WWW.IJRPR.COM

International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

A Review Heat Exchanger with Different Nano Fluid

Prabhash Kumar Jha[1], Surjeet Singh Rajpoot[2]

[1](M. Tech Scholar), SCOPE College of Engineering, Bhopal
[2]Associate Professor, SCOPE College of Engineering, Bhopal

ABSTRACT:

For transferring heat between two or more process fluids is a heat exchanger. Heat exchangers are used extensively in both household and industrial settings. For usage in steam power plants, chemical processing facilities, building heating and cooling systems, transportation power systems, and refrigeration units, several types of heat exchangers have been created. The design of heat exchangers itself is a challenging issue. It goes beyond just looking at heat transmission. From a cost of ownership perspective, the price of manufacturing and installation, weight, and size all have a significant impact on the choice of the final design. Although cost is frequently a crucial aspect, size and footprint frequently seem to be the deciding elements when picking a design.

Keyword: **Heat exchanger, heat, turbulent flow, efficiency, heat transfer**

Introduction:

A heat exchanger is a device used to transfer heat between two or more fluids. Heat exchangers are used in both cooling and heating processes. 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. Another example is the heat sink, which is a passive heat exchanger that transfers the heat generated by an electronic or a mechanical device to a fluid medium, often air or a liquid coolant.

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.

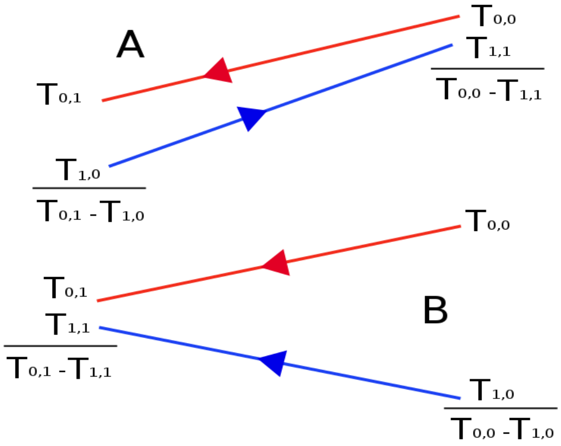

Fig.1 Parallel & Counter Flow Arrangement

In addition to heating up or cooling down fluids in just a single phase, heat exchangers can be used either to heat a liquid to evaporate (or boil) it or used as condensers to cool a vapor and condense it to a liquid. In chemical plants and refineries, re-boilers used to heat incoming feed for distillation towers are often heat exchangers. Distillation set-ups typically use condensers to condense distillate vapors back into liquid. Power plants that use steamdriven turbines commonly use heat exchangers to boil water into steam. Heat exchangers or similar units for producing steam from water are often called boilers or steam generators.

There are three main types of flows in a spiral heat exchanger:

- Counter-current Flow: Fluids flow in opposite directions. These are used for liquid-liquid, condensing and gas cooling applications. Units are usually mounted vertically when condensing vapour and mounted horizontally when handling high concentrations of solids.
- Spiral Flow/Cross Flow: One fluid is in spiral flow and the other in a cross flow. Spiral flow passages are welded at each side for this type of spiral heat exchanger. This type of flow is suitable for handling low density gas, which passes through the cross flow, avoiding pressure loss. It can be used for liquid-liquid applications if one liquid has a considerably greater flow rate than the other.
- Distributed Vapour/Spiral flow**:** This design is that of a condenser, and is usually mounted vertically. It is designed to cater for the sub-cooling of both condensate and non-condensable. The coolant moves in a spiral and leaves via the top. Hot gases that enter leave as condensate via the bottom outlet.

