Engineering Universe for scientific Research and management
ISSN (Online): 2319-3069
Vol. XIV Issue IX
September 2022

CFD ANALYSIS AND PERFORMANCE OF CONCENTRIC PIPE HEAT EXCHANGER WITH VARYING INLET VELOCITIES

Shamsul Arfin [1], Mr. Prakhar Dubey[2] Mr. Sachin Baraskar[3]
Mechanical Department, SSSUTMS University, Sehore, M.P./466001, India[1]
Mechanical Department, SSSUTMS University, Sehore, M.P./466001, India[2]
Mechanical Department, SSSUTMS University, Sehore, M.P./466001, India[3]

Abstract

Heat exchanger is a widely used for transfer of heat energy from one fluid to the other fluid. The heat is transferred in form of conduction and convection. Conduction occurs inside the material whereas convection occurs from materials to fluid and from fluid to fluid. The common application of the heat exchangers are like condenser, cooling tower, intercooler, refrigeration, and many other industrial applications. There are many types of heat exchangers wherein parallel and counter flow heat exchangers are extensively used in parallel flow heat exchanger hot and cold fluids are passes through the tubes in the same direction where as in counter flow heat exchanger both the fluids are passes in opposite direction to produce the desired effect. Baffles are sometimes used in heat exchanger to enhance heat transfer efficiency of heat exchangers. The concentric pipe heat exchanger is designed in present work in which different inlet velocities of fluid were taken and study is carried out for the temperature distribution, wall shear stress, and turbulent KE and pressure distribution for those velocities..

Keywords: Concentric pipe Heat Exchanger, Conduction, Convection, Parallel flow, Counter flow, Hot and cold inlets, pressure distribution, temperature distribution, Velocity inlet, wall shear stress, turbulent KE, ANSYS.

1. Introduction

A heat exchanger is a device that allows heat from a fluid (a liquid or a gas) to pass to a second fluid (another liquid or gas) without the two fluids having to mix together or come into direct contact. If that's not completely clear, consider this. In theory, we could get the heat from the gas jets just by throwing cold water onto them, but then the flames would go out! The essential principle of a heat exchanger is that it transfers the heat without transferring the fluid that carries the heat.

They are widely used in space heating, Air conditioning, Power plants, Chemical plants, Petrochemical plants, Petroleum refineries, Natural gas processing and Sewage treatment.

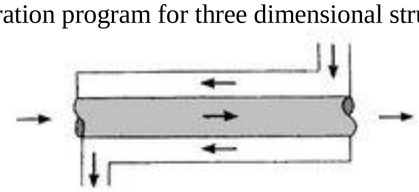
The general function of a heat exchanger is to transfer heat from one fluid to another. The basic component of a heat exchanger can be viewed as a tube with one fluid running through it and another fluid flowing by on the outside. There are thus three heat transfer operations that need to be described:

1. Convective heat transfer from fluid to the inner wall of the tube,
2. Conductive heat transfer through the tube wall, and
3.Convective heat transfer from the outer tube wall to the outside fluid.

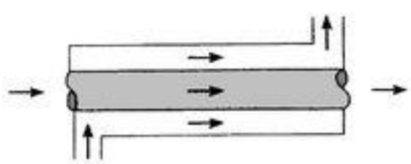
Heat exchangers are typically classified according to flow arrangement and type of construction. The simplest heat exchanger is one for which the hot and cold fluids move in the same or opposite directions in a concentric tube (or double-pipe) construction. In the parallel-flow arrangement of Figure 1.1 (a), the hot and cold fluids enter at the same end, flow in the same direction, and leave at the same end. In the counter-flow arrangement of Figure 1.1 (b), the fluids enter at opposite ends, flow in opposite directions, and leave at opposite ends.reaction of building structures; to decide the base seismic hole among structures and furnish engineers with functional investigative apparatuses for anticipating beating reaction and harm. A practical beating model is utilized for examining the reaction of auxiliary framework under the state of basic beating during elcentro tremors for medium soil condition at seismic zone V. Two contiguous multi-story structures are considered as an agent structure for potential beating issue. Dynamic and sucker examination is completed on the structures to watch removal of the structures because of seismic tremor excitation. The conduct of the structures under static burdens is straight and can be anticipated. At the point

