International Journal of Analytical, Experimental and Finite Element Analysis
Volume 8, Issue 1, March 2021, pp 27 - 34
https://doi.org/10.26706/ijaefea.1.8.20210306

Analysis of heat exchanger (radiator) using Al_2O_3 and SiO_2 nanofluid

Deshmukh. P. Suraj U.[1]
suraj30698@gmail.com

Abhijit. K. Gaikwad[1]
abhijitgaikwad360@gmail.com

Ankush. S. Kokane[1]
ankush.kokane07@gmail.com

Ashwin. A. Kamble[2]
kambleashwin31@gmail.com

[1]Student, Department of Mechanical Engineering, JSPM Narhe Technical Campus, Pune 411041, Maharashtra, India

[2]Assistant Professor, Department of Mechanical Engineering, JSPM Narhe Technical Campus, Pune 411041, Maharashtra, India

Abstract **- Conventional heat transfer fluids like water and engine oil are widely employed in the automobile radiator in recent times. However, to boost the thermal performance of the system, lots more is required from the heat transfer fluid perspective. One of the important techniques to reinforce heat transfer is that to boost the thermal conductivity of working fluid with inclusion of nano-sized solid particles as additives. this paper includes the studies to judge the performance of the heat transfer characteristics of water based nanofluid as a coolant for car radiator. The metal oxide nanoparticles Al2O3 and SiO2 are dispersed into base fluid at 10% volume concentrations. The thermo-physical properties of both nanofluids are calculated and assessed with the assistance literature work. The mass stream rate of nanofluid within the radiator tubes is 4 lit/min. With the assistance of ANSYS FLUENT 19.2 solver, the outlet temperature, convective heat transfer coefficient, Heat Transfer effectiveness and Nusselt numbers for the concentration of every nanofluids is investigated by the mass rate of flow.**

Keywords- Heat transfer, Radiator, Heat Transfer Rate, Effectiveness, Nanofluid CFD, ANSYS.

I. Introduction

Convectional heat transfer fluids such as water, minerals oil and ethylene glycol play an important role in many industrial sectors including power generation, chemical production, air-conditioning, transportation and microelectronics. Although various techniques have been applied to enhance their heat transfer capabilities, their performance is often limited by their low thermal conductivities which obstruct the performance enhancement and compactness of heat exchangers. With the rising demand of modern technology for process intensification and device miniaturization, there was a need to develop new types of fluids that are more effective in terms of heat exchange performance. To achieve this, it has been recently proposed to disperse small amounts of nanometer-sized (10–50 nm) solid particles (nanoparticles) in base fluids, resulting in what is commonly known as nanofluids. The term nanofluid was coined by Choi who was working with the group at the Argonne National Laboratory (ANL), USA, in 1995. The nanoparticles used are ultrafine; therefore, nanofluids appear to behave more like a single-phase fluid than a solid–liquid mixture. The commonly used materials for nanoparticles are metals (Al, Cu, Ag, Au, Fe), nonmetals (graphite, carbon nanotubes), oxides ceramics (Al2O3, CuO, TiO2, SiO2), carbides (SiC), nitrides (AiN, SiN), layered (Al+ Al2O3, Cu+C), PCM and functionalized nanoparticles. The base fluid is usually a conductive fluid, such as water (or other coolants), oil (and other lubricants), polymer solutions, bio-fluids and other common fluids, such as paraffin. Investigations have shown that nanofluids possess enhanced thermophysical properties such as thermal conductivity, thermal diffusivity, viscosity and convective

Research Paper – Peer Reviewed
Published online – 30 March 2021

Cite this article – Deshmukh. P. Suraj U., A. K. Gaikwad, A. S. Kokane, A. A. Kamble, "Analysis of Heat Exchanger (Radiator) Using Al_2O_3 and SiO_2 Nanofluids", International Journal of Analytical, Experimental and Finite Element Analysis, RAME Publishers, vol. 8, issue 1, pp. 27-34, 2021.
https://doi.org/10.26706/ijaefea.1.8.20210306

heat transfer coefficients compared to those of base fluids like oil or water. Owing to their enhanced thermophysical properties, nanofluids have numerous industrial, engineering and biomedical applications such as heat transfer applications: industrial cooling, smart fluids; nanofluid coolant: vehicle cooling, electronics cooling; medical applications: magnetic drug targeting and nano-cryosurgery.

