International Journal of Innovative Research and Reviews, 2(2):24-27

International Journal of Innovative Research and Reviews

ISSN: 2636-8919

Website: www.injirr.com

doi:

Research paper, Short communication, Review, Technical paper

RESEARCH ARTICLE

Numerical Analysis of Heat Transfer of Hot Oil and Cold Water Fluids in a Concentric Type Heat Exchanger with Ansys Fluent

Mansour NASIRI KHALAJI[1], Muhammet Harun OSTA[1*], Kenan YAKUT[1]

[1]*Ataturk University Faculty of Engineering Mechanical Engineering Department, Erzurum, TURKEY*

**Corresponding author E-mail: harun.osta@atauni.edu.tr*

HIGHLIGHTS

> Today's industry requires the removal of high heat accumulation from small surfaces in order to eliminate the efficiency of large machines, and heat transfer rates between the higher two fluids are needed.

> In such systems, different flow directions have been proposed for cooling the fluid.

ARTICLE INFO

Received : 09.11.2018
Accepted : 11.20.2018
Published : 12.15.2018

Keywords:
Engine oil,
CFD,
Heat exchanger,
ANSYS Fluent

ABSTRACT

In this study, the heat transfer performance of a trunk pipe heat exchanger using motor oil as hot fluid and cold water as cold fluid was analyzed by a computational fluid dynamic (Ansys Fluent) program. The engine oil heat exchanger at different temperature ranges (70°C) and at different velocity (0.1m/s). Same case, the cold water at different temperature ranges of the heat exchanger (5°C) and different speed (1m/s) enters. The numerical results and mesh validations are shown below.

Contents

1. Introduction

Today's most current and popular problems are energy and economic problems. These two problems have a high degree of relationship with each other and directly affect each other. Energy and economic issues lie at the heart of many global actions. Especially the acquisition of energy from fossil sources and the control of these resources shape today's global politics. The most important place among the existing energy sources is to keep petroleum products and a part of Petroleum products are used as raw material in production but a large part is used as fuel. Galeazzo et al. (2006), computational fluid in his study in 2006 of a flat plate for use in a 4-channel dynamic prototype heat exchanger was performed. Parallel and serial streams have been tested [1]. Experimental data are compared with the thermal loads obtained from 3D CFD model with 1-D plug flow. The CFD model shows the ducts, plates and water lines in the heat exchanger. In the model, uneven flow distributions between the channels and bad flow distributions in the channel are included in the calculation. A good arrangement has been made so that the CFD results will be serialized with experimental

Cite this article Nasiri Khalaji M, Osta MH, Yakut K. Numerical Analysis of Heat Transfer of Hot Oil and Cold Water Fluids in a Concentric Type Heat Exchanger with Ansys Fluent. *International Journal of Innovative Research and Reviews (INJIRR)* (2018) 2(2):24-27

Link to this article: http://www.injirr.com/article/view/13

data. Experimental data were compared with the one-dimensional plug flow model prepared in Many different working fluids are used in YGS systems [2, 3] in the 1D plug flow model of the study. By Terekhov and Pakhomov (2002), a model was developed to estimate mass and heat transfer in the form of gas, vapor, and droplets as two phase streams on an isothermal flat plate [4]. Because of the complex geometries and non-linear dynamic behavior of the heat exchangers, most models require assumptions and simplifications. Fixed property values, constant heat transfer coefficient and similarity states are some of these assumptions [5]. In his study, Wang et al. (2009) numerically studied convection heat transfer in a vibrating turbulent flow at a constantly swinging width to a constant wall temperature boron [6].

In today's machines, the main machine is used as oil cooling in water cooling. Many industrial processes involve heat transfer using heat transfer fluids such as water, anti-freeze and engine oil. For this reason, many different techniques are used to improve the thermal properties of these fluids, especially thermal conductivity. Heat exchangers are devices that provide heat exchange between two fluids with different temperatures. Heat exchangers are widely used in a wide range of applications, from heating and ventilation systems in homes to chemical processing and power generation in large plants.