Engineering Universe for scientific Research and management
ISSN (Online): 2319-3069
Vol. XIV Issue IX
September 2022

when we go to the dynamic practices, we are mostly worried about the relocations, speed and increasing velocities of the structure under the activity of dynamic burdens or quake loads. Unusualness in basic practices is experienced when the structure goes into the post-versatile or non-straight stage. The idea of push over examination can be used for evaluating the dynamic needs forced on a structure by tremor ground movements and the likely areas of the disappointment zones in a structure can be found out by watching the sort of pivot arrangements. The quality limit of the powerless zones in the post-flexible range would then be able to be expanded by retrofitting. With the end goal of this investigation, SAP2000 has been picked, a straight and non-direct static and dynamic examination and configuration program for three dimensional structures.



[Parallel flow]



[Counter flow]

Figure 1.1: Concentric tubes heat exchangers

1.1. Different types of heat exchanger:

- Parallel-flow and counter-flow heat exchanger.
- Finned and Un-finned tubular heat exchanger.
- U-tube, single pass straight and two pass straight heat exchanger.
- Plate-and-frame heat exchanger.
- Pate-fin heat exchanger.
- •Micro-channel heat exchanger.

1.2. Double Pipe Heat Exchanger Design with Counter flow or Parallel Flow

In double pipe heat exchanger design, an important factor is the type of flow pattern in the heat exchanger. A double pipe heat exchanger will typically be either counter flow or parallel flow. Crossflow just doesn't work for a double pipe heat exchanger. The flow pattern and the required heat exchange duty allows calculation of the log mean temperature difference. That together with an estimated overall heat transfer coefficient allows calculation of the required heat transfer surface area. Then pipe sizes, pipe lengths and number of bends can be determined.

A double pipe heat exchanger, in its simplest form is just one pipe inside another larger pipe. One fluid flows through the inside pipe and the other flows through the annulus between the two pipes. The wall of the inner pipe is the heat transfer surface. The pipes are usually doubled back multiple times as shown in the diagram at the left, in order to make the overall unit more compact.

The term 'hairpin heat exchanger' is also used for a heat exchanger of the configuration in the diagram. A hairpin heat exchanger may have only one inside pipe, or it may have multiple inside tubes, but it will always have the doubling back feature shown. . Some heat exchanger manufacturers advertise the availability of finned tubes in a hairpin or double pipe heat exchanger. These would always be longitudinal fins, rather than the more common radial fins used in a crossflow finned tube heat exchanger.

1.3 Objectives:

1. Pasteurization
2. Digester Heating
3. Heat recovery
4. Pre heating
5. Effluent Cooling

2. Literature review

Imen, D., &Ammar, L. (2021), [1]A solution for the flow in a heat exchanger is presented. An analytical approach is developed and validated by a numerical simulation using the ANSYS CFX code module, taking into account different boundary conditions in the input. The distribution of the laminar flow temperature is indicated. It refers to the Reynolds number and the hydraulic diameter of the heat exchanger.

This study allowed us to develop a simple and effective approach to improve the digital design of heat exchangers. In this work, we presented a study on the performance of a coaxial tube heat exchanger with a spherical end 2 m long and 40.10-3 m outer diameter of the central tube and 80.10-3 m outer diameter of the outer tube. In this study, two approaches were followed:

➢ An analytical approach which enabled us to calculate the outlet temperature of the heat exchanger.

➢ A numerical approach based on numerical simulation using CFX module ANSYS code that we give the distribution of temperature taking into account different parameters such as the heat transfer coefficient, the Reynolds number, the inlet temperature, the geometry and the imposed thermal condition such as temperature.

Gowthaman, P. S. Et.al. (2021) [2]In this project we analyze two different deflectors in a jacket and shell heat exchanger of ANSYS FLUENT. Pipe and shell and tube heat exchangers are widely used in many industrial applications such as power generation, refrigeration, environmental protection and chemical engineering. Based on the results of the numerical experiment, the performance of the heat exchanger in the spiral deflector is improved instead of the segment deflector.

