CFD Analysis of a Double Pipe Heat Exchanger by using Fluid Based Nanomaterials

Golkonda Venkata Apparao, K. Srinivasa Rao
Department of Mechanical Engineering, Sri Vaishnavi College of Engineering, Singupuram, Srikakulam, Andhra Pradesh, India

ABSTRACT
Double pipe heat exchanger design is a device which we using for heat transfer applications straightforward. This maintains one heat exchanger pipe inside another. After determining the required heat exchanger surface area, for either countercurrent flow or parallel flow, the pipe sizes and number of bends for the double pipe heat exchanger can be selected.

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 counterflow or parallel flow. Cross flow just doesn’t work for a double pipe heat exchanger. The flow pattern and the required heat exchange duty allow 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.

The convective heat transfer, friction factor and effectiveness of different volume concentrations of Fe3O4 Nano fluid flow in an inner tube of double pipe heat exchanger with return bend has been estimated experimentally and turbulent flow conditions.

The test section used in this study is of double pipe type in which the inner tube diameter is 0.019 m, the annulus tube diameter is 0.05 m and the total length of inner tube is 5 m. At a distance of 2.2 m from the inlet of the return tube the return bend is provided.

The hot Nanofluids flows through an inner tube, whereas the cold water flows through an annulus tube. The volume concentrations of the nanoparticles used in this study are 0.03% and 0.18%. The flow rate is 100 m³/h with Reynolds number range from 9,000 to 30,000. In the process analyze the CFD analysis is performed in Ansys Fluent 15.0 workbench and its used different nanofluids, which nanofluids better heat transfer rate find base on the results. In this process choose different nanofluids(Al2O3, CuO, Fe3O4) this nanofluids properties taken by different base papers.

KEYWORDS: Heat Exchanger, Nano Fluids, Thermo Physical Properties, Heat Transfer, CFD Analysis

I. INTRODUCTION
A heat exchanger is a device used to transfer heat between one or more fluids. The fluids may be separated by a solid wall to prevent mixing or they may be in direct contact. They are widely used in space heating, refrigeration, air conditioning power stations, chemical plants, petrochemical plants, petroleum refineries, natural-gas processing, and sewage treatment. The classic example of a heat exchanger is found in an internal combustion engine in which a circulating fluid known as engine coolant flows through radiator coils and air flows past the coils, which cools the coolant and heats the incoming air.

Double pipe heat exchangers are the simplest exchangers used in industries. On one hand, these heat exchangers are cheap for both design and maintenance, making them a good choice for small industries. On the other hand, their low efficiency coupled with the high space occupied in large scales, has led modern industries to use more efficient heat exchangers like shell and tube or plate.

However, since double pipe heat exchangers are simple, they are used to teach heat exchanger design basics to students as the fundamental rules for all heat exchangers are the same. To start the design of a double pipe heat exchanger, the first step is to calculate the heat duty of the heat exchanger. It must be noted that for easier design, it’s better to ignore heat loss to the environment for initial design.

Nano fluid is a new kind of heat transfer medium, containing nanoparticles (1-100 nm) which are uniformly and stably distributed in a base fluid. These distributed nanoparticles generally metal or metal oxides greatly enhance the thermal conductivity of the Nano fluid, increases conduction and convection coefficients, allowing for more heat transfer. The nanoparticle material includes chemically stable metals [e.g., gold, copper], metal oxides (e.g., alumina, silica, zirconia, and titania), oxide ceramics (e.g., Al2O3, and CuO), metal carbides (e.g., SiC), metal nitriles (e.g., AlN, SiN), carbon in various forms (e.g., diamond, graphite, carbon nanotubes, and fullerenes) and other functionalized nanoparticles. The Nano fluids have various applications as they provide an efficient thermal energy transfer due to higher heat transfer with a comparable pumping power required.

II. REVIEW OF LITERATURE
N. T. Ravi kumar [1] in their paper discussed that the convective heat transfer, friction factor and effectiveness of different volume concentrations of Fe3O4 nanofluid flow in an inner tube of double pipe heat exchanger with return bend has been estimated experimentally and turbulent flow conditions. The test section used in this study is of double pipe type in which the inner tube diameter is 0.09 m, the annulus tube diameter is 0.05 m and the total length of inner tube is 5 m. At a distance of 2.2 m from the inlet of the inner tube the return bend is provided. The hot Fe3O4 nanofluid...
flows through an inner tube, where as the cold water flows through an annulus tube. The volume concentration of the nanoparticles used in this study are 0.005%, 0%, 0.03% and 0.06% with Reynolds number range from 15,000 to 30,000.

Based on the results, the Nusselt number enhancement is 14.7% for 0.06% volume concentration of nanofluid flow in an inner tube of heat exchanger at a Reynolds number of 30,000 when compared to base fluid data; the pumping penalty of nanofluid is b10%. New correlations for Nusselt number and friction factor have been developed based on the experimental data.

Arjum and Adi[2] in his paper described that the Mini channel flow of Nano fluids has been predicted considering Nano fluid as a single phase homogeneous mixture. The homogeneous mixture model for the Nano fluid holds good to predict the average Nusselt number and friction factor in case of laminar flow. Hence, the presentcomputational model can be a good alternative approach to predict heat transfer and pressure drop characteristics of mini channel flow using Nano fluids. Also, the error in prediction of Nusselt number is less if we consider Brownian motion in our computational model, which is within 4%.