Nano Fluids

The thermal conductivity of heating or cooling of fluids is very important property for the development of energy efficient heat transfer equipment. Meanwhile, in all the processes involving heat transfer, the thermal conductivity of the fluid is one of the basic important properties taken in to account in designing and controlling the processes. Nanofluids are engineered colloids which are made of a base fluid and nanoparticles of (1-100) nm. It has been found by many researchers that the nanofluids provide higher thermal conductivity compared to base fluids. Its value increases with the increase in particle concentration, temperature, particle size, dispersion and stability. Nevertheless, it is expected that other factors like density, viscosity, and specific heat are also responsible for the convective heat transfer enhancement of nanofluids. Nanofluids are having high thermal conductivity and high heat transfer coefficient compared to single phase fluids.

A nanofluid is prepared by dispersing particles of metal or metal oxide with sizes of 100 nm or less, in a base liquid such as water. The purpose of using nanofluids is to achieve higher values of heat transfer coefficient compared with that of the base liquid. This is achieved by the dispersion of solid particles, which have higher thermal conductivity than the base liquid. There are many engineering applications that can benefit from the use of nanofluids, for example absorption refrigeration, micro electromechanical systems, lubrication of automotive systems, the manufacture of advanced miniature camera lenses, coolant in machining, automobile radiator cooling, personal computers, solar water heating, heat exchangers, several medical applications, nuclear reactors, and in several aerospace applications. Recent advances in material technology have made it possible to produce innovative heat transfer fluids by suspending nanometer-sized particles in base fluids, which could change the transport and thermal properties of the liquids.

NFs thermal applications

Cooling is one of the most essential scientific challenges in different industries Therefore, among all applications of NFs, their potential for heat transfer applications have attracted the most attentions. Some capability of NFs for thermal applications. NFs can be used in transport systems such as automotive and automobile radiator. In metal processing they can be utilized in metal cutting. They could also be used as efficient coolant in data centres and electronics cooling systems.

LITERATURE REVIEW

Ferng et al. worked on the effect of changing the Dean Number and pitch of the helically coiled tube on heat transfer. They revealed that the creation of secondary flow becomes weaker when increasing the coiled tube pitch.

Narrein and Mohammed numerically studied the effect of Al2O3, SiO2, CuO, ZnO concentration and size of the nanoparticles, different base fluids such as water, ethylene glycol and engine oil on the heat transfer and fluid flow characteristics. They suggested the nanofluids based on the SiO2 has higher pressure drop than other Al2O3, SiO2, CuO, ZnO nanofluids.

A.J. Shkarah et al A nanofluid is a mixture of a low concentration of solid particles (10-100nm in size at concentrations below 10%vol.) and a carrier fluid (usually conventional coolants). These novel fluids exhibit anomalous heat transfer phenomena which cannot be explained using classical thermodynamic models. The fluids can be designed to offer unsurpassed heat transfer rates for heat transfer related applications at low costs of manufacturing.

L. Boelter et al performed an experimental study of turbulent heat transfer and separation flow in a symmetric expansion plane channel. The experiments were carried out with a Reynolds number that varied from 5000 to 35,000, and at 20 mm height and 200 mm width of the step.

L. Khezzar et al employed standard k-ε model in their study of turbulent flow and heat transfer in an abrupt pipe expansion. Chung and used a new k-ϵ model in their numerical study of turbulent heat transfer in an abrupt expansion whereas they obtained augmentations in calculated turbulent kinetic energy and velocity.

Objective of work:

1. Apply the approach to assess how well a pair of rectangular winglets positioned at various points inside a triangular fin increase heat transmission.

2. To determine the bulk temperature and combined average Nusselt number for heat transmission characteristics. The bulk temperature is a direct indicator of thermal energy, while the Nusselt number measures the efficiency of heat transport.

3. Shorten the heat exchanger's length.

Computational Fluid Dynamics

CFD is useful for studying fluid flow, heat transfer; chemical reactions etc by solving mathematical equations with the help of numerical analysis. CFD resolve the entire system in small cells and apply governing equations on these discrete elements to find numerical solutions regarding pressure distribution, temperature gradients. This software can also build a virtual prototype of the system or device before can be apply to real-world physics to the model, and the software will provide with images and data, which predict the performance of that design. More recently the methods have been applied to the design of internal combustion engine, combustion chambers of gas turbine and furnaces, also fluid flows and heat transfer in heat exchanger. The development in the CFD field provides a capability comparable to other Computer Aided Engineering (CAE) tools such as stress analysis codes. Basic Approach to using CFD.