1) In this work a model was developed to evaluate the analysis of a spiral and segment impingement heat exchanger as well as the comparative analysis of the thermal parameters between the segment angle and the spiral angle.

2) The numerical results of the experiment confirm that the performance of a tube heat exchanger can be improved by spiral deflectors instead of segmented deflectors.

3) The use of spiral baffles in the heat exchanger reduces the pressure drop on the shell side.

4) The heat / transverse area ratio increases the mass flow in the shell. Transmission coefficient / pressure drop higher than that of the segmented deflector.

5) The pressure drop in the coil impact heat exchanger is much lower than that of the segment impact heat exchanger.

Abdolbaqi, M. K., Azwadi, C. S. N., Mamat, R., Azmi, W. H., &Najafi, G. (2019)[3]The need for high thermal performance thermal systems has been eventuated by finding different ways to enhance heat transfer rates. This paper introduces and analyzes numerically the heat transfer enhancement of nano fluids with different volume concentrations under turbulent flow through a straight channel with a constant heat flux condition. Solid nanoparticles of TiO2 and CuO were suspended in water as a base fluid to prepare the nano fluids. CFD analysis is conducted by FLUENT software using the finite volume method.

In the present study, the thermal properties of two types of nanoparticle suspended in water have been investigated numerically in forced convection heat transfer under turbulent flow with the uniform heat flux boundary condition of a straight channel. The heat transfer enhancement resulting from various parameters such as nanoparticle concentration of volume, and Reynolds number are reported.

Raj, R. T. K., &Ganne, S. (2019)[4]In this present study, attempts were made to investigate the impacts of various baffle inclination angles on fluid flow and the heat transfer characteristics of a shell-and-tube heat exchanger for three different baffle inclination angles namely 0°, 10°, and 20°. The simulation results for various shell and tube heat exchangers, one with segmental baffles perpendicular to fluid flow and two with segmental baffles inclined to the direction of fluid flow are compared for their performance. The shell side design has been investigated numerically by modeling a small shell- -and-tube heat exchanger.

Following conclusions are observed:

- The shell side of a small tube-and-shell heat exchanger is modeled with enough detail to resolve the flow and temperature fields.
- For a given geometry, the mass flow must be less than 2 kg / s. When the mass flow exceeds 2 kg / s, the pressure drop increases rapidly with small differences in the outlet temperature.
- The pressure drop is reduced by 4% for heat exchangers with an impact angle of 10° and 16% for heat exchangers with an impact angle of 20°.
- The maximum angle of inclination of the deflector can be 20 °. If the angle exceeds 20°, the middle row of tubes is not supported. Therefore, the baffle can not be used effectively.
- It can be concluded that shell and tube heat exchangers with a 20° angle of impact offer better performance than 10 ° and 0 ° tilt angles..

K. Palanisamy a, P.C. Mukesh Kumar (2019) [5],This study investigates the heat transfer and the pressure drop of cone helically coiled tube heat exchanger using (Multi wall carbon nano tube) MWCNT/water nano fluids. The MWCNT/water nano fluids at 0.1%, 0.3%, and 0.5% particle volume concentrations were prepared with the addition of surfactant by using the two-step method. It is studied that the prepared MWCNT/water nano fluids show good stability even after 45 days of preparation and there is no considerable deposit of nano tubes on the tube inner wall. In this paper, the turbulent flow ($2200 < De < 4200$) heat transfer characteristics and pressure drop of cone helically coiled tube with MWCNT/water nano fluid at 0.1%, 0.3% and 0.5% particle volume concentration have been experimentally studied. It is found that the maximum overall heat transfer coefficient of nano fluids is 52% higher than the water at 0.5% nano fluid in the Dean number 4200. The improved heat transfer coefficient is found to be 14%, 30% and 41% more than the water at 0.1%, 0.3% and 0.5% MWCNT/water nano fluid respectively. The increase in Nusselt numbers is found to be 28%, 52% and 68% at 0.1%, 0.3% and 0.5% MWCNT/water nano fluids respectively when compared with water. The pressure drop of 0.1%, 0.3% and 0.5% nano fluids are found to be 16%, 30% and 42% higher than water respectively.