Nanofluid is one in every of the novel inventions of science. Thermal performance of warmth exchanger systems might be increase by using nanofluids that may save energy likewise as our surroundings. Overall performance of a device depends on the properties of the working fluid. Fundamental properties must be investigated before studying the warmth transfer performance of nanofluids, because the performance measurements like heat transfer, energy, exergy etc. rely upon these fundamental properties. supported the available studies on the basic properties of nanofluids, it's found that, thermal conductivity, viscosity and density augment accordingly with the enhancement of volume concentration. Furthermore, thermal conductivity increases with the rise of temperature. However, viscosity and density decrease with the rise of temperatures. additionally, generally heat capacity of the nanofluids decreases with the rise of particle concentrations nevertheless, there also are some inconsistence results. The effect of temperature on the precise heat of nanofluids remains unclear. Since the properties of the nanofluids are better than conventional fluids therefore, there are strong opportunity to extend the general performance of the warmth exchangers by using nanofluids as working fluids and energy losses can be reduced everywhere the planet.

II. Methodology

In this paper we followed the methodology given in the flow chart below first collect all the related information about the heat transfer and the fins and then collecting some of the literature review.

After the model is created the analysis are done by using ANSYS – CFX 19.2. Then the results of each analysis are compared and then the best fins are selected. The result from the Ansys is compared with the theoretical calculation.

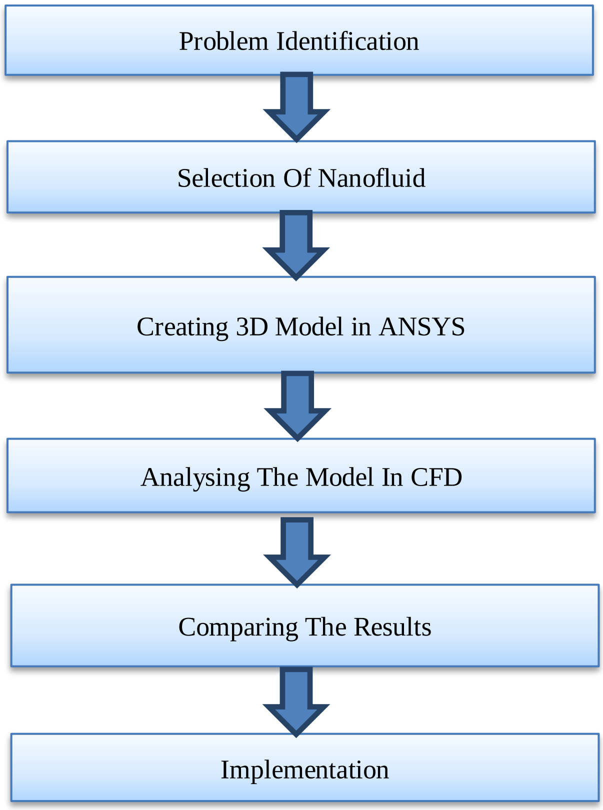


Table 1
Thermo-Physical Properties Of Nanofluids

Properties	SiO_2	Al_2O_3	Base Fluid
ρ	2220	3970	998
C_P	745	765	4182
K	1.38	36	0.6

III. Working Principle

Radiator (usually a cross flow heat exchanger) is a very important component of automobile. It cools down the liquid which is carrying heat from block and protecting it from damage. for prime heat transfer rate, if the dimensions of radiator are increased then it'll increase both the quantity and weight, which is undesirable. Researchers have an interest to extend the effectiveness and compactness of radiators by using coolants with additives like nanoparticles to the bottom fluids. We used Al_2O_3 and SiO_2 nanofluids in automotive cooling system to test the consequences of volumetric flow, inlet temperature and volumetric concentration on Nusselt number. Nusselt number increases because the volume rate of flow, inlet temperature and volume concentration are increased.

