

CFD analysis of TiO_2 /water nanofluid flow in a double pipe heat exchanger

Salman.K, Prasad E Prakash, *Malabar College of Engineering and Technology, Kerala Technological University*

ksalman671@gmail.com, prasadhearath@gmail.com

Abstract—Thermal properties of fluids play a important role in cooling and heating applications in all industrial operations. Thermal conductivity of the fluids is an important physical and thermal property that decides its heat transfer performances. Traditional fluids used for heat transfer have inherently mediocre thermal conductivity which makes them short for higher cooling applications. Researchers are always trying to improve the inherently mediocre thermal conductivity of these Traditional fluids used for heat transfer by adding additives following the classical effective medium theory for effective properties of mixtures. elevated convection between the solid particle and liquid surfaces is the main cause for the improvement . the thermal and transport properties of the base fluid are changed by the suspended particles . Titanium dioxide (TiO_2) is an excellent materials for heat transfer boosting purpose due to its perfect physical and chemical properties. Thermal analysis of drafted geometry(double pipe heat exchanger) is studied by varying concentration of nanoparticle (0-3vol%) in coolant using 3-Dimensional simulations .The aim of this project is to perform cfd analysis(ANSYS FLUENT 18.1) on nano fluid(TiO_2 /water) in a double pipe(tube in tube) heat exchanger and the result is analysed and compared with base fluid

Paper was submitted for review on the date of 25-03-2018 . paper was immensely supported by staff and authority of malabar cet college .

Salman k is student(MASTERS-Thermal engineering) at the malabar college of engineering, thrissur , kerala ,India (e-mail: ksalman671@gmail.com).

Prasidh E Prakash is Asst Professor at the malabar college of engineering, thrissur , kerala ,India (e-mail: PrasadhEPrakash@gmail.com)

I. INTRODUCTION

Thermal properties of fluids has a major role in temperature controlling applications in every industrial procedures. heat transfer performance is dependent on thermal conductivity of the fluids . traditional heat transfer fluids have poor thermal conductivity which makes them less preferred for majority of cooling applications. Researchers' have tried to improve the inherently mediocre thermal conductivity of these Traditional fluids used for heat transfer by adding minute solid particles as per the classical effective medium theory (Maxwell, 1873) properties of mixtures. The nanoparticles are the basic building blocks of nanofluids. in particular. In comparison with particles sized micrometre, nanoparticles possess high surface area to volume ratio it is due to presence of large number of atoms on the boundaries. Thus it make them highly stable in suspensions. There is a high convection rate between solid particle surface and liquid which improve the thermal properties such as thermal conductivity .

Nanofluids have unusual properties that make them a game changer in many applications of heat transfer, including engine cooling, in grinding, vehicle air conditioning management ,microelectronics, mining, hybrid-powered engines, defence, pharmaceutical industry, in space technology , large scale refrigeration systems, solar cell ,chiller, heat exchanger, nuclear reactor coolant and aeroplanes . They show improvement in thermal conductivity and the heat transfer coefficient in comparison with the normal

fluid. More research need to be done to decide their applicability for heat transfer applications.

Numerical simulations are performed in a model of double pipe heat exchanger to study thermal effects and fluid flow of various nanoparticle concentrations(1%,2%,3%). convection heat transfer of nanofluid systems is observed using 3-Dimensional CFD simulations ANSYS 18.1 (FLUENT). The improvement of the overall heat transfer efficiency of nanofluids are compared with coolant (base fluid). Overall heat transfer coefficient, are analysed at different concentration to study the effect of nanoparticle concentration on the heat transfer performance in the model(double pipe heat exchangers)

II. METHODOLOGY

A. Geometry

An model geometry of double pipe heat exchanger is made using ANSYS 18.1 (FLUENT). The dimensions and shape of the geometry is based text book. The model is confined to horizontal portion only . The simplified design of model is shown in Fig. 1. meshing method used is a sweep one at all the points

B. Materials selection

TiO₂ nanofluid systems are used in the study for improvement in heat conduct inside heat exchangers. The base fluid is water and the thermal and physical properties of water are taken from previous literature. Titanium based nanoparticles are chosen on the basis of high thermal conductivity. properties of nanofluids are obtained from Dr.Hiregoudar Yerrennagoudaru[1]copper tube used as inner tube,steel tube as outer one.