Danook, S., Jasim, Q., & Hussein, A. (2018)[6],Heat transfer enhancement employing an elliptical tube inside a circular tube to increase the heat transfer rate without increasing in pressure drop is investigated. The flow rate inside the narrow is in the range of Reynolds number 10,000 to 100,000. Commercial software is used to solve the governing equations (continuity, momentum, and energy) by adopting a finite volume method (FVM). The electrical heater is connected around the circular tube to apply uniform heat flux (3000 W/m2) as a boundary condition.

- In this paper, forced convection heat transfer through an elliptical inside a circular tube under turbulent flow by numerical simulation, with uniform heat flux boundary condition around circular tube has been studied. The Nusselt number, friction factor was obtained through the numerical simulation. The study is concluded as follows:
- The enhancement of friction factor and Nusselt numbers are 14% and 19% for enhanced tube than that of the circular tube at all Reynolds numbers.
- The concentration of volume (1%) of TiO_2 nano fluid has the highest Nusselt number and friction factor values, followed by (0.75%, 0.5%, and 0.25%) finally pure water has the lowest valuesof them.

Haonon change, taoluo(2018) [7].A combined experimental and numerical study is performed aiming to understand the role of buoyancy-driven convection during constrained melting of phase change materials (PCMs) inside a shell and tube heat exchanger.

A series of experiments is conducted to investigate the effect of increasing the inlet temperature of the heat transfer fluid (HTF) on the charging process (melting) of the PCM. The computations are based on an iterative, finite-volume numerical procedure that incorporates a single-domain enthalpy formulation for simulation of the phase change phenomenon. It was observed from experimental results that the melting front appeared at different times at positions close to the HTF tube and progressing at different rates outwards towards the shell. The computational results show that by increasing the inlet water temperature to 80 °C, the total melting time is decreased to 37%.

Ramanathan, S. (2018) [8].Double-walled heat exchangers are mainly used in solar hot water systems (SDHW) because of their high heat transfer and small footprint. They are also used in heat recovery systems for space heating. This project focuses on the performance analysis of the spiral and helical shell heat exchanger to improve the performance of the spiral tube heat exchanger using Nano fluid. As part of this project, thermal transfer analysis on helical and helical spiral tubes was experimentally performed using water and TiO_2-water nano fluid at a volume concentration of 0.5. %.

Pandey, P. K., Lakhani, P. K., Kumar, K., Bohra, P., & Mishra, R.(2017) [9]The heat exchanger plays an important role in the industrial heating process. The heat is transferred by convection and conduction through the walls of the heat exchanger between the liquids. The heat transfer fluids have a low thermal conductivity, which considerably limits the efficiency of the heat exchanger. Various research studies are carried out to improve the thermal properties of liquids by adding heat-conductive solids in liquids. A fluid mixed with nano particles, called "nano fluid," has a thermal conductivity that is significantly higher than that of the corresponding base fluids. In this work, a new nano fluid system was developed in which nano particles of aluminum oxide and silicon carbide were synthesized with water and used in a shell-and-tube heat exchanger.

The performance of the heat exchanger (tube bundle) was investigated using experimental observations and numerical techniques. The results were shown by highlighting the variation of nanoparticle flux and concentration. The experimental results show that the application of an Al_2O_3 / SiC-based nano fluid improves the heat transfer coefficient with concentration or flow (5% increase).