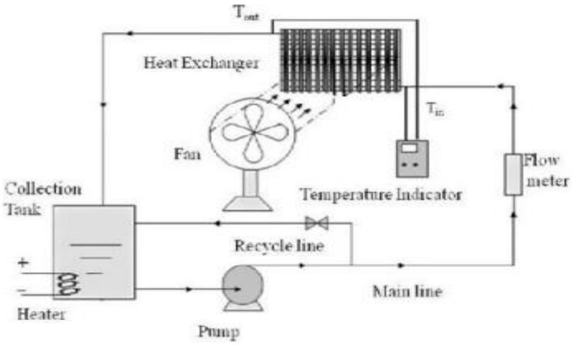


Figure 1: Line Diagram

IV. CFD Simulation Work

CAD modeling was done in Solidworks and fluid model was created in Ansys Design Modeler is used for creating cad models & Ansys CFX 19.2 is used to calculate pressure distribution and velocity profile and torque to calculate power output by performing CFD analysis. ANSYS CFX enables you to perform fluid-flow analysis of incompressible and compressible fluid flow and heat transfer in complex geometries. You import meshes, specify the materials, boundary conditions, and solution parameters, solve the calculations, view the results, then create reports using built-in tools. ANSYS CFX is a general-purpose Computational Fluid Dynamics (CFD) software suite that combines an advanced solver with powerful preprocessing and post-processing capabilities. It includes the following features:

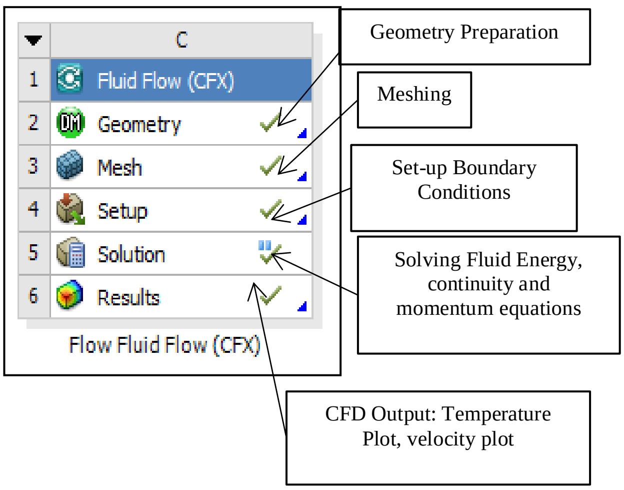


Figure 2: Features of ANSYS CFX for CFD Analysis

An advanced coupled solver that's both reliable and robust. Full Solution of problem definition, analysis, and results presentation.

An intuitive and interactive setup process, using menu and advanced graphics in ANSYS.

CFD Analysis involves following major steps;

a) Pre-Processing
- Geometry Modelling
- Meshing
- Material
- Loading and precondition

b) Solution
- Solving Fluid Energy, continuity and momentum equations

c) Post-Processing
- Temperature Gradient
- Velocity Profile and Path lines

Table II

Heat Transfer Rate

	Inlet Temp. (Tin) [°C]	Outlet Temp. (Tout) [°C]	Specific heat (Cp)	Heat Transfer Coefficient (h)
Air	28	32.82	1	37 [W/m²]
Al2o3	70	51.3	765	2.034 [mW/m²]
Sio2	70	51.1	745	1.361 [mW/m²]

V. The Mathematics of CFD

The set of equations that describe the processes of momentum, heat and mass transfer are referred to as the Navier-Stokes's equations. These partial differential equations were derived within the early nineteenth century and don't have any known general analytical solution but is discretized and solved numerically. Equations describing other processes, like combustion, can even be solved in conjunction with the Navier-Stokes equations. Often, an approximating model is employed to derive these additional equations, turbulence models being a very important example. There are variety of various solution methods that are utilized in CFD codes. the foremost common, and therefore the one on which CFX is predicated, is thought because the finite volume technique. In this technique, the region of interest is split into small sub-regions, called control volumes. The equations are discretized and solved iteratively for every control volume. As a result, an approximation of the worth of every variable at specific points throughout the domain is obtained. during this way, one derives a full picture of the behavior of the flow.

