ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Heat Transfer Performance of Radiator Using Different Nano Fluids

Ungarala Santosh Bharath
M.Tech Student
Department of Mechanical Engineering
Kaushik College of Engineering.

N.Hymavathi
Assistant Professor
Department of Mechanical Engineering
Kaushik College of Engineering.

ABSTRACT

In cooling system of automobile engine the water is evaporate at high temperature, so we need to add water and also water is low capacity of absorb heat. Cooling system plays important roles to control the temperature of car's engine. One of the important elements in the car cooling system is cooling fluid. The usage of wrong cooling fluid can give negatives impact to the car's engine and shorten engine life. An efficient cooling system can prevent engine from overheating and assists the vehicle running at its optimal performance.

In this thesis, different nano fluids mixed with base fluid water are analyzed for their performance in the radiator. The nano fluids are Aluminum Oxide, Silicon Oxide, Titanium carbide and Titanium nitride for two volume fractions 0.2, 0.3. Theoretical calculations are done determine the properties for nano fluids and those properties are used as inputs for analysis.

3D model of the radiator is done in Pro/Engineer. CFD analysis is done on the radiator for all nano fluidsby taking two volume fractions and thermal analysis is done in Ansys fortwo materials Aluminum and Copper for better fluid at better volume fraction from CFD analysis.

INTRODUCTION TO AUTOMOBILE RADIATOR

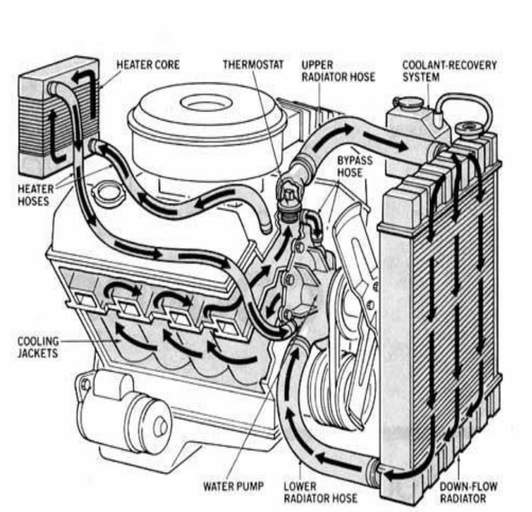
Radiators are heat exchangers used to transfer thermal energy from one medium to another for the purpose of cooling and heating. The majority of radiators are constructed to function in automobiles, buildings, and electronics. The radiator is always a source of heat to its environment, although this may be for either the purpose of heating this environment, or for cooling the fluid or coolant supplied to it, as for engine cooling. Despite the name, radiators generally transfer the bulk of their heat via convection, not by thermal radiation, though the term "convector" is used more narrowly

WORKING OF AUTOMOBILE RADIATORS

Almost all automobiles in the market today have a type of heat exchanger called a radiator. The radiator is part of the cooling system of the engine as shown in Figure below. As you can see in the figure, the radiator is just

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

one of the many components of the complex cooling system.



Coolant path and Components of an Automobile Engine Cooling System

Most modern cars use aluminum radiators. These radiators are made by brazing thin aluminum fins to flattened aluminum tubes. The coolant flows from the inlet to the outlet through many tubes mounted in a parallel arrangement. The fins conduct the heat from the tubes and transfer it to the air flowing through the radiator.

LITERATURE SURVEY

The literature review in this thesis is taken from paper done by Junjanna G.C[1] in which the study uses the computational analysis tool ANSYS Fluent 13.0 to perform a numerical study on a compact heat exchanger. The computational domain is identified from literature and validation of present numerical approach is established first. Later the numerical analysis is extended by modifying chosen geometrical and flow parameters like louver pitch, air flow rate, water flow rate, fin and louver thickness, by varying one parameter at a time and the results are compared. Recommendations has been made on the optimal values and settings based on the variables tested, for the chosen compact heat exchanger. In another paper by JP Yadav and Bharat Raj Singh[2] in which a complete set of numerical parametric studies on automotive radiator has been presented in detail in this study

INTRODUCTION TO CAD

Throughout the history of our industrial society, many inventions have been patented and whole new technologies have evolved. Perhaps the single development that has impacted Manufacturing more quickly and significantly than any previous technology is the digital computer.

