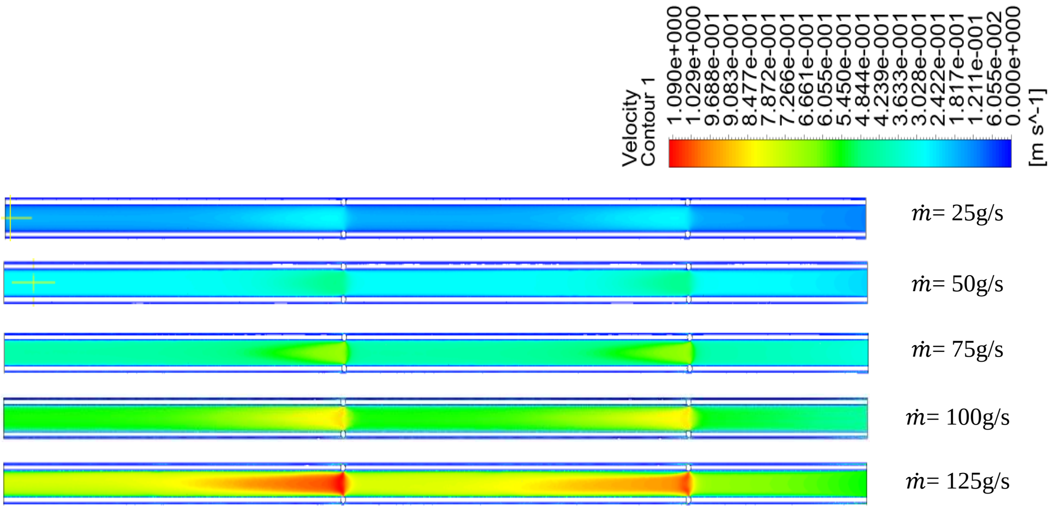

Figure 10. Velocity contours for pipe inserts at different mass flow rates.

To demonstrate the temperature distribution along the pipe equipped with circular turbulators for given mass flow rates, the temperature contours between two turbulators are presented in Figure 11. Temperature variation inside the pipe demonstrates the growth of thermal boundary layers through pipe length and regions around circular turbulators at different fluid mass flow rates. By analyzing temperature contours for different mass flow rates of 25, 50, 75, 100, and 125 g/s, it can be seen that the temperature gradient becomes larger in the boundary layer at lower mass flow rates. The figure also shows the effect of the turbulators on the temperature profile along the pipe.

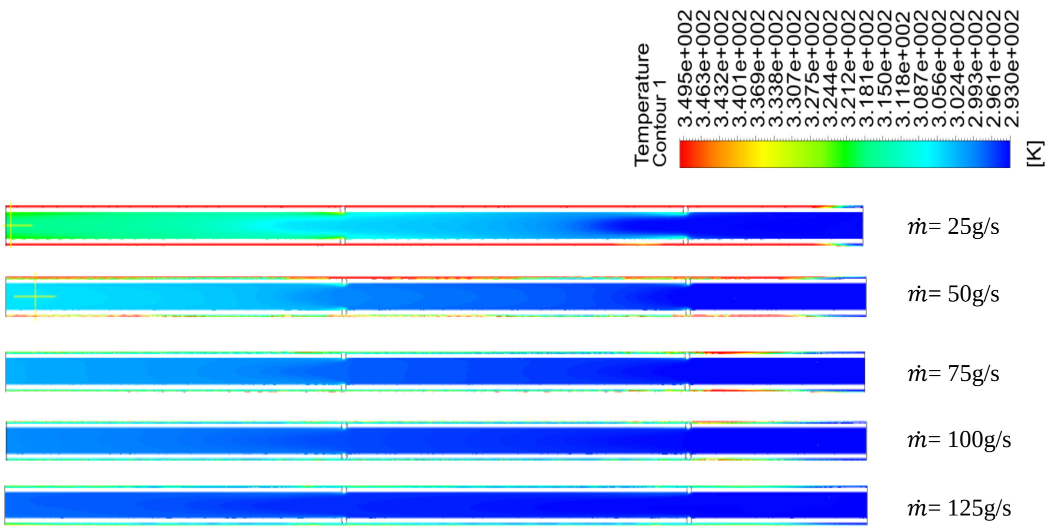

Figure 11. Temperature contours for pipe inserts at different mass flow rates.