A. Geometry Detail

Solid works is use to create CAD geometry and Ansys Design Modeler is for de-featuring of geometry. Geometry prepared in solid works is as below.

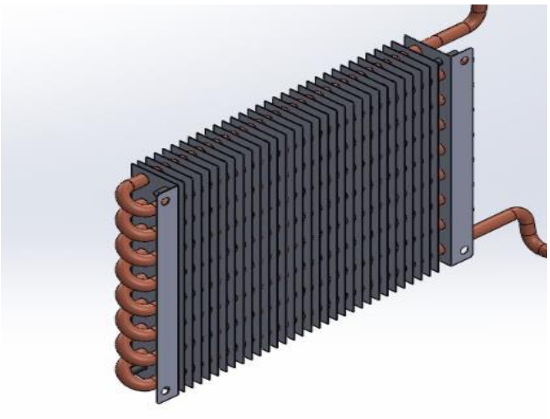


Figure 3: Radiator

Figure 4: Solid Model of Radiator

After doing de-featuring in ANSYS DM, below is the final fluid domain considered for CFD analysis as below

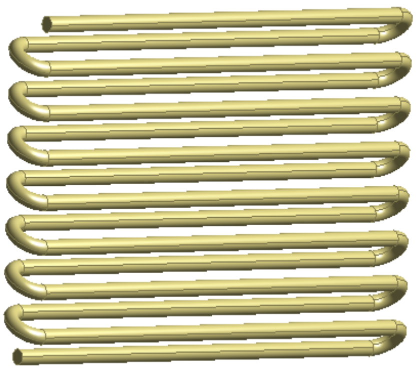


Figure 5: Fluid geometry

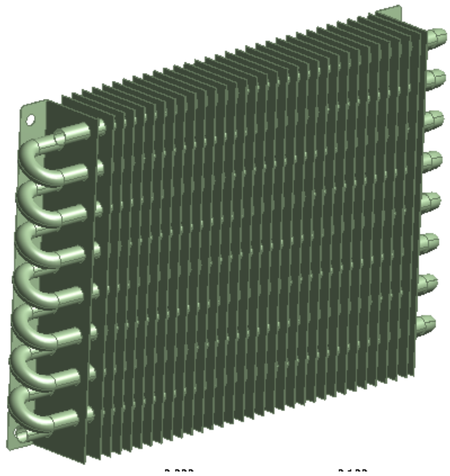


Figure 6: Solid Geometry of Radiator

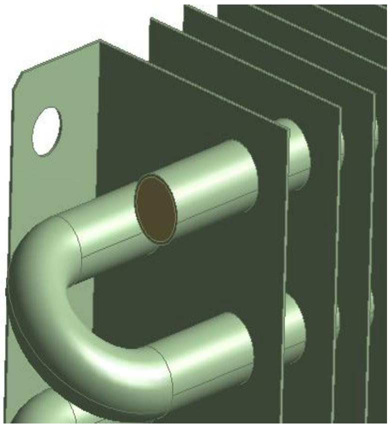


Figure 7: Fluid Solid Combined geometry considered for CFD analysis.

B. Meshing

Final model from design modeler then imported in ANSYS Workbench. At all wall surfaces uses inflations layers for having proper flow and vary fine meshed used to capture the accuracy of solution. Body size of 1mm used for fluid domain and 2mm element size.

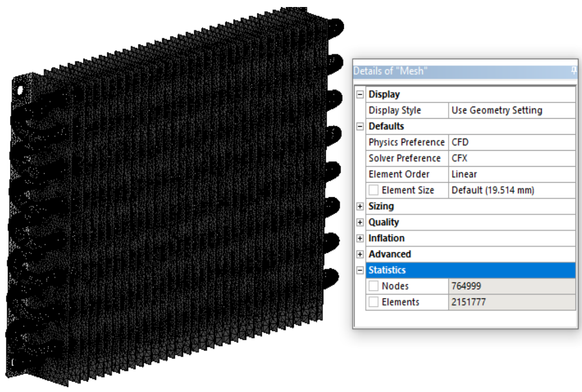


Figure 8: Meshing of Solid Model of Radiator.

