International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 06, June-2019

Numerical Investigation of Concentric Tube Heat Exchanger using a Nanofluid

Mr. M Sai Krishnakanth
Pursuing PhD in Osmania University
Mechanical Engineering Department
Hyderabad, India

Col T A Aravind
Dean, Faculty of Degree Engineering
MCEME
Secunderabad, India

Mr. A Ashok
Assistant Professor, Faculty of Degree Engineering
MCEME
Secunderabad, India

***Abstract*— Double tube or Concentric tube heat exchangers have been used in the field of power plants, refrigeration and chemical plants to improve the heat transfer rate at greater efficiency. Here, in this work, an analysis is to be done for double tube heat exchanger with varying flow rates and also by varying the concentrations of nanoparticles. Study of increasing the heat transfer rate in heat exchanger using Al_2O_3 nanofluid as base fluid. The analysis is to be done by using CFD methodology using ANSYS Fluent 14.0. The heat transfer rate is varied for counter flow pattern. Nanofluids are added and used as base fluid which increases the heat transfer rate. Different parameters can be calculated from the results obtained and graphs can be plotted between various parameters such as Nusselt number, friction factor, Reynolds number, and change in heat transfer rate (some of them are discussed here). These graphs have to be analyzed and discussed to find out the optimal result for which the heat exchanger would give the best performance. Finally, the heat transfer rate is improved by varying the flow of fluid and the efficiency of heat. It is considered to have great potential for heat transfer enhancement and is highly suited to application in heat transfer processes.**

Keywords*— Double tube/pipe heat exchanger; parallel and counter flows; heat transfer rate; nanofluid; CFD

I. INTRODUCTION

The Temperature can be defined as the amount of energy that a substance has. Heat exchangers are used to transfer that energy from one substance to another. In process units it is necessary to control the temperature of incoming and outgoing streams. These streams can either be gases or liquids. Heat exchangers raise or lower the temperature of these streams by transferring heat to or from the stream. The temperature gradient or the differences in temperature facilitate this transfer of heat. Transfer of heat happens by three principle means: radiation, conduction and convection. However, in comparison to conduction and convection, radiation does not play a major role. To maximize the heat transfer the wall should be thin and made of a very conductive material. The biggest contribution to heat transfer in a heat exchanger is made through convection.

The double-pipe heat exchanger is one of the simplest types of heat exchangers. It is called a double-pipe exchanger because one fluid flows inside a pipe and the other fluid flows between that pipe and another pipe that surrounds the first. This is a concentric tube construction. Flow in a double-pipe heat exchanger can be parallel flow or counter flow. There are two flow configurations: parallel flow is when the flow of the two streams is in the same direction, counter flow is when the flow of the streams is in opposite directions. Counter flow is chosen for the present study. In this double-pipe heat exchanger a hot process fluid flowing through the inner pipe transfers its heat to cooling water flowing in the outer pipe. The system is in steady state until conditions change, such as flow rate or inlet temperature. These changes in conditions cause the temperature distribution to change with time until a new steady state is reached. The new steady state will be observed once the inlet and outlet temperatures for the process and coolant fluid become stable. In reality, the temperatures will never be completely stable, but with large enough changes in inlet temperatures or flow rates a relative steady state can be observed. In an heat exchanger, forced convection allows for the transfer of heat of one moving stream to another moving stream. Conventional heat transfer fluids have inherently poor thermal conductivity which makes them inadequate for ultra high cooling applications.

All physical mechanisms have a critical scale below which the properties of a material changes totally. Modern nanotechnology offers physical and chemical routes to prepare nanometer sized particles or nanostructured materials engineered on the atomic or molecular scales with enhanced thermo-physical properties compared to their respective bulk forms. These nanoparticle-fluid suspensions are termed nanofluids, obtained by dispersing nanometer sized particles in a conventional base fluid like water, oil, ethylene glycol etc. Nano particles of materials such as metallic oxides (Al_2O_3, CuO), Nitride ceramics (AlN, SiN), Carbide ceramics (SiC, TiC), metals (Cu, Ag, Au), semiconductors (TiO_2, SiC), single, double or multi walled carbon nanotubes (SWCNT, DWCNT, MWCNT), alloyed nanoparticles ($Al_{70}Cu_{30}$), etc., have been used for the preparation of nanofluids. These nanofluids have been found to possess an enhanced thermal conductivity as well as improved heat transfer performance at low concentrations of nanoparticles. Even at very low volume fractions (< 0.1%) of the suspended particles, an attractive enhancement up to 40% in thermal conductivity has been reported on these nanotechnology based fluids and the percentage of enhancement is found to increase with temperature as well as concentration of nanoparticles.