5. CONCLUSION

In this study, the numerical results obtained for plain pipe were compared to those obtained for the pipe equipped with circular inserts. Adding the circular pipe inserts into the pipe and producing turbulence flow definitely improved the heat transfer characteristics of the water flow throughout the circular pipe, which received a constant heat flux. In the numerical experiments, different water mass flow rates were examined and by using simulation and visualization techniques, the thermal behavior and structure of the flow were characterized and significant contours were provided.

ACKNOWLEDGMENT

This research has not received any grants.

REFERENCES

[1] Min, C., Li, H., Gao, X., Wang, K., and Xie, L. (2021). Numerical investigation of convective heat transfer enhancement by a combination of vortex generator and in-tube inserts, International Communications in Heat and Mass Transfer, 127, 105490, doi: 10.1016/j.icheatmasstransfer.2021.105490.

[2] Sarada, S. N., Raju, A. V. Sita Rama, and Radha, K. K. (2010). Experimental numerical analysis enhancement of heat transfer in a horizontal circular tube using mesh inserts in turbulent region, European Journal of Mechanical and Environmental Engineering, 2, 3–18. [Online]. Available: http://www.bsmee.be/ejmee/ejmee20102.pdf

[3] Cao, Z., Wu, Z., Luan, H., and Sunden, B. (2017). Numerical study on heat transfer enhancement for laminar flow in a tube with mesh conical frustum inserts, Numerical Heat Transfer, Part A: Applications, 72, 1, 21–39, doi: 10.1080/10407782.2017.1353386.

[4] Mousa, M. H., Miljkovic, N., and Nawaz, K. (2021). Review of heat transfer enhancement techniques for single phase flows, Renewable and Sustainable Energy Reviews, 137, 110566, doi: 10.1016/j.rser.2020.110566.

[5] Jassim, N. A., Abdul Hussin, K., and Abdul Abbass, N. Y. (2017). Numerical investigation of Heat Transfer Enhancement in Circular Tube using Twisted Tape Inserts and Nanotechnology, ejuow, 5, 2, 42–54, doi: 10.31185/ejuow.Vol5.Iss2.57.

[6] Jinxing, W., Chao, W., Mingqiang, W., Yanhui, L., and Yafei, L. (2017). Numerical Simulation of Turbulent Fluid Flow and Heat Transfer in a Circular Tube with Twisted Tape Inserts, Journal of Zhengzhou University (Engineering Science). [Online]. Available: https://en.cnki.com.cn/article_en/cjfdtotal-zzgy201703003.htm

[7] Chamoli, S., Lu, R., Xie, J., and Yu, P. (2018). Numerical study on flow structure and heat transfer in a circular tube integrated with novel anchor shaped inserts, Applıed Thermal Engıneerıng, 135, 304–324, doi: 10.1016/j.applthermaleng.2018.02.052.

[8] Liu, P., Zheng, N., Shan, F., Liu, Z., and Liu, W. (2018). An experimental and numerical study on the laminar heat transfer and flow characteristics of a circular tube fitted with multiple conical strips inserts, Internatıonal Journal of Heat and Mass Transfer, 117, 691–709, doi: 10.1016/j.ijheatmasstransfer.2017.10.035.

[9] Waghole, D. R. (2018). Experimental and numerical investigation on heat transfer augmentation in a circular tube under forced convection with annular differential blockages/inserts (in En;en). Heat Mass Transfer, 54, 6, 1841–1846, doi: 10.1007/s00231-018-2276-8.

[10] Mohammed, H. A., Ali Abuobeida, I. A., Vuthaluru, H. B., and Liu, S. (2019). Two-phase forced convection of nanofluids flow in circular tubes using convergent and divergent conical rings inserts, International Communications in Heat and Mass Transfer, 101, 10–20, doi: 10.1016/j.icheatmasstransfer.2018.12.010.

[11] Nikoozadeh, A., Behzadmehr, A., and Payan, S. (2020). Numerical investigation of turbulent heat transfer enhancement using combined propeller-type turbulator and nanofluid in a circular tube (in En;en). J Therm Anal Calorim, 140, 3, 1029–1044, doi: 10.1007/s10973-019-08578-x.

[12] Liu, H., Zheng, G., Man, C., Jiang, K., and Lv, X. (2021). Numerical and Experimental Studies on Heat Transfer Enhancement in a Circular Tube Inserted with Twisted Tape Inserts, AJEE, 9, 2, 30, doi: 10.11648/j.ajee.20210902.12.

[13] Shivamallaiah, M. M. and Fernandes, D. V. (2021). Numerical investigation of heat transfer and friction factor characteristics of circular tube fitted with an array of semi-elliptical vortex generator inserts, Cogent Engineering, 8, 1, doi: 10.1080/23311916.2021.1968742.