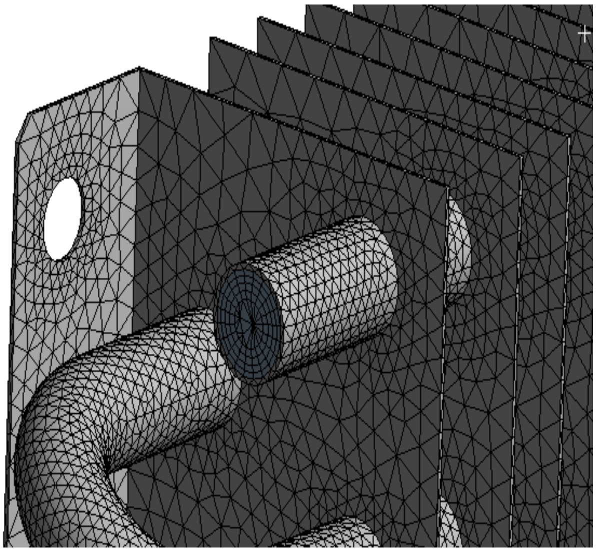


Figure 9: Meshing

VI. Design calculations

To find the effectiveness of radiator and nanofluid, following equations are used the results are plotted in table 2.

Heat capacity rate = Ch=m×Cp

Heat capacity rate = Ch=m×Cp

Heat capacity rate ratio = C_r = Cmin/Cmax

Heat Transfer Coefficient Calculations

Hydraulic Diameter (Dh) = (4×Ao)/P

Mass Flow Rate per Unit Area = G=m/Ao

Reynolds no. = Re=(G×Dh)/μ

Prandtl no. Pr = μCp/k

Nusselt no. Nu = $0.0265 \times Re^{0.8} \times Pr^{0.3}$

Heat transfer coefficient= h=(Nu×k)/Dh

Radiator Effectiveness Calculation

$$\epsilon = \frac{1-e^{NTU(1+c)}}{1+c}$$

Effectiveness of Nanofluids

$$\epsilon = \frac{1-e^{-NTU(1+c)}}{1+c}$$

TABLE II

CALCULATIONS FOR BOTH NANOFLUIDS

Sr. No.		For Al_2O_3	For SiO_2
1	Heat capacity rate	183.6 J/hr	178.8 J/hr
2	Heat capacity rate ratio	1.026	
3	Hydraulic Diameter	9mm	
4	Mass Flow Rate per Unit Area	2782.14 Kg.m/hr	
5	Reynolds no.	2381.42	
6	Prandtl no.	12.486	12.657
7	Nusselt no.	28.42	28.52
8	Heat transfer coefficient	2.034	1.961
9	Radiator Effectiveness	135.40 KW/m^2°C	131.97 KW/m^2°C
10	Effectiveness of Nanofluids	0.4950	0.4961

VII. CFD Analysis

A. CFD Modelling

1. The partial differential equations are integrated over all the control volumes within the region of interest. this can be reminiscent of applying a basic conservation law (for example, for mass or momentum) to every control volume.
2. These integral equations are converted to a system of algebraic equations by generating a group of approximations for the terms within the integral equations.
3. The algebraic equations are solved iteratively. An iterative approach is required due to the nonlinear nature of the equations, and because the solution approaches the precise solution, it's said to converge. for every iteration, an error, or residual, is reported as a measure of the general conservation of the flow properties. How close the ultimate solution is to the precise solution depends on variety of things, including the scale and shape of the control volumes and therefore the size of the ultimate residuals. Complex physical processes, like combustion and turbulence, are often modelled using empirical relationships. The approximations inherent in these models also contribute to differences between the CFD solution and therefore the real flow. The solution process requires no user interaction and is, therefore, usually disbursed as a batch process. The solver

produces a results file that's then passed to the post-processor.

Computational Fluid Dynamics (CFD) is the discipline of obtaining a numerical description of the whole flow field of interest by calculating the numerical solution of the governing equation for fluid flow while advancing the solution through space or time. In viscid or viscous flows, including non-ideal and reactive fluid behaviour, the equation can describe steady or unstable, compressible or incompressible, and in viscid or viscous flows. The form that is chosen is determined by the intended application. The complexity of the geometry, the flow physics, and the computing time required to obtain a solution define the state of the art.

