International Journal of Research
Available at https://edupediapublications.org/journals

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 04 Issue14
November 2017

Thermal Performance of Nano Fluids In Ducts With Double Forward-Facing Steps

Dr. P . Srinivasulu , Mr . P. Raju , K.Prashanth
, professor, [2]Assistant professor, [1]M.Tech (Thermal Engineering)Student
Dept. Of Mechanical Engineering, "Vaagdevi College Of Engineering" Bollikunta, Warangal, Telangana 506005.

Abstract:

Human comfort in cars is of prime importance nowadays, in which thermal comfort plays an important role. With the rapid development of technology and increasing demands by customers, the climate control of the passenger cabin has to be taken into account in any vehicle development process. Duct is used to carry the air in air conditioning. This air conditioning is divided in to 1. Summer air conditioning, 2. Winter air conditioning, 3. Year round air conditioning. Usage of A.C type is dependent on the atmosphere condition. For this air conditioning duct design is very important. This design depends on the amount of air carrying through ducts, shape of the duct In this project the AC is to be designed for Summer Air Conditioning type. Because in our city conditions throughout the year, the temperature doesn't fall below 15oC.

In this project, we are introduced nano fluid magnesium oxide at different volume fraction(0.1,0.2,0.3 and 0.4%) In this project, nano fluid load calculations, duct design, CFD and Thermal analysis for main duct are to be done. For thermal analysis, materials of duct are Galvanized Iron and carbon fiber and Glass Fiber.

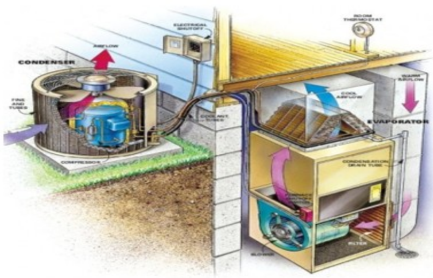
Duct design is done in 3D modeling software CREO parametric software and CFD analysis in ANSYS.

Keywords: Air conditioning, Duct, Glass fiber, CFD, FEA.

.I. INTRODUCTION

Air conditioning (often referred to as AC, A.C., or A/C) is the process of removing heat from a confined space, thus cooling the air, and removing humidity. Air conditioning can be used in both domestic and commercial environments. This process is used to achieve a more comfortable interior environment, typically for humans or animals; however, air conditioning is also used to cool/dehumidify rooms filled with heat-producing electronic devices, such as computer servers, power amplifiers, and even to display and store artwork. Air conditioners often use a fan to distribute the conditioned air to an occupied space such as a building or a car to improve thermal comfort and indoor air quality. Electric refrigerant-based AC units range from small units that can cool a small bedroom, which can be carried by a single adult, to massive units installed on the roof of office towers that can cool an entire building. The cooling is typically achieved through a refrigeration cycle, but sometimes evaporation or free cooling is used. Air conditioning systems can also be made based on desiccants (chemicals which remove moisture from the air) and subterraneous pipes that can distribute the heated refrigerant to the ground for cooling. In the most general sense, air conditioning can refer to any form of technology that modifies the condition of air (heating, cooling, (de-)humidification, cleaning, ventilation, or air movement). In common usage, though, "air conditioning" refers to systems which cool air. In construction, a complete system of heating, ventilation, and air conditioning is referred to as heating, ventilation, and air conditioning (HVAC – as opposed to AC) Early commercial applications of air conditioning were manufactured to cool air for industrial processing rather than personal comfort. The increase in use of air conditioning over the years has been implicated as a contributor to increasing obesity, because appetite naturally decreases in uncomfortably high temperatures.

International Journal of Research
Available at https://edupediapublications.org/journals
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 04 Issue14
November 2017



II. LITERATURE REVIEW

Generally in the sugar mills, they are using biogases as the fuel, to burn the biogases primary and secondary air is supplied. Forced draught fan supply primary air and sail air fan supply secondary air. Primary air is sucked by forced draught fan and passed through the airpreheater to furnace. In airpreheater the flue gases coming out of economizer are further utilized for preheating the atmosphere air before supplying to the furnace. Atmosphere wet air enters into the airpreheater, reacts with the metal tubes causes corrosion. The main aim of this project is to avoid the corrosion of cold end in airpreheater and increase temperature of the supply air to furnace. To resist the corrosion, the atmosphere air temperature is increases up to above dew point temperature (65°C). Air flow analysis takes place from FD fan to furnace using computational fluid dynamics.

INTRODUCTION TO CAD

Computers are being used increasingly for both design and detailing of engineering components in the drawing office. Computer-aided design (CAD) is defined as the application of computers and graphics software to aid or enhance the product design from conceptualization to documentation. CAD is most commonly associated with the use of an interactive computer graphics system, referred to as a CAD system.