C. Meshing of geometry

Structured meshing method in ANSYS 18.1 WORKBENCH was used for the geometry. The element for meshing considered is square shape with number of elements of 789412 to 1440000. selections also done at required places and were Naming

D. Governing equations

Generally steady state simulations were taken out by solving mass, momentum and energy equations for single phase fluid, which are expressed as:

Continuity equation:

$$\partial\rho/\partial t + \partial/\partial X (\rho u_x) + \partial/\partial r (\rho u_r) + (\rho u_r)/r = 0 \quad (1)$$

Momentum equation:

$$(\partial(\partial u_x)/\partial t + (\rho(u_x u_x))) = \rho g - \nabla P + \nabla \cdot (\tau) \quad (2)$$

Energy equation:

$$(\partial(\partial E)/\partial t + \nabla(u_x \rho E + P)) = \nabla \cdot (k_{eff} \nabla T) \quad (3)$$

Where ρ is the density, u is the velocity, P is the pressure, τ is the viscous stress, E is the energy and k_{eff} is the effective thermal conductivity. Turbulent flows main feature is changing velocity fields. Because of these, changes can be of small scale and high frequency, they are so computationally affluent to simulate directly in practical engineering calculations. solution process result in a set of equations which contain unknown variables, and turbulence models are used to find these variables in terms of given quantities.

E. Boundary Conditions

A Velocity inlet, uniform mass flow inlets and a constant inlet temperature were assigned at the channel inlet. At the exit, pressure was specified.

S.No	Boundary condition	Outer pipe	Inner pipe
1	Mass flow rate at Inlets	0.134 kg/s	0.134 to 0.267 kg/s
2	Temperatures	300	384
3	Constant heat flux at pipe wall (Insulation)	0	----

F. Method of Solution

The CFD analysis is done by the use of commercial software ANSYS FLUENT 18.1 used for solving thermal problems. After determining the important features of the problem following procedure is followed for solving the problem in which first of all we need to specify the solution method, and initialize the solution, then run the calculation. Initially create geometry model in the ANSYS workbench18.1, as per the experimental set up design. Meshing was done on the geometry model by program controlled and sizing was done to get the required element size, nodes and smoothening. After getting the required size of element and meshing, naming selection was done to the domain before getting the results. After meshing is completed, open the setup in the project schematic in fluent, where governing equations are selected like energy, viscous-standard $k-\epsilon$ (2 eq), standard wall function to be given to necessary equations to simulate, material creating and boundary conditions to be given and methods to calculate the moment, pressure etc., by using standard FEM equations. By selecting second order for solving, finally after converging the equations results were obtained

Fig 1 model of HE created in ANSYS 18.1

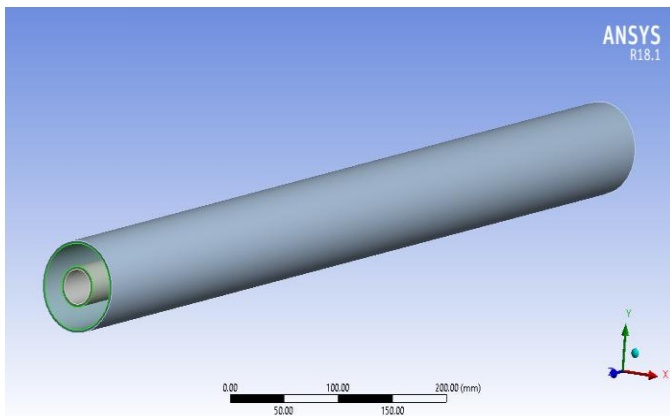
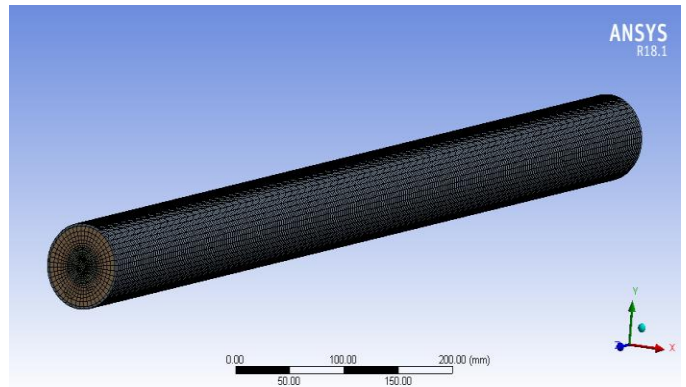