B. *Methodology*

There are three key steps in CFD calculations:

Pre-Processing, Solver Execution, and Post-Processing are the three stages of the process

The modelling goals are determined and a computational grid is created during the pre-processing stage. The solver is started in the second phase by setting numerical models and boundary conditions. The solver continues to run until convergence is achieved. The findings are analysed once the solver has finished, which is the post-processing phase. Procedure.

Step 1: In ANSYS Workbench or Space Claim 3D Modellers, draw the cylindrical fin, rectangular fin, hexagonal fin, and square fin.

Step 2: Run more simulations with the produced fins in ANSYS software.

Step 3: Analyse the data and calculate efficiency and other parameters using it.

Inlet: mass flow rate of 0.048Kg/s & temperature of 70DegC is applied at inlet of heat Exchange (HE) as per figure 10.

Outlet: Outlet defined with outflow at atmospheric pressure as shown in figure 11.

Convection due to Fan: Considering velocity of fan is 1m/s, the HTC is considered or current simulation as 37W/m2K.

Solver Setting: 100 iterations are defined with high resolution solved for steady state CFD analysis to get better accuracy.

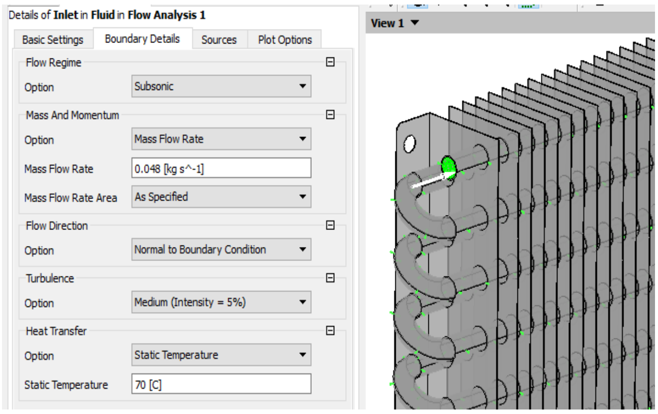


Figure 10: Input temp and flow rate

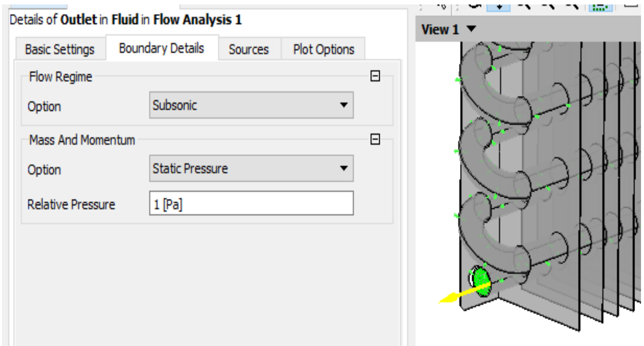


Figure 11: outlet

C. *CFD Results*

Temperature plot: The temperature distribution along the length of the tube clearly shows the energy separation. The energy separation takes place from inlet. The hot inlet temperature is reduced from 70Dec C to 51.3DegC for AL2O3 nano fluid as shown in below plot.

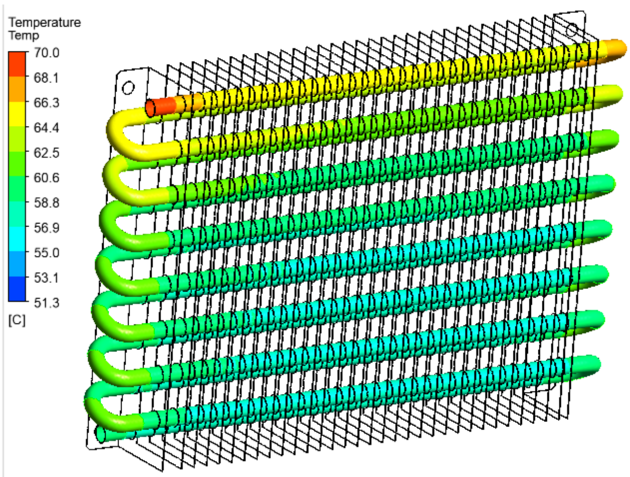


Figure 12: Temperature drop in RADIATOR (Al2O3)