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 06, June-2019

Most of the literature surveyed yields that the shell & tube type heat exchangers has been given a great respect among all the classes of heat exchangers due to their virtues like comparatively large ratios of heat transfer area to volume and weight and many more. Moreover well designed as well as described methods are available for its designing and analysis. It is also shown by the literature survey that the Computational Fluid Dynamics through softwares like ANSYS, etc. have been successfully used and implemented to secure the economy of time, materials and efforts.

Etemad and *Farajollah*i (2010) [6] performed experiments for wide ranges of Peclet numbers and nano particle concentrations and observed that heat transfer characteristics enhance significantly with Peclet number. *Hosseini et al.* (2011) [4] reviewed thermo-physical characteristics of nanofluids and their role in heat transfer enhancement and observed that thermo physical properties of nanofluids are dependent on the nanoparticles size, shape, and volume fraction. *Yerrennagoudaru et al.* (2016) [7] observed the enhancement in the thermal conductivity upon experimentation of various nanofluids in varied range of their size. *Bharat and Hatkar* (2017) [5] experimented observed that for high Reynolds number, low concentration of nanofluid is useful and found a promising enhancement in heat transfer rate using nanofluid, by experimenting on a double pipe heat exchanger.

II. METHODOLOGY

A. *Computational Fluid Dynamics*

Usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical analyse and algorithms to solve and analyse problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions with high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests. The fundamental basis of almost all CFD problems are the Navier–Stokes equations which define any single-phase (gas or liquid, but not both) fluid flow. These equations can be simplified by removing terms describing viscous actions to yield the Euler equations further simplification, by removing terms describing vortices yields the full potential equations. Finally, for small perturbations in subsonic and supersonic flows (not transonic or hypersonic) these equations can be linearized to yield the linearized potential equations Computational fluid dynamics (CFD) study of the system starts with the construction of desired geometry and mesh for modelling the dominion. Generally, geometry is simplified for the CFD studies. Meshing is the discretization of the domain into small volumes where the equations are solved by the help of iterative methods. Modelling starts with the describing of the boundary and initial conditions for the dominion and leads to modelling of the entire system. Finally, it is followed by the analysis of the results, discussions and conclusions.

B. *Methodology conceived*

Here the analysis is done using ANSYS Fluent 14.0 software. The analysis procedure involves the following steps:

- Geometrical modelling
- Meshing
- Solution
- Material selection
- Defining zones
- Boundary conditions
- Solution methods
- Solution initialization
- Iteration
- Plot results and contour

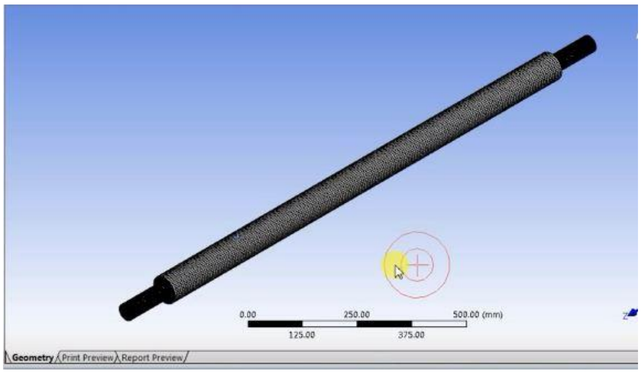

Fig. 1. Meshed geometry of double pipe heat exchanger

Geometry has been created using ANSYS Design Modeller. In free meshing a relatively coarser mesh is generated. It contains both tetrahedral and hexahedral cells having triangular and quadrilateral faces at the boundaries. Later, a fine mesh is generated using edge sizing. In this, the edges and regions of high pressure and temperature gradients are finely meshed. The mesh is checked and quality is obtained. The analysis type is changed to Pressure Based type. The velocity formulation is changed to absolute and time to steady state. Gravity is defined as y = -9.81 m/s^2.

