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

Design and Analysis of Turbulent Flow of Duct Using In Centralised A.C.

Vidyasagar Budumuru
Department of Mechanical Engineering
Malla Reddy College of Engineering and Technology, Maisammaguda, Dhulapally Post, Secunderabad-500100, India.

Mr. K. Bicha
Department of Mechanical Engineering
Malla Reddy College of Engineering and Technology, Maisammaguda, Dhulapally Post, Secunderabad-500100, India.

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 15°C.

In this project, cooling 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 Glass Fiber.

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

INTRODUCTION

Air conditioning (often referred to as AC, A.C., or A/C) is the process [1]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 [2]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

Cite this article as: Vidyasagar Budumuru & Mr. K. Bicha, "Design and Analysis of Turbulent Flow of Duct Using In Centralised A.C.", International Journal & Magazine of Engineering, Technology, Management and Research, Volume 5 Issue 1, 2018, Page 46-51.

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

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. In 1902 the first modern electrical air conditioning was invented by Willis Carrier in Syracuse, New York. Designed to improve manufacturing process control in a printing plant, his invention controlled not only temperature but also humidity. The low heat and humidity were to help maintain consistent paper dimensions and ink alignment. Later Carrier's technology was applied to increase productivity in the workplace, and The Carrier Air Conditioning Company of America was formed to meet rising demand. Over time air conditioning came to be used to improve comfort in homes and automobiles. Residential sales expanded dramatically in the 1950s.

DUCT DESIGN

The purpose of air conditioning ductwork is to deliver air from [3]the fan to the diffusers which distribute the air to the room. Air Moves through the Ductwork in Response to a Pressure Difference created by the Fan. The necessary pressure difference will be a function of the way the ductwork is laid out and sized. The objective of duct design is to size the duct so as to minimize the pressure drop through the duct, while keeping the size (and cost) of the ductwork to a minimum. Proper duct design requires a knowledge of the factors that effect pressure drop and velocity in the duct.

LITERATURE REVIEW

Numerical Simulation and Analysis of HVAC Duct #1Mr. Shivanand Doddaganiger,[4] #2Dr. Narendra Deore

Simulation of passenger compartment climatic conditions is becoming increasingly important as a complement to wind-tunnel and field testing to achieve improved airflow comfort while reducing vehicle development time and cost. Computational Fluid Dynamics (CFD) analysis of a passenger compartment involves not only geometric complexity but also strong interactions of airflow. Temperature and velocities are major factors responsible for cabin temperature. Primary focus of the study is to assess existing airflow and thermal comfort performance and propose improvement in its duct shape and vent orientation for passenger comfort. Air-flow management inside a vehicle cabin because of airflow distribution over manikin is also part of the study. Investigation of fluid flow through HVAC duct form different outlets of an automotive heating ventilation and air conditioning (HVAC) system will carried out in the present work. The CAD model was developed and analysis will be done. To analyze the air flow, a simulation is performed using Computational Fluid Dynamics, and with the help of this simulation we can roughly estimate the behavior of air. The performance of the HVAC system is judged by parameters like air discharge rate at cabin level, pressure drop through the system, uniformity of the air flow at the outlet faces and distribution between different duct outlets. Pressure loss is another aspect which will require a lot of attention at this stage of development. It is one of the major characteristic which will ensure a smooth flow of air inside the HVAC system. All these parameters are predicted by computational fluid dynamics (CFD) analysis. should meet the standards. Comparison between CFD and testing results will be made in the HVAC system development by incorporating CFD as a design tool.

Analysis of Flow through Solar Dryer Duct Using CFD 1Prof. A.I. Ambesange, 2 Prof. Kusekar S.K[5]

Energy is a crucial input in the process of economic, social and industrial development of any nation. During past several decades, energy demand of the world has been increasing continuously at an alarming rate due to increase in population, industrialization, transportation etc. Continuous use of fossil fuels have resulted energy crisis and environment degradation at global level. On the many alternatives, solar energy is an important renewable energy resource that has the potential of fulfilling all energy needs. Some important applications of solar energy are solar water heating, solar space

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

heating/cooling, solar cooking, solar crop drying, solar power generation etc. Simplest method to utilize solar radiation is to convert it into thermal energy for heating applications by using solar collectors. Solar air Dryer because of their inherent simplicity are cheap and are used for many domestic and commercial applications like space heating, crop drying, wood seasoning etc. In this paper the objective of the CFD flow study is to design, test and optimize flow-conditioning devices, as appropriate, to guide the gas flow through the duct. In this a two-dimensional numerical simulation of the heat transfer, Nussult number, Velocity and temperature was conducted using the CFD code FLUENT VERSION 14.5. The CFD modeling involves numerical solutions of the conservation equations for mass, momentum and energy. These three equations are used to model the convective heat transfer process with the following assumptions; (a)The flow is steady, fully developed, turbulent and two dimensional. (b) Incompressible fluid and flow.(c) The duct wall, absorber plate and roughness material are homogeneous and isotropic.