2. Material and Method

Different heat transfer applications require different types of equipment and heat transfer devices in different layouts. The attempts to find the heat transfer equipment to meet the requirements of heat transfer with certain limitations have revealed numerous new types of heat exchanger design. The simplest type of heat exchanger, as shown in Figure 1, consists of two pipes with different coaxial diameters and is called a double-tube heat exchanger. In the double tube heat exchangers, a fluid flows through the small pipe, while the other fluid flows through the ring between the two tubes. There are two types of flow patterns in the double tube heat exchanger: both the hot and cold fluid in the parallel flow enters the heat exchanger on the same side and move in the same direction. On the other hand, in the reverse flow, the hot and cold fluids enter the heat exchanger from the opposite side and move in the opposite direction.

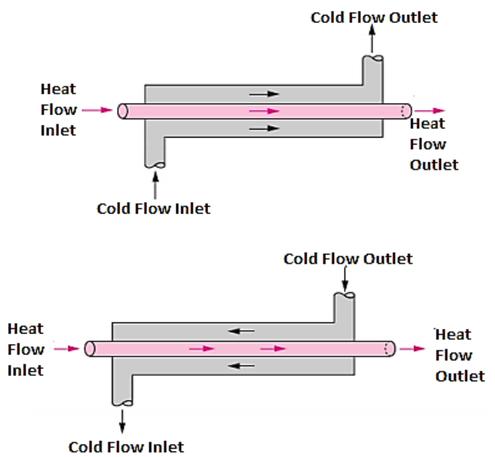

Figure 1 Different flow regimes and related temperature profiles in a double tube heat exchanger

Another heat exchanger specially designed to achieve a high heat transfer surface area per unit volume is the compact heat exchanger. Compact heat exchangers ensure high heat transfer rates between a small volume of two fluids, and are generally used in heat exchanger applications with significant limitations in weight and volume.

In compact heat exchangers, the large surface area is obtained by adding a thin plate or slotted wings to the two fluid separating walls. Compact heat exchangers are generally used in gas-gas and gas-liquid heat exchangers to prevent the low heat transfer coefficient due to the gas flow in the expanded surface area.

The behavior of fluids is often not reasonable, and if this is not impossible, the product makes it difficult to predict the effect of fluid flow. ANSYS CFD provides tools that enable you to simulate the behavior of fluid flows - even with complex interactions between one or the other, and to reliably analyze the results, as well as the results during design and production. ANSYS 'industry-leading fluid dynamics (CFD) solutions, together with chip-level thermal integrity simulation software, provide everything needed to perform electronic cooling simulation and thermal analysis for chip-package, PCB and systems. You can also do thermomechanical stress analysis and airflow analysis to choose the ideal heat sink or fan solution. Integrated workflow provides enhanced reliability and performance by enabling you to perform design balancing.

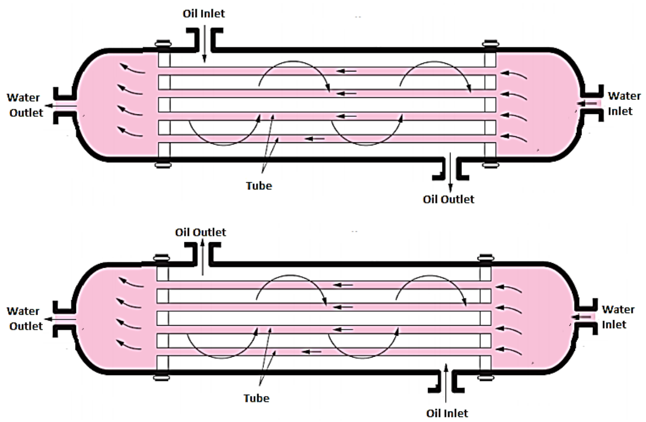

Figure 2 Schematic representation of the body tube heat exchanger (with a trunk bore and a pipe passage)

2.1. Mesh validation

Our boundary conditions shall be between 70°C temperatures and 0.1 m / s flow, cold water shall be at 5°C temperature and 1 m / s shall be heat exchanger. The flow is opposite in the same direction and in the opposite direction. The insulation of the heat exchanger is considered an adiabatic system, assuming it is excellent.

In this study, 5 different numerical models are used and these models have 5*10^6, 10*10^6, 15*10^6, 20*10^6, 25*10^6, element number respectively. The outlet temperature of the melt salt was determined by reference to the experimental results.