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. This powerful and rich design approach is used by companies whose product strategy is family-based or platform-driven, where a prescriptive design strategy is critical to the success of the design process by embedding engineering constraints and relationships to quickly optimize the design, or where the resulting geometry may be complex or based upon equations. Pro/ENGINEER provides a complete set of design, analysis and manufacturing capabilities on one, integral, scalable platform.

INTRODUCTION TO FEM

FEM/FEA helps in evaluating complicated structures in a system during the planning stage. The strength and design of the model can be improved with the help of computers and FEA which justifies the cost of the analysis. FEA has prominently increased the design of the structures that were built many years ago.

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. ANSYS provides a cost-effective way to explore the performance of products or processes in a virtual environment. This type of product development is termed virtual prototyping.

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.

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 04 Issue14
November 2017

CALCULATIONS TO DETERMINE PROPERTIES OF NANO FLUID BY CHANGING VOLUME FRACTIONS

At volume fraction-0.4

DENSITY OF NANO FLUID

$$\rho_{nf} = \phi \times \rho_s + [(1-\phi) \times \rho_w]$$

$\rho_{nf} = 2022.92$ kg/m^3

SPECIFIC HEAT OF NANO FLUID

$$C_{p\,nf} = \frac{\phi \times \rho s \times Cps + (1-\phi)(\rho w \times Cpw)}{\phi \times \rho s + (1-\phi) \times \rho w}$$

At $\phi = 0.4$

$C_{p\,nf} = 1910.408$ j/kg-k

VISCOSITY OF NANO FLUID

$$\mu_{nf} = \mu_w (1+2.5\varphi)$$

At $\phi = 0.4$, $\mu_{nf} = 0.002006$ kg/m-s

THERMAL CONDUCTIVITY OF NANO FLUID

$$K_{nf} = \frac{Ks + 2Kw + 2(Ks - Kw)(1+\beta)^3 \times \phi}{Ks + 2Kw - (Ks - Kw)(1+\beta)^3 \times \phi} \times k_w$$

At $\phi = 0.4$, $\beta = 0.1$ taken from journal

$K_{nf} = 0.1.84577$ (W/m-k)

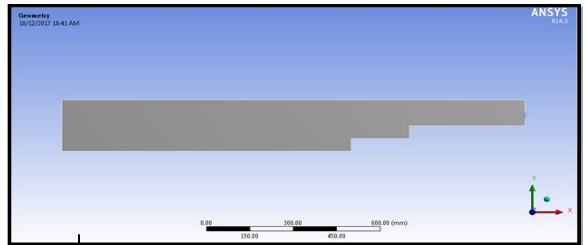
CFD ANALYSIS OF AC DUCT

FLUID-Mgo nano fliud

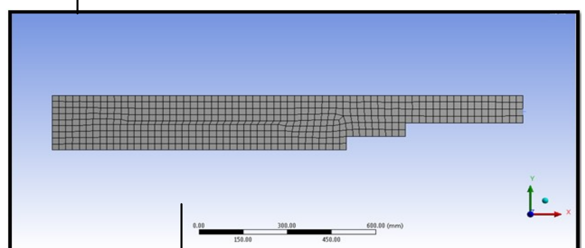
Condition-laminar flow

At volume fraction-0.4%

Geometry Model

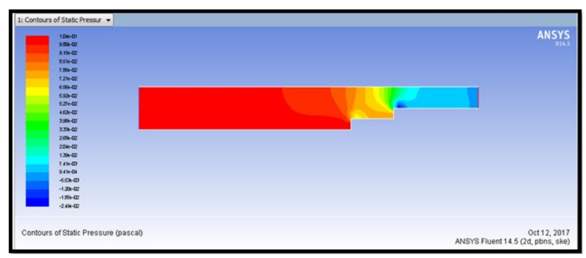


Meshed Model

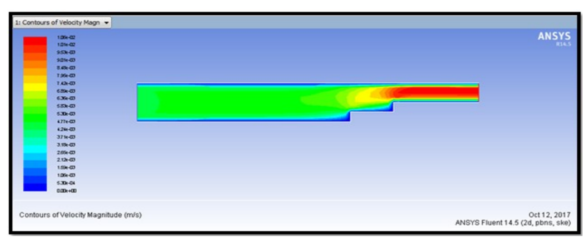


Boundary Conditions

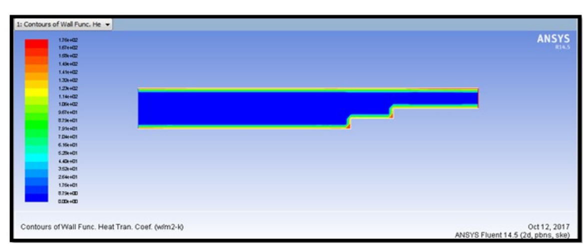
PRESSURE



VELOCITY



HEAT TRANSFER COEFFICIENT



MASS FLOW RATE

International Journal of Research
Available at https://edupediapublications.org/journals
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 04 Issue14
November 2017