Energy is set to ON position. Viscous model is selected as 'k-ϵ model (2 equations)'. The k-epsilon (k-ϵ) model [3] for turbulence is the most common to simulate the mean flow characteristics for turbulent flow conditions. This is a 2 equation model which gives a general description of turbulence by means of two transport equations (PDEs), which accounts for the history effects like convection and diffusion of turbulent energy. The 2 transported variables are turbulent kinetic energy k, which determine the energy in turbulence, and turbulent dissipation ϵ, which determines the rate of dissipation of the turbulent kinetic energy. The k-ϵ model is shown to be applicable for free-shear flows, such as the ones with relatively small pressure gradients [1], but might not be the best model for problems involving large adverse pressure gradients [2]. Hence this model might not be suitable for inlets and compressors.

Water-liquid as fluid and copper as solid was selected from the fluent database by clicking change/create. Different parts were assigned as solid or fluid accordingly. Different boundary conditions were applied for different zones. Since it is a tube-in-tube heat exchanger, there are two inlets and two outlets. The inlets were defined as mass flow rate inlets and outlets were defined as outlets. The inlet mass flow rate of the

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 06, June-2019

hot fluid was kept constant i.e. 0.025 kg/s, whereas mass flow rate of cold fluid was varied from 0.03 kg/s to 0.05 kg/s for different experiments. Thermo-physical properties [4] of Al_2O_3-water nanofluid are calculated for various volume fractions of Al_2O_3 nanoparticles and tabulated as below.

TABLE I. THERMOPHYSICAL PROPERTIES OF AL_2O_3-WATER NANOFLUID FOR DIFFERENT VOLUME FRACTIONS OF AL_2O_3 NANOPARTICLES

Volume fraction, ppm	**Thermal conductivity (k) W/mK**	**Specific heat (C_p) J/kg-K**	**Density (ρ) kg/m³**	**Viscosity (μ) kg/m-s**
0.002	0.603	4156	1004	0.001008
0.004	0.606	4134	1009	0.001013
0.006	0.61	4106	1015	0.001018

The outlet pressures were kept default i.e. atmospheric pressure. The hot fluid temperature at inlet was 373 Kelvin and cold fluid inlet temperature was kept 300 Kelvin. The other wall conditions were defined accordingly. The outer wall is taken as the adiabatic wall.

For solution control and initialization:

Pressure = 0.3 Pa
Density = 1 kg/m^3
Body forces = 1 kg-m /s^2
Momentum = 0.7 kg-m/s
Turbulent kinetic energy = 0.8 kg-m^2/s^2

Then the solution initialization method is set to Standard Initialization whereas the reference frame is set to all zones. The number of iterations was set to 1000 with step size 1. Then the calculation was started and it continued till the results converged.

Various contours were plotted and different parameters were calculated such as weighted average of total temperatures at outlet and inner wall.

III. RESULTS AND DISCUSSION

Analysis using ANSYS FLUENT 14.0 on double pipe heat exchanger has been carried out as per methodology. The overall cold fluid outlet temperatures obtained and temperature distribution of fluids are shown in figures.

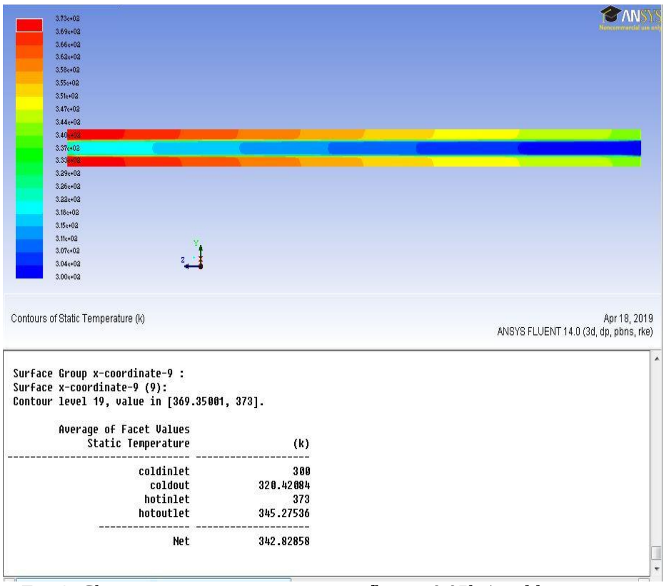

Fig. 2. Change in temperature in counter flow at 0.05lit/s cold water mass flow rate