[14] Karagoz, S., Afshari, F., Yildirim, O., and Comakli, O. (2017). Experimental and numerical investigation of the cylindrical blade tube inserts effect on the heat transfer enhancement in the horizontal pipe exchangers, Heat Mass Transfer, 53, 9, 2769–2784, doi: 10.1007/s00231-017-2021-8.

[15] Zheng, N., Liu, P., Wang, X., Shan, F., Liu, Z., and Liu, W. (2017). Numerical simulation and optimization of heat transfer enhancement in a heat exchanger tube fitted with vortex rod inserts, Applied Thermal Engineering, 123, 471–484, doi: 10.1016/j.applthermaleng.2017.05.112.

[16] Raheemah, S. H., Ashham, M. A., and Salman, K. (2019). Numerical investigation on enhancement of heat transfer using rod inserts in single pipe heat exchanger, JMES, 13, 4, 6112–6124, doi: 10.15282/jmes.13.4.2019.24.0480.

[17] Anvari, A. R., Javaherdeh, K., Emami-Meibodi, M., and Rashidi, A. M. (2014). Numerical and experimental investigation of heat transfer behavior in a round tube with the special conical ring

inserts, Energy Conversion and Management, 88, 214–217, doi: 10.1016/j.enconman.2014.08.030.

[18] Adiguzel, N. and Göcücü, A. (2021). Experimental Investigation of the Effects of Ring Turbulators on Heat Transfer in Two-Phase Flow (in En;en). Iran J Sci Technol Trans Mech Eng, 1–10, doi: 10.1007/s40997-021-00472-y.

[19] Outokesh, M., Ajarostaghi, S. S. M., Bozorgzadeh, A., and Sedighi, K. (2020). Numerical evaluation of the effect of utilizing twisted tape with curved profile as a turbulator on heat transfer enhancement in a pipe (in En;en). J Therm Anal Calorim, 140, 3, 1537–1553, doi: 10.1007/s10973-020-09336-0.

[20] Mashayekhi, R., Arasteh, H., Toghraie, D., Motaharpour, S. H., Keshmiri, A., and Afrand, M. (2020). Heat transfer enhancement of Water-Al2O3 nanofluid in an oval channel equipped with two rows of twisted conical strip inserts in various directions: A two-phase approach, Computers & Mathematics with Applications, 79, 8, 2203–2215, doi: 10.1016/j.camwa.2019.10.024.

[21] Agrebi, S., Solano, J. P., Snoussi, A., and ben Brahim, A. (2015). Numerical simulation of convective heat transfer in tube with wire coil inserts in *2015 World Symposium on Mechatronics Engineering & Applied Physics (WSMEAP)*.

[22] Abas, E. F., Weis, M. M., and Ridha, A. S. (2018). Experimental and Numerical Study of Heat Transfer Enhancement in a Shell and Tube Heat Exchanger using Helical Coiled Wire Inserts, Tikrit j. eng. sci., 25, 2, doi: 10.25130/tjes.25.2.10.

[23] Tatsumi, K., Iwai, H., and Inaoka, K. (2002). Numerical simulation for heat and fluid characteristics of square duct with discrete rib turbulators, Internatıonal Journal of Heat and Mass Transfer, 45, 21, 4353–4359, doi: 10.1016/S0017-9310(02)00141-2.

[24] Yakut, K., Sahin, B., and Canbazoglu, S. (2004). Performance and flow-induced vibration characteristics for conical-ring turbulators, Applied Energy, 79, 1, 65–76, doi: 10.1016/j.apenergy.2003.11.002.

[25] Salman, S. D., Kadhum, A. A. H., Takriff, M. S., and Mohamad, A. B. (2013). Numerical investigation of heat transfer and friction factor characteristics in a circular tube fitted with V-cut twisted tape inserts, Scıentıfic World Journal, 2013, 492762, doi: 10.1155/2013/492762.

[26] Omeroglu, G., Comakli, O., Karagoz, S., and Sahin, B. (2013). Experimental Research of Dynamic Instabilities in the Presence of Coiled Wire Inserts on Two-Phase Flow, Scıentıfic World Journal, doi: 10.1155/2013/714180.

[27] Alam, T., Saini, R. P., and Saini, J. S. (2014). Heat and flow characteristics of air heater ducts provided with turbulators—A review, Renewable and Sustainable Energy Reviews, 31, 289–304, doi: 10.1016/j.rser.2013.11.050.