CAD/CAM Software

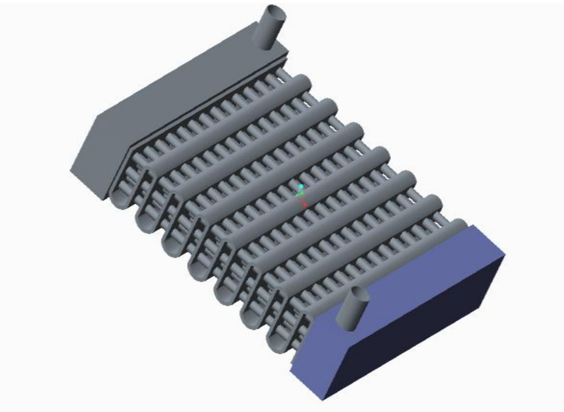
Software allows the human user to turn a hardware configuration into a powerful design and Manufacturing system. CAD/CAM software falls into two broad categories,2-D and 3-D, Based on the number of dimensions are called 2-D representations of 3-D objects is inherently confusing. Equally problem has been the inability of manufacturing personnel to properly read and interpret complicated 2-D representations of objects. 3-D software permits the parts to be viewed with the 3-D planes-height, width, and depth-visible. The trend in CAD/CAM is toward 3-D representation of graphic images. Such representation approximate the actual shape and appearance of the object to be produced; therefore, they are easier to read and understand

INTRODUCTION TO PRO/ENGINEER

Pro/ENGINEER, PTC's parametric, integrated **3D CAD/CAM/CAE solution**, is used by discrete manufacturers for mechanical engineering, design and manufacturing.

Created by Dr. Samuel P. Geisberg in the mid-1980s, Pro/ENGINEER was the industry's first successful parametric, 3D CAD modeling system. The parametric modeling approach uses parameters, dimensions, features, and relationships to capture intended product behavior and create a recipe which enables design automation and the optimization of design and product development processes.

MODEL OF RADIATOR



INTRODUCTION TO FEA

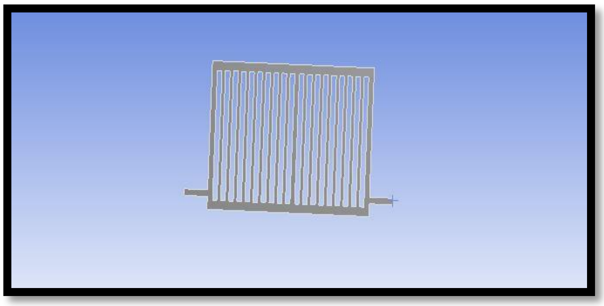
Finite Element Analysis (FEA) was first developed in 1943 by R. Courant, who utilized the Ritz method of numerical analysis and minimization of variational calculus to obtain approximate solutions to vibration systems. Shortly thereafter, a paper published in 1956 by M. J. Turner, R. W. Clough, H. C. Martin, and L. J. Top established a broader definition of numerical analysis. The paper centered on the "stiffness and deflection of complex structures".

INTRODUCTION TO ANSYS

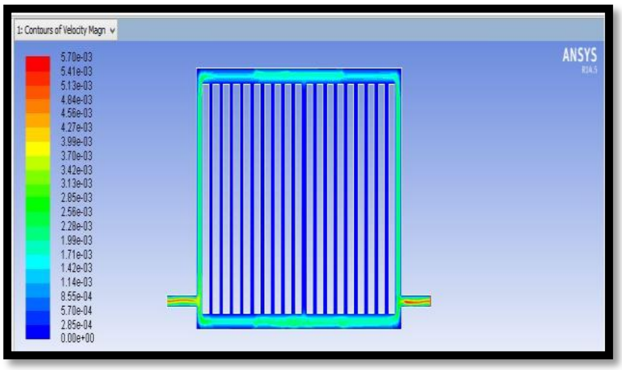
ANSYS is general-purpose finite element analysis (FEA) software package. Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces (of user-designated size) called elements. The software implements equations that govern the behaviour of these elements and solves them all; creating a comprehensive explanation of how the system acts as a whole. These results then can be presented in tabulated, or graphical forms. This type of analysis is typically used for the design and optimization of a system far too complex to analyze by hand. Systems that may fit into this category are too complex due to their geometry, scale, or governing equations.