Above figure (Fig. 2.) indicates the static temperature of cold fluid outlet is 320.42 K for water in counter flow.

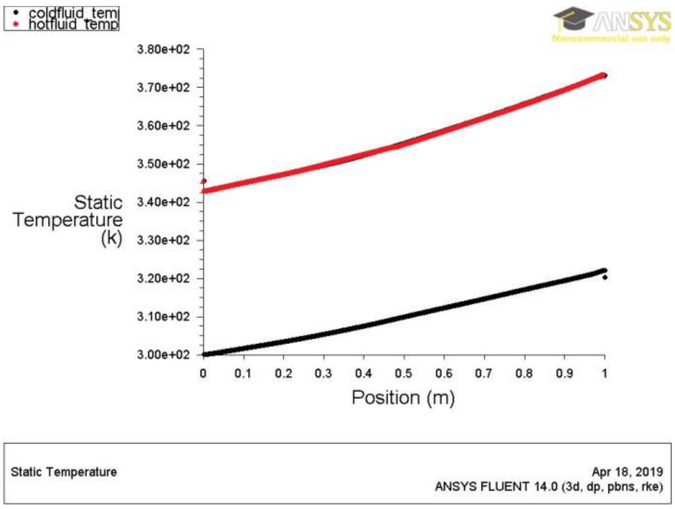

Fig. 3. Temperature variation of hot and cold fluids along the length in counter flow.

From the above figure (Fig. 3.), the outlet temperature of cold fluid is 320.42K

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 06, June-2019

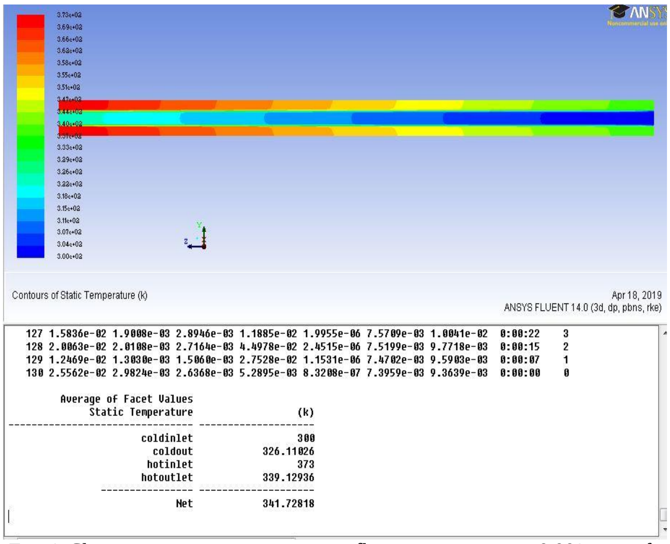

Fig. 4. Change in temperature in counter flow at concentration 0.002 ppm of cold nanofluid at 0.05 lit/s mass flow rate.

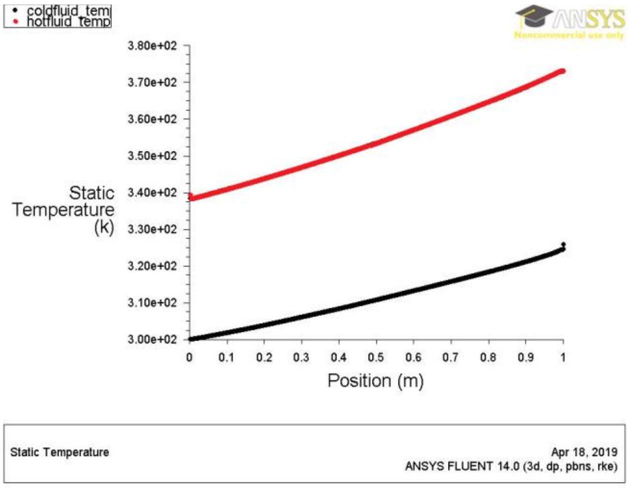

Fig. 5. Temperature variation of hot and cold fluids along the length with 0.002 ppm concentration nanoparticles in cold water in counter flow.