The maximum number of elements to choose is 20*10^6, since after this point the temperature change does not show a significant increase and at the same time increases the duration of the numerical analysis as the number of elements increases. The digital network image of the model is shown in Figure 3.

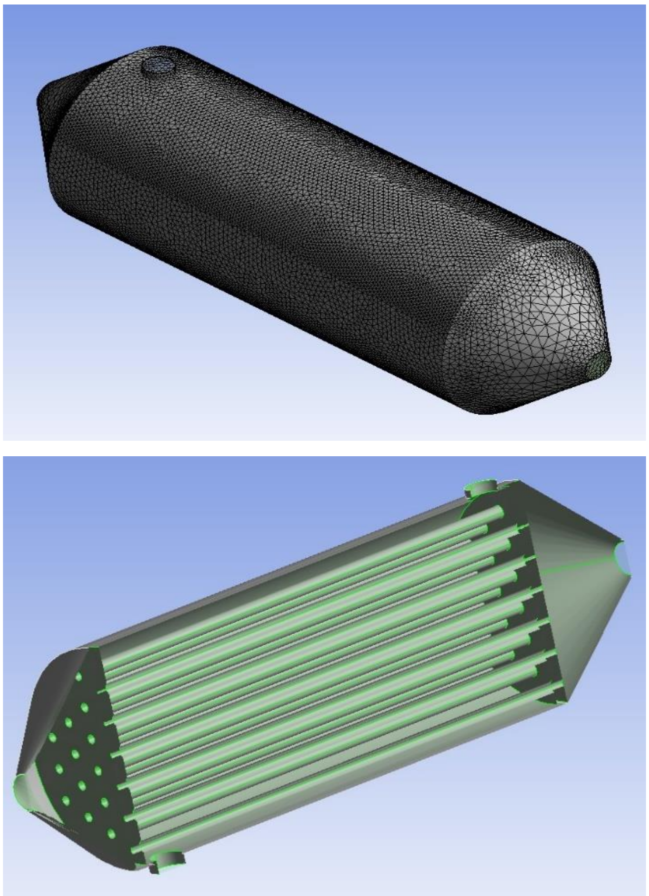

Figure 3 Digital network image of the selected model

Figure 4 is the graph showing the ratio of the number of elements to experimental and numerical results.

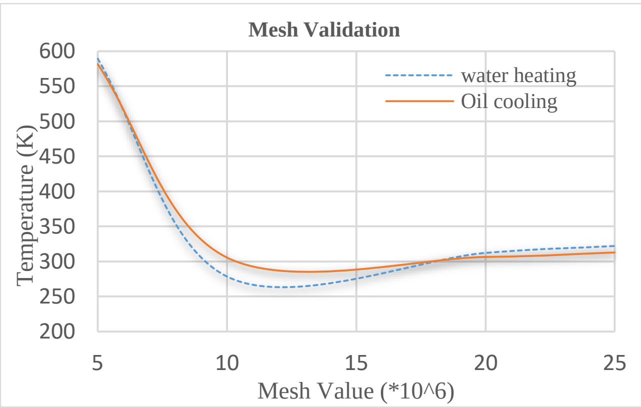

Figure 4 Representation of the independence value of the hot oil outlet temperature from different element numbers

The hot oil flows at very low flow rates and exhibits a laminar flow. The other flowing water flows in the turbulent area. The turbulence model used for water is the standard k-ϵ model. The standard k-ϵ model, which is generally used in literature, is also used in this study. Other turbulence models have been tried but the best result is the k-ϵ model.