INTRODUCTION TO CFD

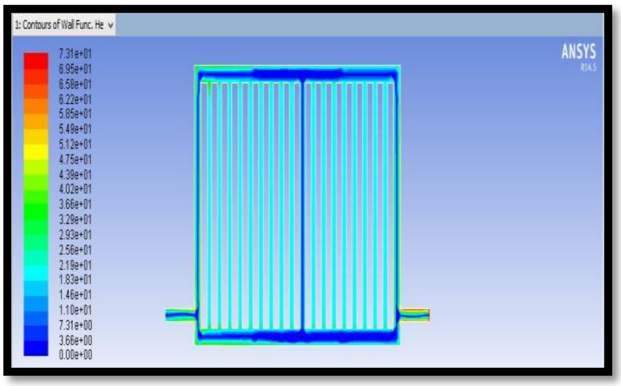
Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces 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.



VELOCITY MAGNITUDE

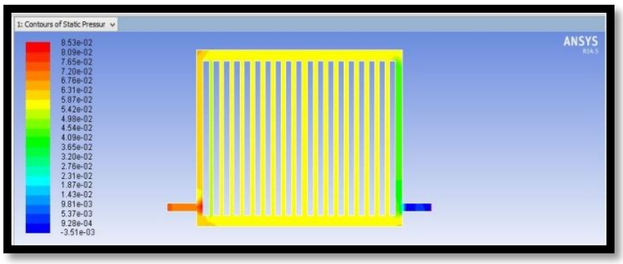


HEAT TRANSFER CO-EFFICIENT

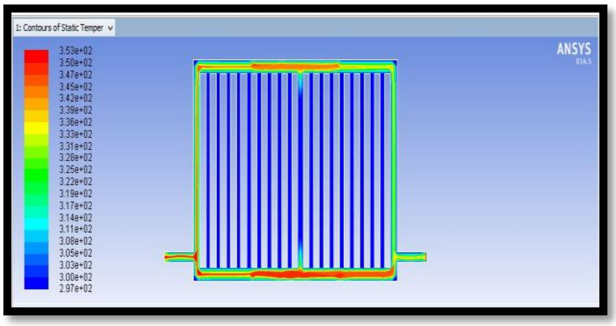


ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

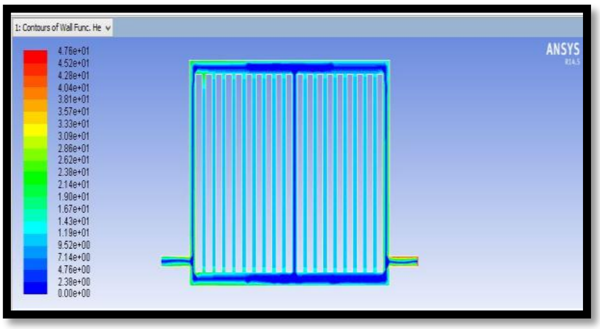