[28] Razzaghi, H., Layeghi, M., Goodarzi, S., and Lotfizadeh, H. (2014). Numerical analysis of the effects of changeable transverse and longitudinal pitches and porous media inserts on heat transfer from an elliptic tube bundle, Journal of Theoretical and Applied Mechanics, Vol. 52 nr 3. [Online]. Available: https://yadda.icm.edu.pl/baztech/element/bwmeta1.element.baztech-f94cf59a-7087-451e-870a-ccd81ae53e56

[29] Yeşilyurt, M. K., Experimental Investigation of the Effects of Conical Springs on In-Pipe Heat Transfer and Flow Instability in Two-Phase Flows. Master Thesis, Atatürk University, 2015.

[30] Sheikholeslami, M. and Ganji, D. D. (2016). Heat transfer enhancement in an air to water heat exchanger with discontinuous helical turbulators; experimental and numerical studies, Energy, 116, 341–352, doi: 10.1016/j.energy.2016.09.120.

[31] Li, P., Liu, P., Liu, Z., and Liu, W. (2017). Experimental and numerical study on the heat transfer and flow performance for the circular tube fitted with drainage inserts, Internatıonal Journal of Heat and Mass Transfer, 107, 686–696, doi: 10.1016/j.ijheatmasstransfer.2016.11.094.

[32] Karuppasamy, M., Saravanan, R., Chandrasekaran, M., and Muthuraman, V. (2020). Numerical exploration of heat transfer in a heat exchanger tube with cone shape inserts and Al2O3 and CuO nanofluids, Materials Today: Proceedings, 21, 940–947, doi: 10.1016/j.matpr.2019.08.242.

[33] Yadav, S. and Sahu, S. K. (2019). Heat transfer augmentation in double pipe water to air counter flow heat exchanger with helical surface disc turbulators, Chemical Engineering and Processing - Process Intensification, 135, 120–132, doi: 10.1016/j.cep.2018.11.018.

[34] Jiang, G., Gao, J., and Shi, X. (2019). Flow and heat transfer characteristics of mist/steam two-phase flow in the U-shaped cooling passage with 60 deg. ribs, International Communications in Heat and Mass Transfer, 105, 73–83, doi: 10.1016/j.icheatmasstransfer.2019.02.023.

[35] Sheikholeslami, M., Abohamzeh, E., Jafaryar, M., Shafee, A., and Babazadeh, H. (2020). CuO nanomaterial two-phase simulation within a tube with enhanced turbulator, Powder Technology, 373, 1–13, doi: 10.1016/j.powtec.2020.06.017.

[36] Jiang, G., Gao, J., Shi, X., Li, F., and Xu, L. (2020). Reprint of: Flow and Heat Transfer Characteristics of the Mist/Steam Two-Phase Flow Cooling the Rectangular Channel with Column-Row-Ribs, Internatıonal Journal of Heat and Mass Transfer, 161, 120236, doi: 10.1016/j.ijheatmasstransfer.2020.120236.

[37] Khetib, Y., Sedraoui, K., Melaibari, A. A., and Alsulami, R. (2021). The numerical investigation of spherical grooves on thermal–hydraulic behavior and exergy efficiency of two-phase hybrid MWCNT-Al2O3/water nanofluid in a parabolic solar collector, Sustainable Energy Technologies and Assessments, 47, 101530, doi: 10.1016/j.seta.2021.101530.

[38] Xiong, Q., Izadi, M., Shokri rad, M., Shehzad, S. A., and Mohammed, H. A. (2021). 3D Numerical Study of Conical and Fusiform Turbulators for Heat Transfer Improvement in a

Double-Pipe Heat Exchanger, Internatıonal Journal of Heat and Mass Transfer, 170, 120995, doi: 10.1016/j.ijheatmasstransfer.2021.120995.

[39] Bashtani, I., Esfahani, J. A., and Kim, K. C. (2021). Effects of water-aluminum oxide nanofluid on double pipe heat exchanger with gear disc turbulators: A numerical investigation, Journal of the Taiwan Institute of Chemical Engineers, 124, 63–74, doi: 10.1016/j.jtice.2021.05.001.

[40] Ajarostaghi, S. S. M., Aghanezhad, M., Davudi, H., and Amiri, M. M. (2021). Numerical evaluation of the heat transfer enhancement in a tube with a curved conical turbulator insert, International Journal of Ambient Energy, 1–14, doi: 10.1080/01430750.2021.1945490.