Mass Flow Rate	(kg/s)
inlet	1.9015447
interior-_trm_srf	-41.505836
outlet	-1.9016074
wall-_trm_srf	0
Net	-6.2704086e-05

HEAT TRANSFER RATE

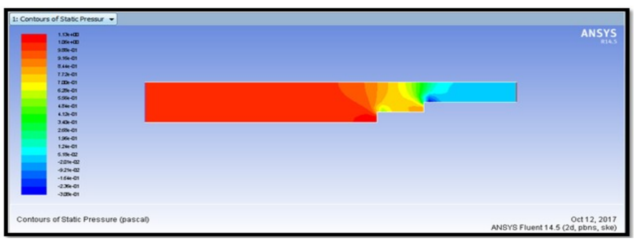
Total Heat Transfer Rate	(w)
inlet	90281
outlet	-81934.844
wall-_trm_srf	-8349.2725
Net	-3.1162109

CONDITION-TURBULENT FLOW

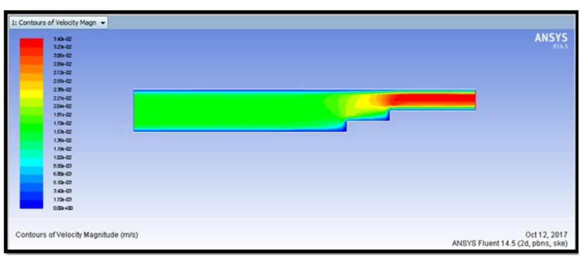
FLUID-Mgo nano fliud

AT VOLUME FRACTION-0.4%

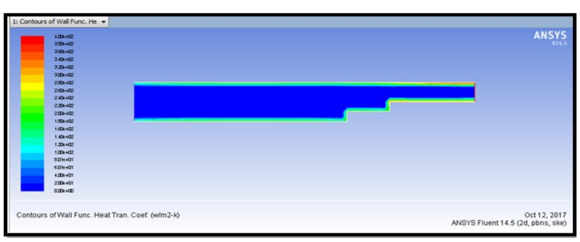
PRESSURE



VELOCITY



HEAT TRANSFER COEFFICIENT



MASS FLOW RATE

Mass Flow Rate	(kg/s)
inlet	6.190135
interior-_trm_srf	-134.89757
outlet	-6.190114
wall-_trm_srf	0
Net	2.0980835e-05

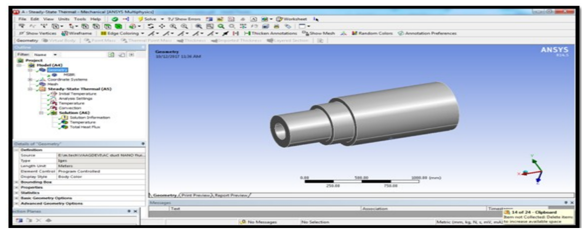
HEAT TRANSFER RATE

Total Heat Transfer Rate	(w)
inlet	293872.5
outlet	-278998.53
wall-_trm_srf	-14872.914
Net	1.0546875

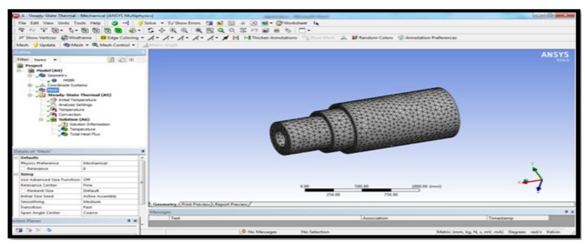
THERMAL ANALYSISOF AC DUCT

MATERIAL- CARBON FIBER

IMPORTED MODEL

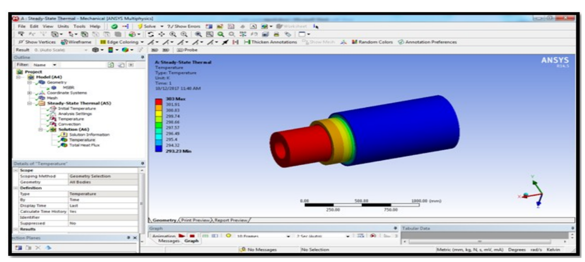


MESHED MODEL



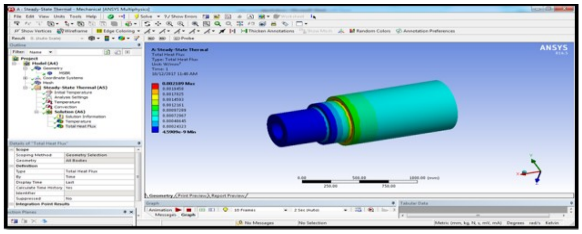
BOUNDARY CONDITIONS

TEMPERATURE



International Journal of Research
Available at https://edupediapublications.org/journals
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 04 Issue14
November 2017