SILICON OXIDE NANO FLUID STATIC PRESSURE



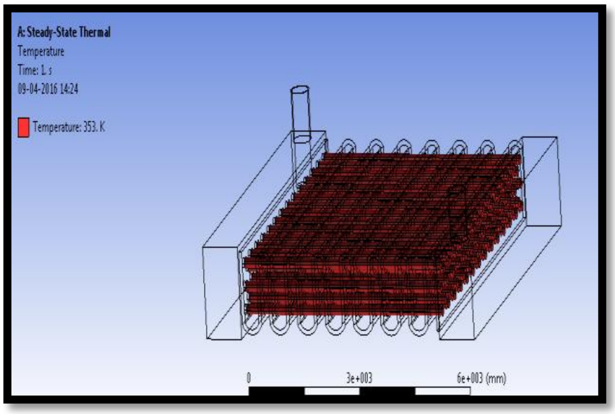
STATIC TEMPERATURE



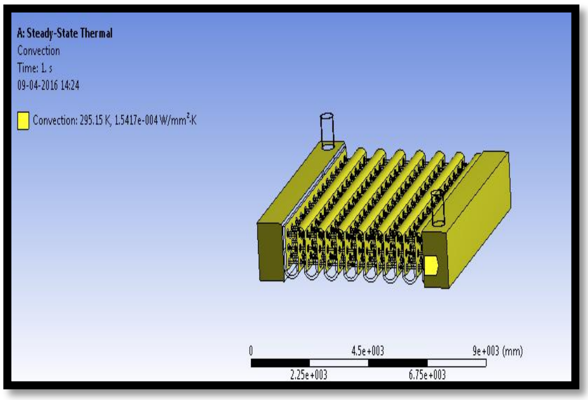
HEAT TRANSFER CO-EFFICIENT



APPLIED TEMPERATURE



APPLIED CONVECTION

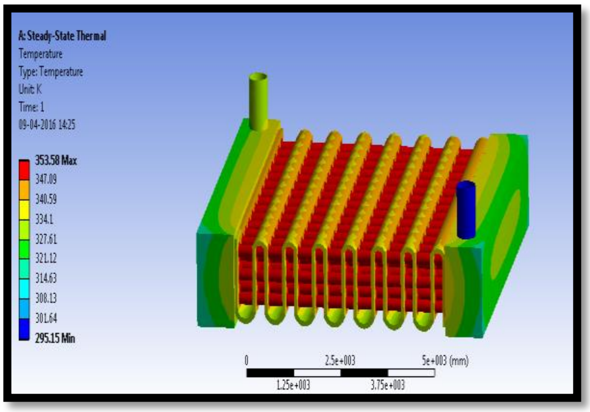


Select steady state thermal >right click>insert>select heat flux

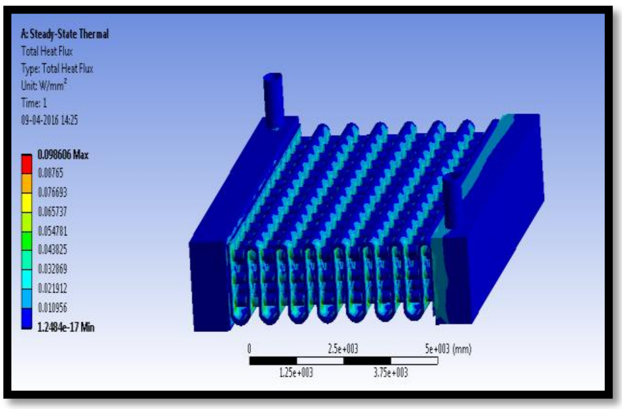
Select steady state thermal >right click>solve