De, D., Pal, T. K., &Bandyopadhyay, S. (2017) [10] ,At present, tube and shell heat exchanger is the most common type of heat exchanger widely used in petroleum refining and other large-scale chemical processes, as it is suitable for high-pressure applications. pressure. The objective of this work is the construction of a coil heat exchanger with spiral deflector and the comparison with a right deflector with CFD analysis using ANSYS FLUINT software tools. All models are designed using CATIA software tools. In this work, it has been studied how the pressure drop and the overall heat transfer coefficient change due to the different helix angle when the flow rate remains the same. The heat exchanger hull side flow configuration with continuous helical baffles must rotate and operate in a helix due to the geometry of the continuous helical baffles, resulting in a significant increase in the heat transfer coefficient by unit of pressure drop in the heat exchanger.

1. The results shows the clear idea that the helical baffle heat exchanger has better overall heat transfer coefficient than the straight baffle Heat Exchanger

2.In shell side the pressure drops are lower than the conventional straight baffle heat exchanger. The pressure drop is decreases with the increases of helix angle in all the cases considered. However, the effects of helix angles on pressure drop are small when helix angle greater than 12 degrees.

3.In Tube Side Heat Transfer Coefficient increases withthe increase in Inlet Velocity. Results in greater Heat transfer.

Alimoradi, A., &Veysi, F. (2017) [11].In this study, calculations of heat transfer and entropy generation were

Engineering Universe for scientific Research and management
ISSN (Online): 2319-3069
Vol. XIV Issue IX
September 2022

performed for stationary forced convection heat transfer in shell and tube heat exchangers. The effect of the geometric parameters of the heat exchanger, in particular: diameter of the tube (dt), diameter of the coil (dc), diameter of the entry of the hull (dv), diameter of the hull (dsh), height of the helix (Hc), shell height (Hsh), the slope (p) and the distance between the inlet and the outlet of the shell (f) on the heat transfer rate and the entropy production were studied simultaneously. The critical and optimum values of these parameters were obtained, which minimizes or maximizes COD (heat transfer rate through entropy generation).

Hameed, V. M., & Al-Khafaji, A. R. (2017) [12], an experimental and numerical study had been done on shell and helical tube heat exchanger. The perplex tube of 1000 mm length, 150 mm diameter and 2 mm thickness was used as shell. The helical tube was made of Cu material. Its diameter is 12.7 mm and 0.1 mm thickness. The working fluid was water for both shell and tube sides. The experimental rig consist of two water tanks to supply the cold water to the shell (35oC) and hot water to the tube (65oC). Eight thermocouples type K were installed at the inlet and out let of each sides and the other are distributed along the shell length. Two rotameters were used to measure the flow rate of hot and cold water. The Numerical analysis was done by using SNSYS-Fluent V.16 to predict the results of what had done experimentally. The main keys of this study were coil pitch and mass flow rate of water for both sides. Where the helical coil pitch was changed in each case. The first case the helical coil pitch was 52.7mm, in the second case the pitch was changed to 42.7mm. In the last case (case 3) the pitch distance was 32.7mm.

3. Modeling and Simulation

First of all the model of heat exchanger is made using some mechanical modeling software in this work we used CATIA for this purpose. The CATIA version 5 and release 12 software is being used in which first CATIA sketcher module is used to make the 2D sketch of part to be made and then in CATIA 3D workbench the depth is given to the sketch. Both concentric pipes are made in different part design files and then in CATIA assembly module these are called to fix concentrically over each other.

After successful modeling and assembly of the parts the assembled product is saved in a suitable graphical exchange format so that the file can be exported to the ANSYS. In present work we used ANSYS 14.0 version for the fluent analysis of the heat exchanger. Here again 14.0 stands for version 14 of ANSYS. Figure 3.1 represents the geometry made in CATIA