The body about which flow is to be analyzed requires modeling. Approximations of the geometry and simplifications may be required to allow an analysis with reasonable effort. Concurrently, decisions are made as to the extent of the finite flow domain in which the flow is to be simulated. Portions of the boundary of the flow domain coincide with the surfaces of the body geometry. Other surfaces are free boundaries over which flow enters or leaves. The geometry and flow domain are modeled in such a manner as to provide input for the grid generation. Thus, the modeling often takes into account the structure and topology of the grid generation.

Method of Solution

The CFD method follows the use of commercial software ANSYS FLUENT 15.0 for solving the problem. The solver in ANSYS-FLUENT used is a pressure correction based SIMPLE algorithm with 2ndorder upwind scheme for discretise the convective transport terms. The criteria for convergence dependent variables are specified as 0.001. In the present analysis, the analytical values of heat transfer coefficients are calculated. The heat transfer coefficients are also obtained using CFD methods and compared with analytical values. 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 workbench, as per the experimental setup design.

Conclusion:

Heat exchangers are used to transfer heat from a hot fluid to a cool fluid. They play a crucial role in the process by adjusting the temperature of the more precious fluid that will be utilized later on. We get the conclusion that the heat transfer in the counter-flow configuration is larger than the heat transfer in the parallel flow after doing this experimental computation. The real values given in the literature are different from the experimental values for several parameters.

References:

[1] W.R. Dean, Note on the motion of fluid in A curved pipe, Phil. Mag. Ser. 7 (4) (1927) 208– 223.

[2] W.R. Dean, The streamline motion of fluid in A curved pipe, Phil. Mag. Ser. 7 (5) (1928) 673– 695.

[3] K. Palanisamy, P.C. Mukesh Kumar, Experimental investigation on convective heat transfer and pressure drop of cone helically coiled tube heat exchanger using carbon nanotubes/ water nanofluids, Heliyon 5 (2019) e01705.

[4] Y.M. Ferng, W.C. Lin, C.C. Chieng, Numerically investigated effects of different dean number and pitch size on flow and heat transfer characteristics in a helically coil-tube heat exchanger, Appl. Therm. Eng. 36 (2012) 378–385.

[5] K. Narrein, H.A. Mohammed, Influence of nanofluids and rotation on helically coiled tube heat exchanger performance, J. Comput. Theor. Nanosci. 11 (2014) 911–927.

[6] T. Hussein, T. Abdulrazzaq, S.N. Kazi, A. Badarudin, A.A.H. Kadhum, E. Sadeghinezhad, A review of studies on forced, natural and mixed heat transfer to fluid and nanofluid flow in an annular passage, Renew. Sust. Energ. Rev. 39 (2014) 835–856.

[7] L. Boelter, G. Young, H.W. Iversen, An Investigation of Aircraft Heaters XXVII—Distribution of Heat Transfer Rate in the Entrance Section of a Circular Tub. NACA-TN-1451, 1948.

[8] L. Khezzar, S.R.N. De Zilwa, J.H. Whitelaw, Combustion of premixed fuel and air downstream of a plane sudden-expansion, Exp. Fluids 27 (1999) 296–309.

[9] S. De Zilwa SR, J.H. Whitelaw Sivasegaram, Active control of isothermal and combusting flows in plane sudden-expansions, Proc. Transp. Phenom. Thermal Sci. Process Eng. (1997) 325–330.

[10] S.K. Park, T. Ota, An experimental approach to turbulent heat transfer using a symmetric expanded plane channel, J. Mech. Sci. Technol. 24 (2010) 857–863.

[11] C.C. Chieng, B.E. Launder, On the calculation of turbulent heat transport downstream from an abrupt pipe expansion, Numer. Heat Transfer 3 (1980) 189–207.