Above figure (Fig. 4.) shows the static temperature of cold fluid outlet is 326.11K of water with 0.002 ppm concentration Al_2O_3 nanoparticles in counter flow.

From Fig. 5., the cold fluid outlet temperature is 326.11K which is 5.69K higher than water as working fluid in counter flow.

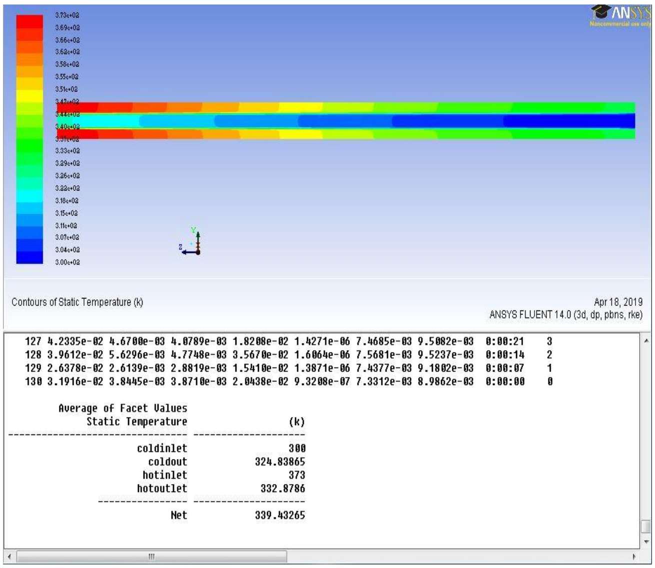

Fig. 6. Change in temperature in counter flow at concentration 0.004 ppm of cold nanofluid at 0.05 lit/s mass flow rate

Above figure (Fig. 6.) shows the static temperature of cold fluid outlet is 324.83K of water with 0.004 ppm concentration Al_2O_3 nanoparticles in counter flow.

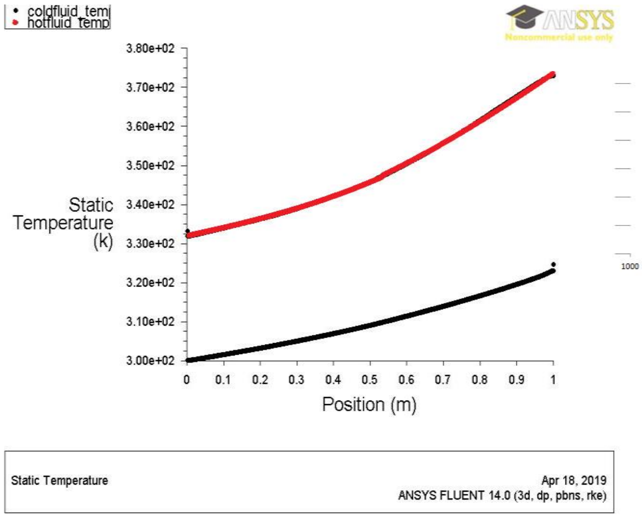

Fig. 7. Temperature variation of hot and cold fluids along the length with 0.004 ppm concentration nanoparticles in cold water in counter flow.

From the above figure (Fig. 7.) the cold fluid outlet temperature is 324.83K which is 4.41K higher than water as working fluid in counter flow.