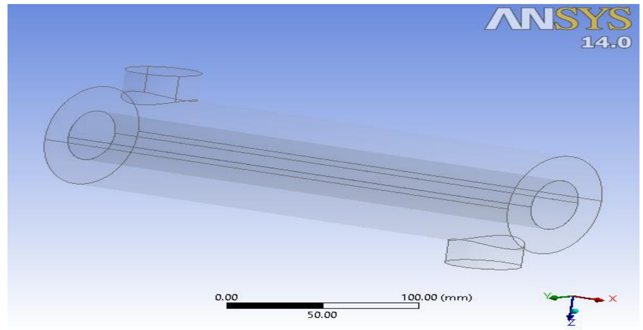


Figure 1. Geometry of Heat Exchanger

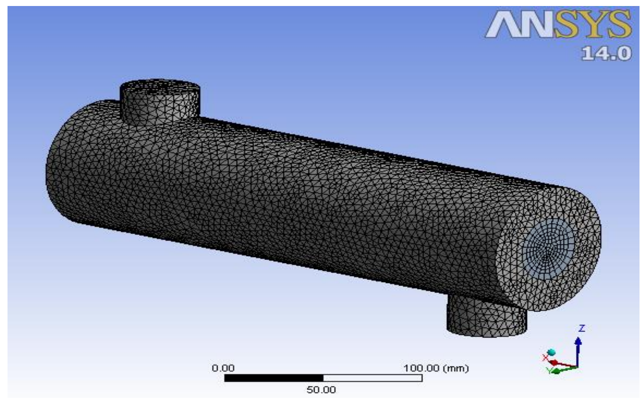


Figure 2. Meshed Model of Heat Exchanger

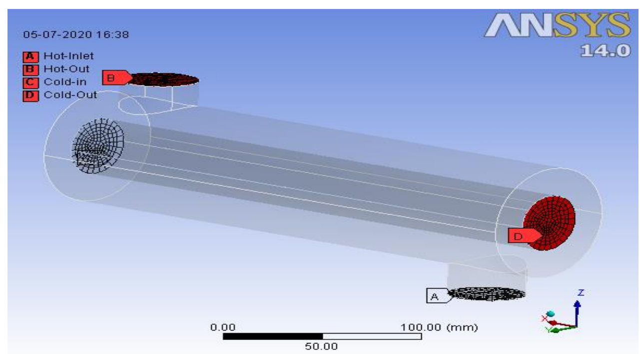


Figure 3 Named selection for providing the boundary conditions

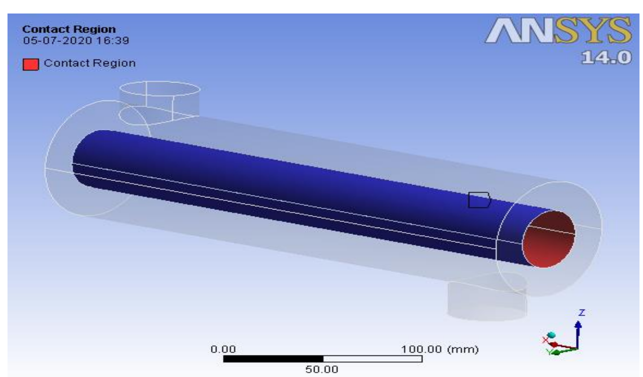


Figure 4: Contact region between hot and cold fluid chambers

Engineering Universe for scientific Research and management
ISSN (Online): 2319-3069
Vol. XIV Issue IX
September 2022