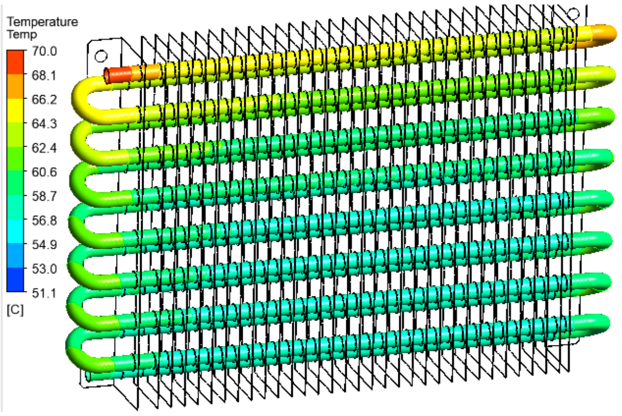
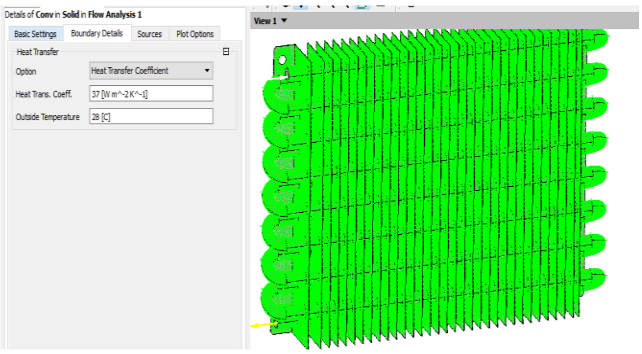


Figure 13: Temperature drop in RADIATOR (SiO2)



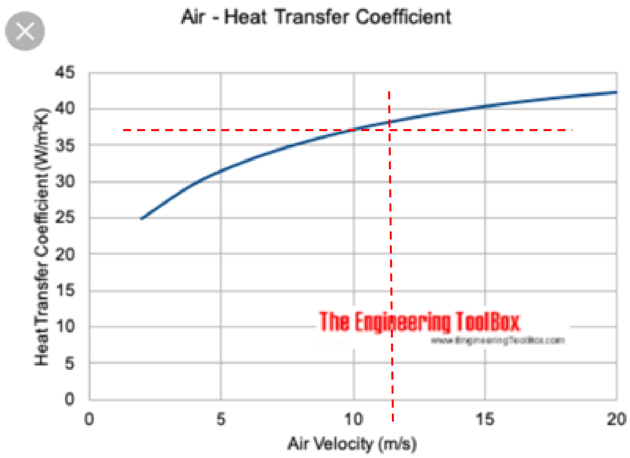


Figure 14: Air velocity and outlet

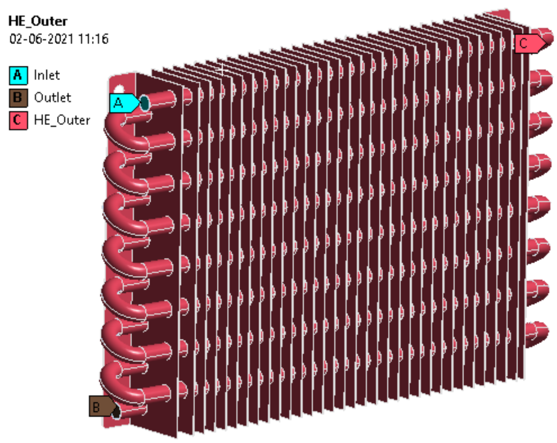


Figure 15: Name selection to define boundary condition

VII. Results, Discussion and Conclusion

• Heat exchanger CFD analysis in ANSY CFX was successfully completed at 4lit/Min mass flow rate and 70Deg nanofluid inlet temperature.

• Nano fluid AL2O3 and SIO2 are considered for current study.