HEAT FLUX



RESULT TABLES

CFD results

Condition -Laminar

Volume fraction(%)	Pressure(Pa)	Velocity (m/s)	Heat transfer coefficient(W/m²k)	Mass flow rate (Kg/sec)	Heat transfer rate(w)
0.1	7.26e-02	1.13e-02	9.59e+01	2.53e+05	2.577
0.2	7.22e-02	1.02e-02	1.1e+02	3.099e+05	1.504
0.3	8.77e-02	1.04e-02	1.38e+02	8.334e-06	0.18505
0.4	1.04e-01	1.06e-02	1.76e+02	6.27e-05	3.1162

Condition -Turbulent

Volume fraction(%)	Pressure(Pa)	Velocity (m/s)	Heat transfer coefficient(W/m²k)	Mass flow rate (Kg/sec)	Heat transfer rate(w)
0.1	6.71e-01	3.3e-02	2.19e-+02	1.955e-05	1.217
0.2	8.08e-01	3.2e-02	2.64e+02	3.33786e-05	2.95
0.3	9.78e-01	3.38e-08	3.23e+02	2.954e-5	1.6210
0.4	1.13e+00	3.40e-02	4.00e+02	2.098e-05	1.233

THERMAL ANALYSIS RESULTS

Material	Temperature (K)		Heat flux
	Min	Max	
Galvanized iron	292.99	303	0.0000843
Carbon fiber	293.23	303	0.002189
E glass fiber	292.92	303	0.0001136

CONCLUSION

G.I Sheets Can Bear 350 to 600 MPa

Out pressure is also suitable For Seminar Hall from CFD Analysis we got Out let pressure as 4 to 6

Residences: 3 m/s to 5 m/s

Theatres: 4 to 6.5 m/s

Restaurants: 7.5 m/s to 10 m/s

From the above calculations we can take the 6" X 8" Duct for our Seminar Hall.

1. Smaller ducts and hence, lower initial cost and lower space requirement

2. Higher pressure drop and hence larger fan power consumption.

3. In this project we calculated amount of refrigeration required for the seminar hall.

4. 23 Tones of refrigeration required for this seminar hall.

We have done CFD analysis on the duct by varying the nano fluids at different volume fractions By observing the results, by increasing the volume fractions the pressure and velocity in the duct is increasing, outlet velocity and heat transfer coefficient is increasing and temperature is decreasing.

We have also done thermal analysis on the AC Duct. By observing the analysis results, thermal flux is more for carbon fiber at volume fraction 0.4%, the heat transfer rate is more when carbon fiber is taken.

REFERENCES

[1] J. J. Alonso. A Crash-Course on the Adjoint Method for Aerodynamic Shape Optimization (2003).

[2] ANSYS FLUENT Manual. ANSYS FLUENT Adjoint Solver. Version 14.5. ANSYS, Inc., 2012.

[3] N. Bakhvalov. Courant–Friedrichs–Lewy Condition. Encyclopedia of Mathematics (2001).

[4] A. C. Duffy. An introduction to gradient computation by the discrete adjoint method. Tech. rep. Technical report, Florida State University, 2009.

[5] J. H. Ferziger and M. Peri´c. Computational methods for fluid dynamics. Vol. 3. Springer Berlin, 1996. [6] T. Han et al. Adjoint Method for Aerodynamic Shape Improvement. Training 2006 (), 06–28. [7] M. Peric and S. Ferguson. The advantage of polyhedral meshes. Tech. rep. Technical report, CD Adapco Group, 2005. www. cd-adapco. com/news/24/TetsvPoly. htm, 2012.

[8] O Pironneau. Optimal shape design for elliptic systems. 1984.

[9] J. Reuther et al. Aerodynamic shape optimization of complex aircraft configurations via an adjoint formulation. Research Institute for Advanced Computer Science, NASA Ames Research Center, 1996. [10] Z. Shang. Performance analysis of large scale parallel CFD computing based on Code Saturne. Computer Physics Communications 184.2 (2013), 381–386.

[11] D. Th´evenin and G. Janiga. and Computational Fluid Dynamics (2008).

[12] User guide. Version 4.3. HARPOON.