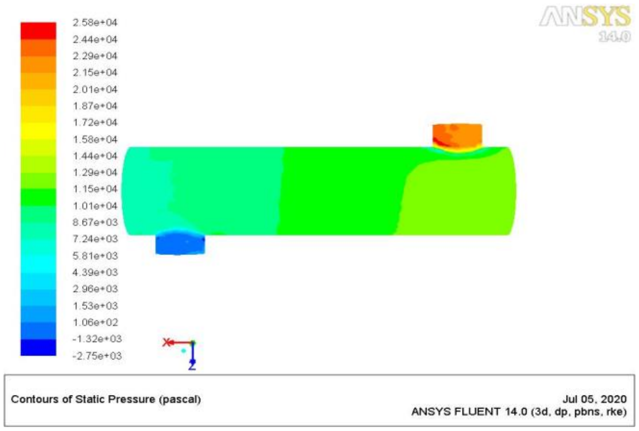


Figure 5. Pressure Distribution for inlet velocity flow rate

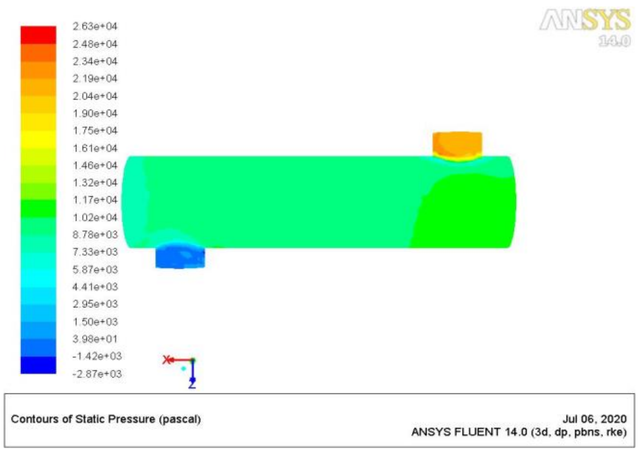


Figure 6. Pressure Distribution for inlet velocity flow rate of 0.65 m/s

4. Conclusions

1.Ansys 14.0 is an effective tool to analyze the heat transfer related problems and its fluent workbench is very helpful tool for fluid flow problems.
2. It can be stated that on increasing the inlet velocities of hot and cold fluid chambers pressure drops and the rest of the parameters are increases namely temperature, turbulence and wall shear stress values.
3. If the model is small then the velocity must be kept around 0.5 m/s or lower as we can see that on a small increase in velocity the heat transfer is not taking place effectively in heat exchanger applications. (In present case a length of 1m is taken for heat exchanger)
4. We can see on a small increase in inlet velocity from 0.6 m/s to 0.65 m/s adversely effects all the considered parameters and will be more significant where the length of heat exchanger is even larger.
5. With increase in temperature the turbulence K.E and wall shear stress vales also increases and is not a good sign for the heat exchanger.

References

[1] Imen, D., &Ammar, L. (2021). Distribution of the temperature in the coaxial tube heat exchanger with spherical end. Case studies in thermal engineering, 10, 621-627.
[2] Gowthaman, P. S., &Sathish, S. (2021). Analysis of segmental and helical baffle in shell and tube heat exchanger. Int. J. Curr. Eng. Technol, 2, 625-628.
[3] Abdolbaqi, M. K., Azwadi, C. S. N., Mamat, R., Azmi, W. H., &Najafi, G. (2019). Nano fluids heat transfer enhancement through straight channel under turbulent flow. International Journal of Automotive and Mechanical Engineering, 11, 2294
[4] Raj, R. T. K., &Ganne, S. (2019). Shell side numerical analysis of a shell and tube heat exchanger considering the effects of baffle inclination angle on fluid flow. Therm. Sci., 16(4), 1165-1174.
[5] K. Palanisamy a, P.C. Mukesh Kumar (2019), Experimental investigation on convective heat transfer and pressure drop ofcone helically coiled tube heat exchanger using carbon nanotubes/water nano fluids, Heliyon 5 (2019) e01705.
[6] Danook, S., Jasim, Q., & Hussein, A. (2018). Nano fluid Convective Heat Transfer Enhancement Elliptical Tube inside Circular Tube under Turbulent Flow. Mathematical and Computational Applications, 23(4), 78.
[7] Haonon change, taoluo(2018). A combined experimental and computational study on the melting behavior of a medium temperature phase change storage material inside shell and tube heat exchanger, energy procedia 152(2018) 990-996.
[8] Ramanathan, S. (2018). Experimental Investigation of Shell and helical coiled tube heat exchanger using TiO2-Water based Nano fluid. International Research Journal of Automotive Technology, 1(6), 122-127.
[9] Pandey, P. K., Lakhani, P. K., Kumar, K., Bohra, P., & Mishra, R. (2017). Heat Transfer Analysis of Shell and Tube Heat Exchanger using Al2O3/SiCNano fluid. European Journal of Advances in Enginee
[10] 10 De, D., Pal, T. K., &Bandyopadhyay, S. (2017). Helical baffle design in shell and tube type heat exchanger with CFD analysis. International Journal of Heat and Technology, 35(2), 378-383.