• Temperature of inlet for AL3O2 is reduced from 70Dg c to 51.3Deg c whereas for SIO2 is reduced to 51.1Dec which is slightly lower than AL2O3.

• Hence SIO2 is observed to be better than AL2O2 nano fluid in terms of temperature drop at 70Dg C.

• However, the Cp is not changing much in both nano fluid and hence results are not varied much.

As a result, the experimental and analytical results for hexagonal fins are exactly the same.

Due to the highest area of contact, the hexagonal fin is determined to be the best for maximal heat transfer rate and efficiency based on the aforementioned statistics.

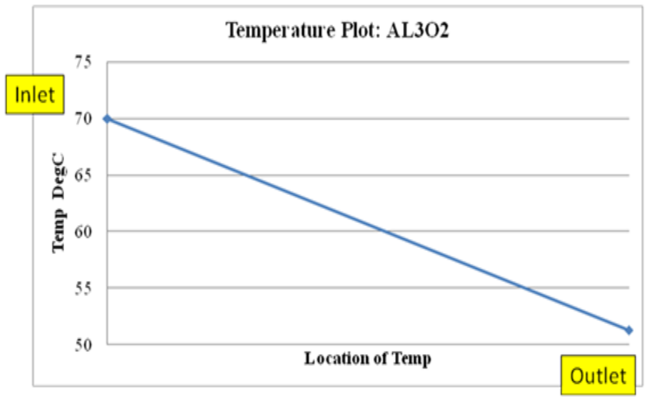


Figure 16: Temperature Plot for AL2O3

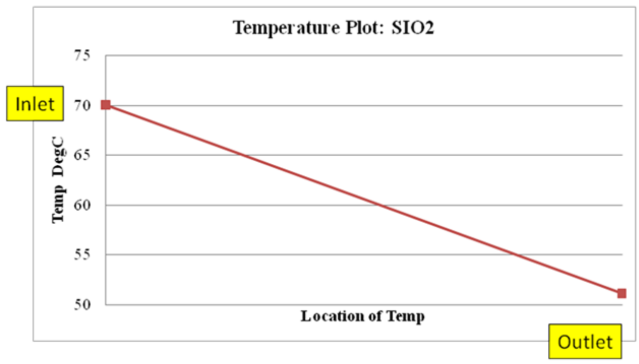


Figure 17: Temperature Plot for SiO2

References

[1] Matthew Carl, Dana Guy, Brett Leyendecker, Austin Miller, and Xuejun Fan, The Theoretical and Experimental

Investigation of the Heat Transfer Process of an Automobile Radiator, *ASEE Gulf Southwest Annual Conference*,Texas, 2012

[2] R. Esmaeili Sany, M. H. Saidi, J. Neyestani,Experimental Prediction of Nusselt Number and Coolant Heat Transfer Coefficient in Compact Heat Exchanger Performed with ε-NTU Method, *The Journal of Engine Research*, Vol.18, Spring, 2010.

[3] K.Y. Leong, R. Saidur, S.N. Kazi, A.H. Mamun, Performance investigation of an automotive car radiator operated with nanofluid-based coolants (nanofluid as a coolant in a radiator), *Applied Thermal Engineering*,30, 2010.

[4] S. K. Saripella, W. Yu, J. L. Routbort, D. M. France, Rizwan-uddin, Effects of Nanofluid Coolant in a Class 8 Truck Engine, *SAE Technical Paper*, 2141, 2007.

[5] D. Ganga Charyulu, Gajendra Singh, J.K. Sharma, Performance evaluation of a radiator in a diesel engine- a case study, Applied Thermal Engineering 19, 1999.

[6] P. S. Amrutkar, S. R. Patil, Automotive Radiator Sizing and Rating – Simulation Approach, *IOSR Journal of Mechanical and Civil Engineering* (IOSR-JMCE, ISSN(e): 2278-1684, ISSN(p): 2320–334X, 2013.

[7] Pawan S. Amrutkar, Sangram R. Patil, Automotive Radiator Performance – Review, *International Journal of Engineering and Advanced Technology* (IJEAT), ISSN: 2249 – 8958, Volume-2, Issue-3, February 2013