Solution>right click on solution>insert>select temperature

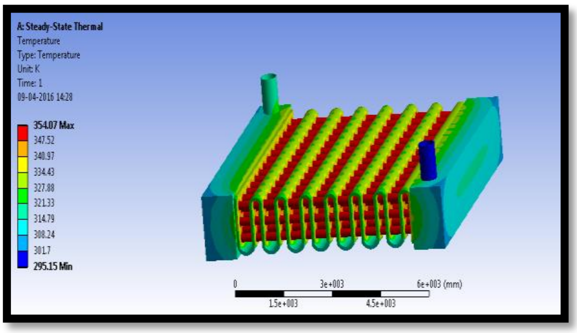
TEMPERATURE



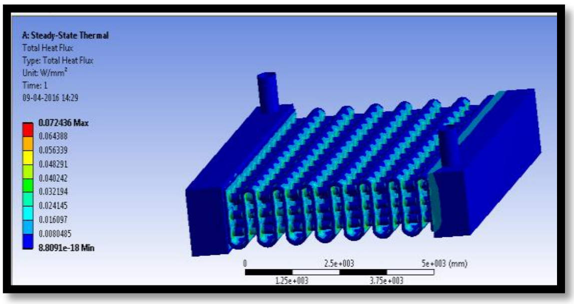
HEAT FLUX

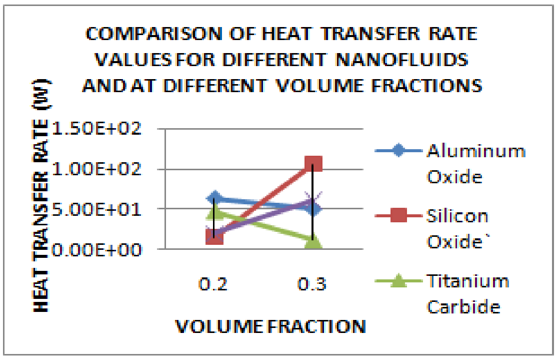


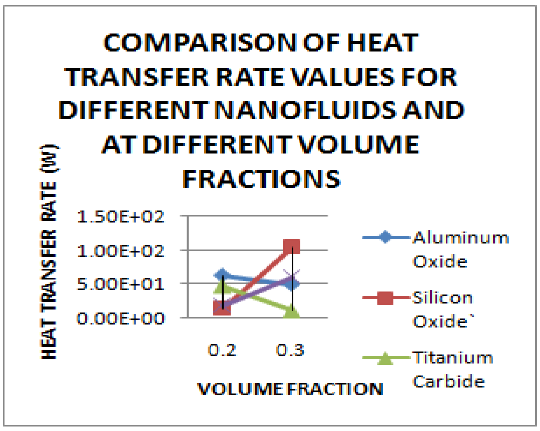
MATERIAL - ALUMINUM ALLOY TEMPERATURE

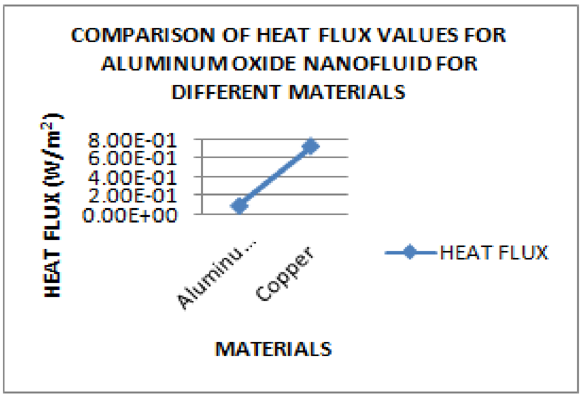


HEAT FLUX









CONCLUSION

3D model of the radiator is done in Pro/Engineer.

CFD analysis is done on the radiator for all nano fluids Aluminum Oxide, Silicon Oxide, Titanium Carbide and Titanium Nitride and at different volume fractions 0.2, 0.3.

By observing the CFD analysis results,The heat transfer coefficient and heat transfer rate are more for Aluminum oxide at volume fraction of 0.3. the pressure, velocity are more for Silicon Oxide at volume fraction of 0.2 and mass flow rate is more for Silicon Oxide at volume fraction of 0.3.

Thermal analysis is done for two materials Aluminum and Copper taking heat transfer coefficient value of Aluminum oxide at 0.3 volume fractions from CFD analysis. By observing thermal analysis results, heat flux is more when Copper is used than Aluminum alloy.

REFERENCES

[1] Performance Improvement of a Louver-Finned Automobile Radiator Using Conjugate Thermal CFD Analysis by Junjanna G.C

[2] Study on Performance Evaluation of Automotive Radiator by JP Yadav and Bharat Raj Singh

[3] Performance Investigation of an Automotive Car Radiator Operated With Nanofluid as a Coolant by DurgeshkumarChavan and Ashok T. PiseSahin

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

[4] Heat Transfer Enhancement of Automobile Radiator with TiO2/Water Nanofluid by PareshMachhar, FalgunAdroja

[5] Yan, W.M., Sheen, P.J., Heat transfer and friction characteristics of fin and tube heat exchangers, International Journal of Heat and Mass Transfer, 43 (2000), pp.1651-1659.

[6] Wolf, I., Frankovic, B., Vilicic, I., A numerical and experimental analysis of neat transfer in a wavy fin and tube heat exchanger, Energy and the Environment (2006) pp.91-101.

[7] Tang, L.H., Zeng, M., Wang, Q.W., Experimental and numerical investigation on air side performance of fin and tube heat exchangers with various fin patterns, Experimental Thermal and Fluid science, 33(2009), pp.818-827.

[8] Wang, C.C., Lo, J, Lin, Y.T. Wei, C.S., Flow visualization of annular and delta winlet vortex generators in fin and tube heat exchanger application, International Journal of Heat and Mass Transfer, 45, (2002), pp.3803-3815.

[9] Fiebig, M., Valencia, A., Mitra, N.K.., Local heat transfer and flow losses in fin and tube heat exchangers with vortex generators: A comparison of round and flat tubes, Experimental Thermal and Fluid Science, 8(1994), pp.35-45.

[10] Leu, J.S., Wu, Y.H., Jang, J.Y., Heat transfer and fluid flow analysis in plate-fin and tube heat exchangers with a pair of block shape vortex generators, International Journal of Heat and Mass Transfer, 47 (2004), pp. 4327-4338.