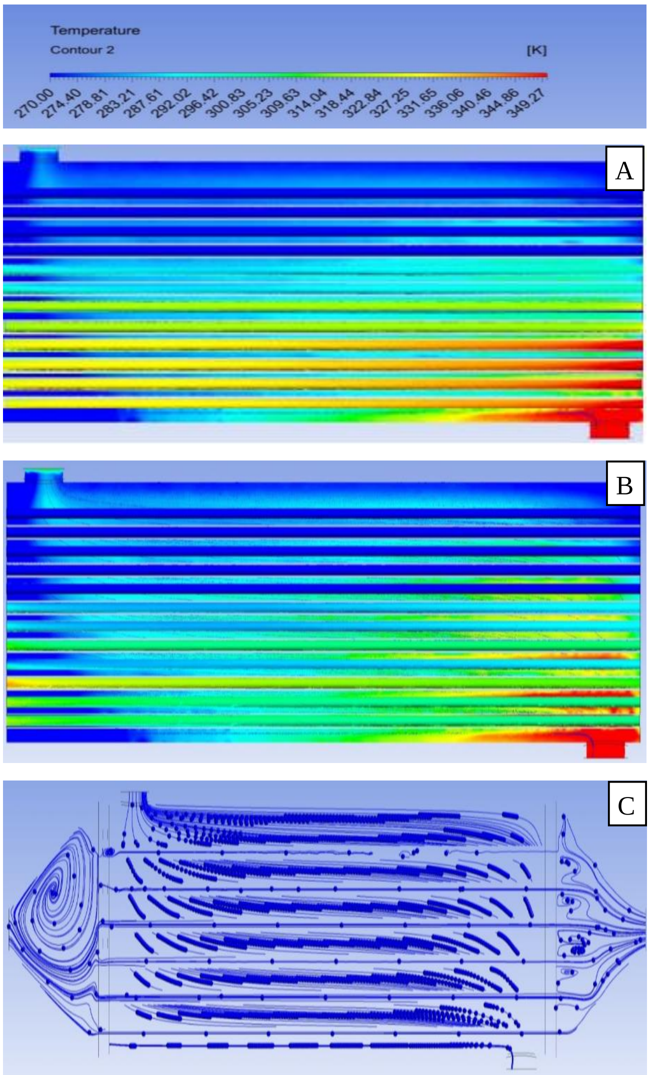

Figure 5 A) Temperature distribution for opposite flow B) Temperature distribution for parallel flow C) Flow rate

3. Results and Discussion

Two different fluid directions were added. When the temperature distributions in the numerical calculations are examined, the temperature difference at the outlet will change as the cold water or hot oil discharge the flow. After the mesh geometry was drawn, mesh verification was performed and given in the diagram. also as shown in Figure 5-A and B, the heat of the opposite flow, according to the parallel flow, transferred the heat to the cold water well and this means that the heat transfer is better. Figure 5-C shows the distribution of the opposite flow.

4. Conclusions

Computational fluid dynamics package with Ansys Fluent analysis results; Instead of establishing high cost and long-time experiment systems, three dimensional complex flow problems can be solved in a short time by high performance calculations with computational fluid dynamics.

References

[1] Galeazzo FCC, Miura RY, Gut JAW, Tadini CC. Experimental and numerical heat transfer in a plate heat exchanger. *Chemical Engineering Science* (2006) **61**(21):7133–7138. doi:10.1016/j.ces.2006.07.029.

[2] Mussard M, Nydal OJ. Comparison of oil and aluminum-based heat storage charged with a small-scale solar parabolic trough. *Applied Thermal Engineering* (2013) **58**(1-2):146–154. doi:10.1016/j.applthermaleng.2013.03.059.

[3] Pacio J, Singer C, Wetzel T, Uhlig R. Thermodynamic evaluation of liquid metals as heat transfer fluids in concentrated solar power plants. *Applied Thermal Engineering* (2013) **60**(1-2):295–302. doi:10.1016/j.applthermaleng.2013.07.010.

[4] Terekhov VI, Pakhomov MA. Numerical study of heat transfer in a laminar mist flow over a isothermal flat plate. *International Journal of Heat and Mass Transfer* (2002) **45**(10):2077–2085. doi:10.1016/S0017-9310(01)00318-0.

[5] Diaz GC. *Simulation and Control of Heat Exchangers Using Artificial Neural Networks*. PhD Thesis. University of Notre Dame. Notre Dame, Indiana (2000) [cited 2018 Dec 17]. 179 p. Available from: https://pdfs.semanticscholar.org/4e50/8b4823dd65c082be1f4e864df64d8a0fe3d1.pdf.

[6] Wang Q-W, Lin M, Zeng M. Effect of lateral fin profiles on turbulent flow and heat transfer performance of internally finned tubes. *Applied Thermal Engineering* (2009) **29**(14-15):3006–3013. doi:10.1016/j.applthermaleng.2009.03.016.