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 06, June-2019

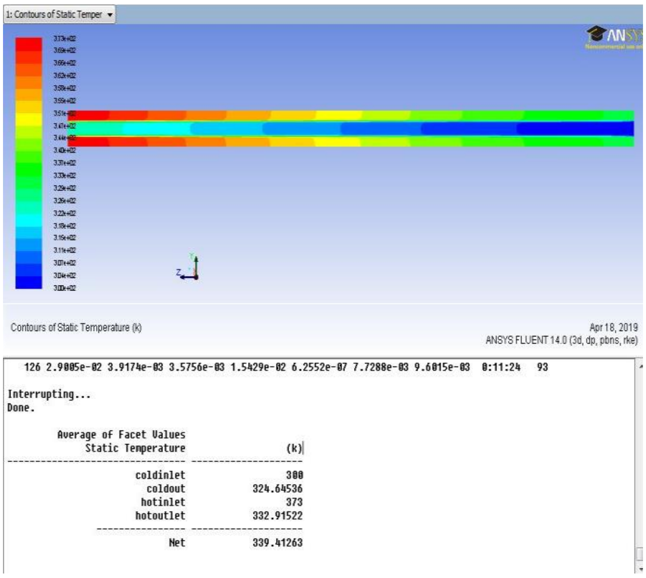

Fig. 8. Change in temperature in counter flow at concentration 0.006 ppm of cold nanofluid at 0.05 lit/s mass flow rate.

Above figure (Fig. 8.) shows the static temperature of cold fluid outlet is 324.64 K of water with 0.006 ppm concentration Al_2O_3 nanoparticles in counter flow.

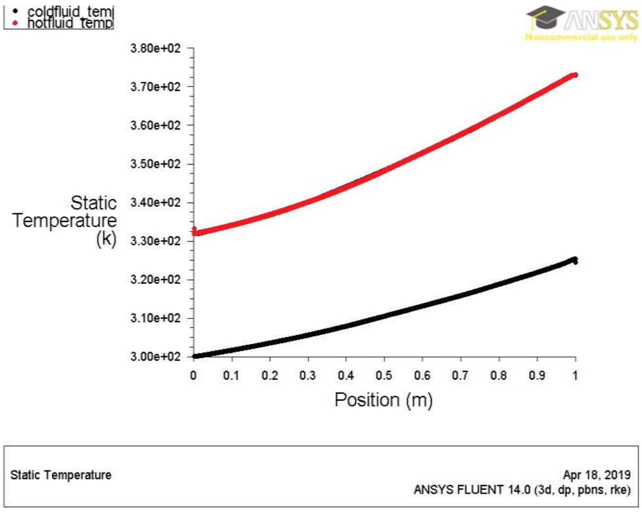

Fig. 9. Temperature variations of hot and cold fluids along the length with 0.006 ppm concentration nanoparticles in cold water in counter flow.

From the above figure (Fig. 9.) the cold fluid outlet temperature is 324.64K which is 4.22K higher than water as working fluid in counter flow.

The following graph (Fig. 10.) shows the variation of cold water exit temperature by the addition of various volume fractions of nanoparticles.

Fig. 10. Cold water exit temperature for water and water mixed with nanomaterials in varying composition

IV. CONCLUSION

In this study, the CFD software was implemented successfully to standard *k*-ϵ turbulent model in double pipe heat exchanger. The temperatures inlet in tube and annulus were maintained at 373K and 300K and comparing the results with addition of nanomaterials in varying compositions in the cold fluid in annulus.

It can be concluded that by observing the results:

- The temperature of the cold fluid exit increases in comparison with cold fluid without nanomaterials.
- The temperature of hot fluid with nanomaterials decreases at the exit in comparison with hot fluid with no nanomaterials.
- The outlet temperatures are varied based upon mass flow rate as discussed above. For the taken input flow condition as the concentration of nanomaterials is increased the temperature of cold fluid outlet as compared to water as working fluid therefore by using of nanofluids in heat exchanger devices more amount of heat can be transferred.
- It has been observed that by addition of nanomaterials into the cold water the rate of heat transfer increased, which can be inferred from the increase in cold water outlet temperature.

REFERENCES

[1] Bardina, J.E., Huang, P.G., Coakley, T.J. (1997), “Turbulence Modeling Validation, Testing, and Development”, NASA Technical Memorandum 110446

[2] Wilcox, David C (1998). “Turbulence Modeling for CFD”, Second edition. Anaheim: DCW Industries, 1998. pp. 174

[3] https://www.simscale.com/docs/content/simulation/model/turbulenceModel/kEpsilon.html

[4] S.Sh.Hosseini, A.Shahrjerdi, Y.Vazifeshenas,“A Review of Relations for Physical Properties of Nanofluids” Australian Journal of Basic and Applied Sciences, 5(10): 2011.