Fig 2 meshed model of HE



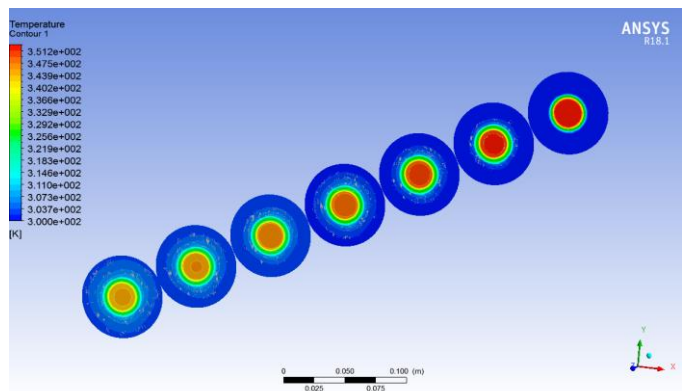
III. RESULTS AND DISCUSSION

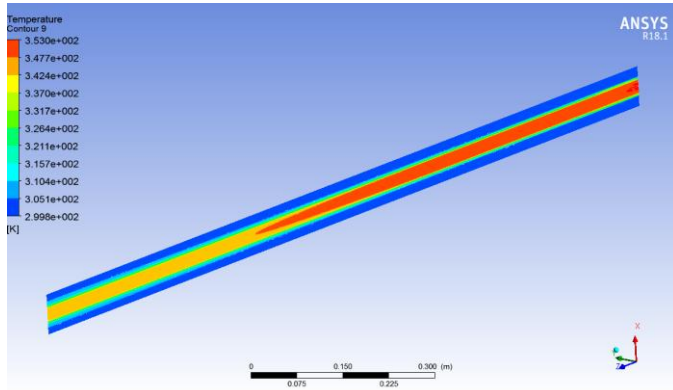
A. Temperature contours

The temperature contours of inside pipe of double pipe heat exchanger are shown from Fig. 3. From the figure it was observed that temperature profile of inside fluid i.e. hot water gradually decreased from inlet to the outlet of inner pipe Fig. 4 shows the temperature contours of outer pipe of double pipe heat exchanger. From the figure it was observed that temperature of annulus fluid i.e., TiO_2 nanofluid gradually increased from inlet to the outlet of pipe.

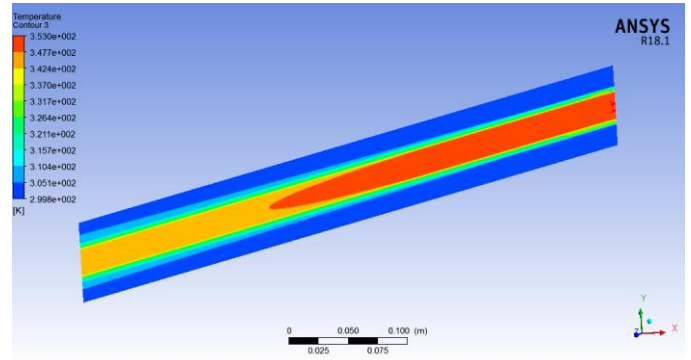
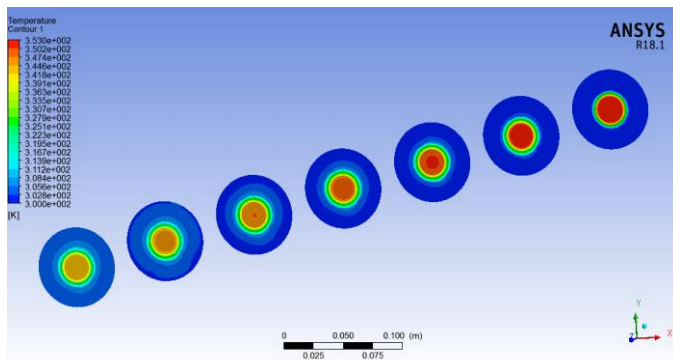
Temperature contours of 4 cases were obtained