M. Esfandiary[3] in their study discussed that the problem of turbulent forced convection flow of water-alumina Nano fluid in a uniformly heated pipe has been thoroughly investigated. In numerical study, single and two-phase models have been used. In single-phase modeling of Nano fluid, thermal and flow properties of Nano fluid have been considered to be dependent on temperature and volume fraction. Effects of volume fraction and Reynolds number (3000<Re<9000) on convective heat transfer coefficient and pressure drop were investigated for various axial locations of the tube.

III. MODELING OF DOUBLE PIPE HEAT EXCHANGER

The modeling of a double pipe heat exchanger is done in CATIA V5R20 modeling software.

Select the file  click on model properties  assign the material properties, system of units, accuracy click on ok.

IV. COMPUTATION ANALYSIS

Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reaction (e.g., combustion), and related phenomena by solving the mathematical equations that govern these processes using a numerical algorithm on a computer. The technique is very powerful and spans a wide range of industrial and non-industrial application areas.

All the CFD codes contain three main elements. They are as follows,
- Preprocessor
- Solver
- Post processor

The geometry is created in ANSYS ICEM CFD as per the given data for each of the model and a domain is created to encompass the flow inside the domain to the walls of the body. In order to study domain independence, three cylindrical domains are considered to trial and error method taking the distances from nose and tail ends of the model and taking the radius from the axis of the model. Three dimensional hexahedral grids were generated to discretize the body and the domain.

Three dimensional segregated implicit solvers is used in the present analysis, the k-ω, k-ε turbulence models in addition to the continuity and momentum equations were used as governing equations. Boundary conditions used in the present analysis are inlet as velocity inlet, outlet as Pressure Outlet, far field, and body as walls. All the three models are computed in the solver Fluent. The solution was stopped when changes in solution variables from one iteration to the next is negligible. Solution is iterated till the convergence is observed. Then forces and moments results were extracted from it. This data is saved as the data file in the solver itself.

Solve the momentum equations to find the velocity components (U0, Vo).

Solve the pressure-corrrection equation to find 'p' at each grid point.

Replace the previous intermediate values of pressure and velocity with the new corrected values and return to the original step. Repeat the step until the step converges.

A. Geometry and Domain are created in ANSYS 15.0. Blocking and Meshing is done. Checking the mesh quality and saving the file to solver Fluent. Export it into fluent software. Computing and monitoring the solution in Fluent. Examine and save the results.

B. pressure distribution

![Image](Fig1 0.034% volume concentration for 8 lpm pressure distribution Fe\textsubscript{3}O\textsubscript{4})
Fig 2. 0.034% volume concentration for 10 lpm pressure distribution CuO

Fig 3. 0.034% volume concentration for 10 lpm pressure distribution Al₂O₃

Fig 4. 0.034% volume concentration for 10 lpm pressure distribution Fe₂O₄
In the above graph represent by pressure vary from point to point on y-axis, taken different Nano fluids but same volume concentration(0.03%) and different mass flow rate(8 lpm) litblue colour and (10 lpm) represent by red colour. Aluminium oxide < copper oxide < ferrous oxide.

**CONCLUSION**

The convective heat transfer performance and flow characteristics of CuO, Al2O3, Fe3O4 nano fluid flowing in a double pipe heat exchanger has been experimentally investigated. Experiments have been carried out under turbulent conditions. The effect of particle concentration and the Reynolds number on the heat transfer performance and flow behavior of the nanofluid has been determined. Dispersion of the nano particles into the base fluid (water) increases the average heat transfer coefficient with the increase in the rate of fluid. Taken three nanofluids obeyed by thermal physical properties(CuO, Al2O3, Fe3O4), CuO Heat transfer rate 8 lpm heat transfer rate 4.797.793 (w), pressure 2.52e+03(pa), temperature 3.07e+02(k), velocity distribution 2.35e+01(m/s), Aluminium oxide heat transfer rate 8 lpm heat transfer rate 9658.334 (w), pressure 2.66e+03(pa), temperature 3.10e+02(k), velocity distribution 6.1e+01(m/s) ferrous oxide heat transfer rate 8 lpm heat transfer rate 12829.021 (w), pressure 3.41e+03(pa), temperature 3.33e+02(k), velocity distribution 7.36e+01(m/s) CUO Heat transfer rate 10 lpm heat transfer rate 15047.63 (w), pressure 3.81e+03(pa), temperature 3.20e+01(k), velocity distribution 2.47e+02 (m/s) Aluminium oxide Heat transfer rate 10 lpm heat transfer rate 17102.24 (w) pressure 2.73e+02(pa), temperature 3.00e+01(k), velocity distribution 6.22e+02(m/s) ferrous oxide Heat transfer rate 10 lpm heat transfer rate 24761.22 (w) pressure 4.27e+03(pa), temperature 3.33e+04(k), velocity distribution 7.56e+02(m/s). (0.03%) volume concentration and 8 lpm mass flow rate copper oxide < Aluminium oxide < ferrous oxide. (0.03%) volume concentration and 10 lpm mass flow rate, y-axis taken by heat transfer rate. copper oxide < Aluminium oxide < ferrous oxide. Fe3o4 is better nanofluid.

**REFERENCES**


[2] Byung-Hee Chun, Hyun Uk Kang, and Sung Hyun Kim Effect of alumina nanoparticles in the fluid on heat transfer in double-pipe heat exchanger system Department of Chemical and Biological Engineering, Applied Rheology Center, Korea University, Seoul 136-701, Korea (Received 5 October 2007 • accepted 5 March 2008)