[5] Bharat B. Bhosle, Prof.D.N.Hatkar, “Analysis of Heat Transfer Enhancement of Heat Exchanger using Nano fluid”, International Research Journal of Engineering and Technology (IRJET) Volume: 04 Issue: 04 | Apr -2017 www.irjet.net p-ISSN: 2395-0072, e-ISSN: 239500567]

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

[6] S.Gh. Etemad, B. Farajollahi, "Heat Transfer Of Nanofluids in A Shell And Tube heat Exchange", International Journal of Heat and Mass Transfer · January 2010

[7] Dr.Hiregoudar Yerrennagoudaru, Manjunatha.k ,B.Vishnu Prasad , Sandeep .k ,S.Veeresh Kumar, "Nano Fluids for Heat Exchanger", International Journal of Engineering Science and Innovative Technology (IJESIT)Volume 5, Issue 4, July 2016, ISSN: 2319-5967

[8] Heris, S., Esfahany, M., and Etemad, S. G. "Experimental investigation of convective heat transfer of Al2O3/water nano fluid in circular tube", International Journal of Heat and Fluid Flow, 28, no. 2, 203–210 (2007)

[9] Jorge A.W. Gut, Jos_e M. Pinto "Modelling of plate heat exchangers with generalized configurations" International Journal of Heat and Mass Transfer 46 (2003) 2571–2585.

[10] Manoj Kumar Pandita, Prof. Ashok Kumar Gupta, Kailash Kumar Lahadotiya "CFD analysis of a single shell and single tube heat exchanger and determining the effect of baffle angle on heat transfer" International journal of engineering sciences & research technology [Pandita*, 4.(6): June, 2015] ISSN: 2277-9655.

[11] Patel Dharmik A, V. D. Dhiman, Jignesh J. Patel, Ravi Engineer "CFD analysis of triple concentric tube heat exchanger" University Journal of Research ISSN (Online) 0000–0000, ISSN (Print) 0000–0000.

[12] Anil Kumar Samal "Shell and tube heat exchanger design using CFD tools".

[13] Thundil Karuppa Raj R, Srikanth Ganne "Shell side numerical analysis of a shell and tube heat exchanger considering the effects of baffle inclination angle on fluid flow using CFD".

[14] Vindhya Vasiny Prasad Dubey, Raj Rajat Verma, Piyush Shanker Verma, A.K.Srivastava "Performance Analysis of Shell & Tube Type Heat Exchanger under the Effect of Varied Operating Conditions" IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume11, Issue 3 Ver. VI (May- Jun. 2014), PP 08-17.

[15] Che´rif Bougriou, Khireddine Baadache "Shell-and-double concentric-tube heat exchangers" Heat Mass Transfer (2010) 46:315–322 DOI 10.1007/s00231-010-0572-z.

[16] Kallalu Harika, Tummala Likhitha ,KarnatiHema and Penumala Pavani, "Fabrication of Shell and Tube Heat Exchanger using Helical Baffles based on Kern's Principle", International Journal of Current Engineering and Technology E-ISSN 2277 – 4106, P-ISSN 2347–5161©2017

[17] A.Gopi Chand, A. V. N. L. Sharma, G. Vijay Kumar, A.Srividya, "Thermal Analysis Of Shell And Tube Heat Exchanger using Mat Lab And Floefd Software", International Journal of Research in Engineering and Technology ISSN: 2319-1163

[18] Das S.K, Putra N, Thiesen P, Roetzel W., Temperature dependence of thermal conductivity enhancement for nano fluids, ASME J Heat Transfer 125 (2003) 567–574.

[19] Yang Y, Zhang Z.G., Grukle A.K., Anderson W.B., Wu G., "Heat transfer properties of nanoparticle-influid dispersions (nano fluids) in laminar flow", Int. J. Heat and Mass Transfer, 48 (2005) 1107-1116

(This work is licensed under a Creative Commons Attribution 4.0 International License.)