1 coolant

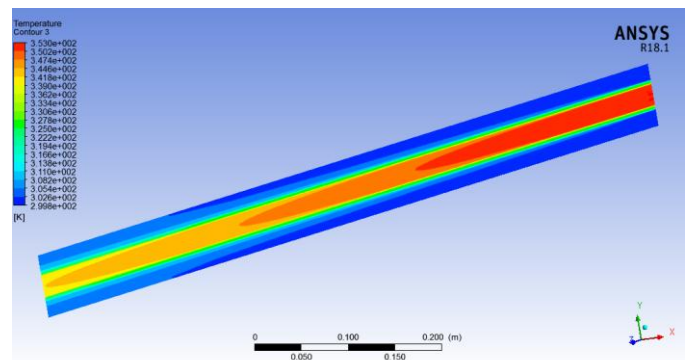
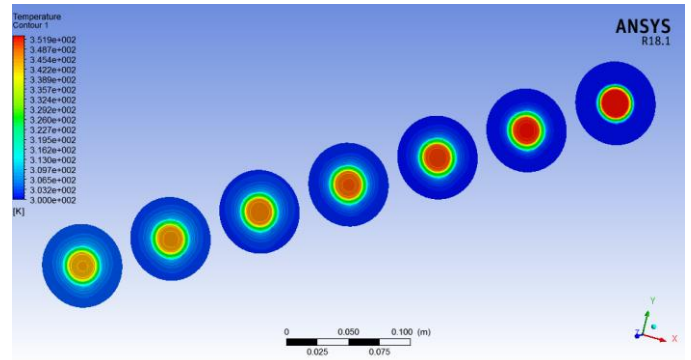




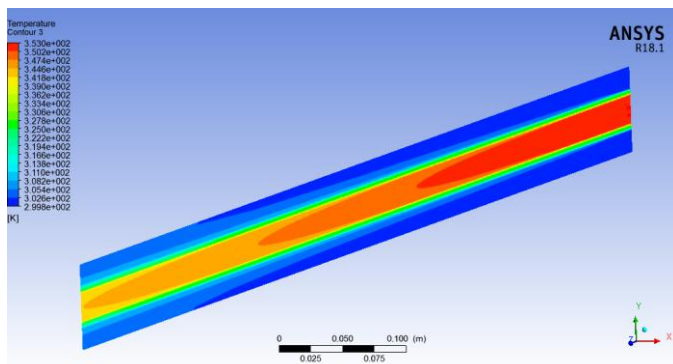
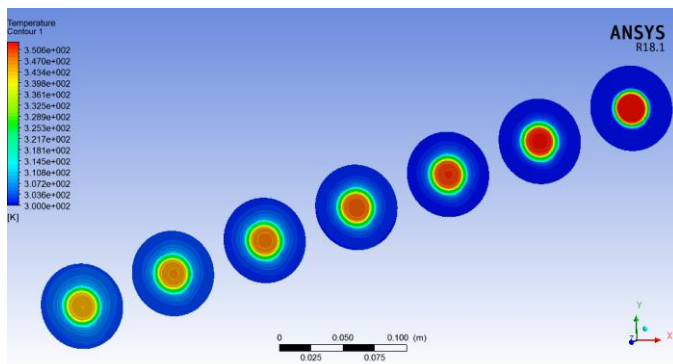
2 coolant (1%TiO₂)



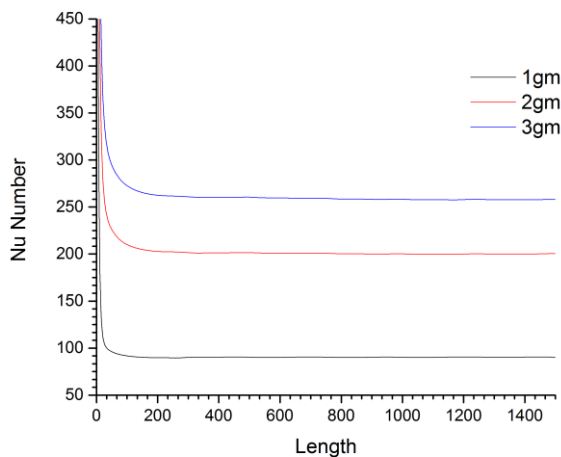
3 coolant (2%TiO₂)



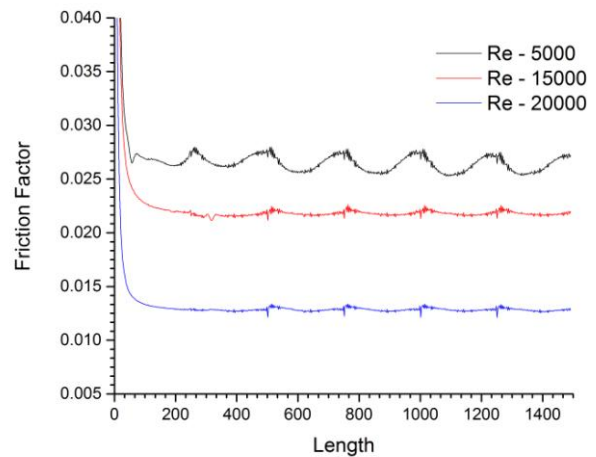
3 coolant (3%TiO₂)



B. Nusslet number vs length



C. Friction factor vs length



IV. CONCLUSIONS

A steady state computational fluid dynamics (CFD) models were simulated by ANSYS FLUENT 18.1 and the effect of change in concentration on the flow behavior of the nanofluid in the pipe were studied and the variations in the properties are presented. The heat transfer enhancement is observed to be better in the turbulent region compared to that of laminar region for all volume fractions considered in the analysis. It is observed that according to simulation results there is a 18% enhancement in heat transfer coefficient at 0.3% volume concentration of TiO₂ nanofluid. Nusselt number saw a significant increase when percentage of TiO₂ increased. The friction factor is increase is proportional with the increase of volume concentration and decrease with increase in velocity but it is inferred that the friction factor enhancement is less compared to the enhancement to the heat transfer for volume fraction considered in the analysis.

Nomenclature

- K Thermal conductivity of the material (Wm-1K-1)
- H Coefficient of convective heat transfer (Wm-2K-2)
- Qc & Qh Heat transfer rates of the cold and hot water (W)
- mc & mh Mass flow rates of cold and hot water (Kg/Sec)
- Cpc & Cph Specific heat of cold and hot water (J/Kg-K)
- Tc-in&Tc-out Inlet and outlet temperatures of cold water(0oC)
- U Overall heat transfer coefficient (W/m2K)
- A Cross sectional area of heat Transfer (m2)\

V	Velocity of water (m2/Sec)	
ρ	Density of water (Kg/m3)	2 Lazarus Godson et al., 2011 "Convective heat transfer of nanofluids with correlations", <i>Particuology</i> , Vol.9 pp 626– 631.
hh & hc	Heat transfer coefficient on hot and cold water side (W/m2K).	3. Liu Yang et al., 2011 "Preparation and stability of Al2O3 suspension of ammonia water solution", <i>Applied Thermal Engineering</i> , Vol.31 pp 3643-3647.
di & do	Inside and outside tube diameter (m)	
Di & Do	Inside and outside diameter of annulus Pipe (m)	
Q	Rate of heat transfer (W)	
Re, Nu &Pr	Reynolds, Nusselt and Prandtl numbers	

REFERENCES

1. Dogan Ciloglu et al., 2011 "The quenching behaviour of aqueous nanofluids around rods with high temperature", *Nuclear Engineering and Design*, Vol.241 pp 2519–2527.