THE MAIN MODULES ARE

- Part Design
- Assembly
- Drawing
- Sheet Metal

MODEL OF AC DUCT

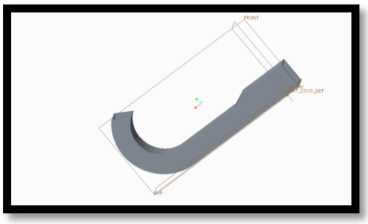


Fig 5.1 Model of AC duct

2D MODEL

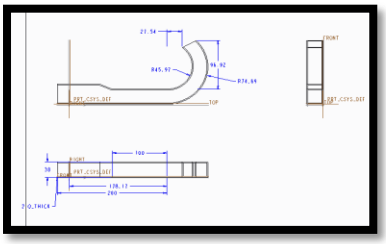


Fig 5.2 2D Model of Duct

CFD ANALYSIS OF AC DUCT PRESSURE

➤ PRESSURE

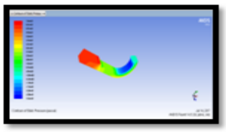


Fig 9.5 Pressure (Inlet velocity 7.5 m/s)

➤ VELOCITY

Fig 9.6 Velocity(Inlet velocity7.5 m/s)

➤ TEMPERATURE

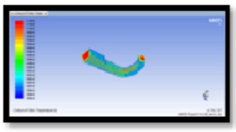


Fig 9.7 Temperature (Inlet velocity 7.5 m/s)

➤ HEAT TRANSFER COEFFICIENT

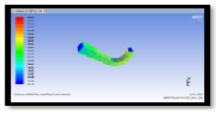


Fig 9.8 Heat Transfer Coefficient (Inlet velocity 7.5 m/s)

MASS FLOW RATE (Inlet velocity 7.5 m/s)

Mass Flow Rate	(kg/s)
inlet	0.011025003
interior-____msbr	0.85617602
outlet	-0.011031443
wall-____msbr	0
Net	-6.4400956e-06

INLET VELOCITY - 15 m/s

➤ PRESSURE

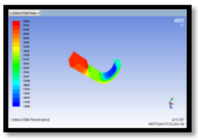


Fig 9.9 Pressure (Inlet velocity 15 m/s)

➤ VELOCITY

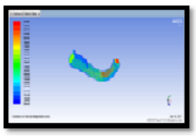


Fig 9.11 Temperature (Inlet velocity 15 m/s)

➤ TEMPERATURE

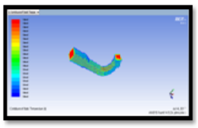


Fig 9.11 Temperature (Inlet velocity 15 m/s)

➤ HEAT TRANSFER COEFFICIENT

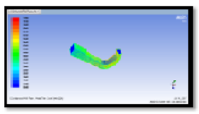


Fig 9.12 Heat Transfer Coefficient (Inlet velocity 15 m/s)

MASS FLOW RATE (Inlet velocity 15 m/s)

Mass Flow Rate	(kg/s)
inlet	0.022050006
interior-____msbr	1.7132169
outlet	-0.022078812
wall-____msbr	0
Net	-2.8805807e-05

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

INLET VELOCITY – 22.5 m/s

➤ PRESSURE

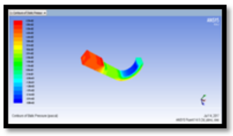


Fig 9.13 Pressure (Inlet velocity 22.5 m/s)

➤ TEMPERATURE

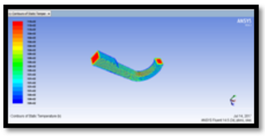


Fig 9.15 Temperature (Inlet velocity 22.5 m/s)

➤ HEAT TRANSFER COEFFICIENT

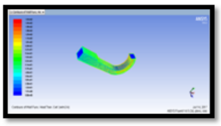


Fig 9.16 Heat Transfer Coefficient (Inlet velocity 22.5 m/s)

VELOCITY

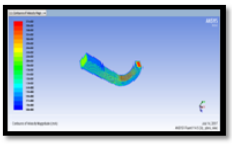


Fig 9.14 Velocity (Inlet velocity 22.5 m/s)

MASS FLOW RATE (Inet velocity 22.5 m/s)

Mass Flow Rate	(kg/s)
inlet	0.033074994
interior-____msbr	2.5711396
outlet	-0.0331119
wall-____msbr	0
Net	-3.6906451e-05

INLET VELOCITY – 30 m/s

➤ PRESSURE

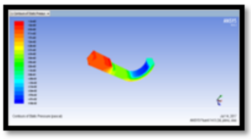


Fig 9.17 Pressure (Inlet velocity 30 m/s)

➤ VELOCITY

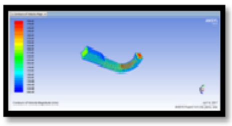


Fig 9.18 Velocity (Inlet velocity 30 m/s)

➤ TEMPERATURE

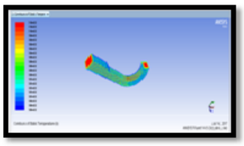


Fig 9.19 Temperature (Inlet velocity 30 m/s)

➤ HEAT TRANSFER COEFFICIENT

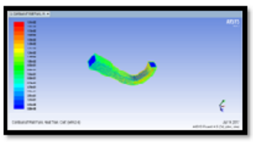


Fig 9.20 Heat Transfer Coefficient (Inlet velocity 30 m/s)

MASS FLOW RATE (Initial velocity 30 m/s)

Mass Flow Rate	(kg/s)
inlet	0.044100013
interior-____msbr	3.4281702
outlet	-0.044150792
wall-____msbr	0
Net	-5.0779432e-05

INLET VELOCITY - 37.5 m/s

➤ PRESSURE

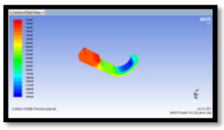


Fig 9.21 Pressure (Inlet velocity 37.5 m/s)

➤ VELOCITY

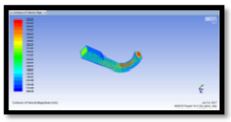


Fig 9.22 Velocity (Inlet velocity 37.5 m/s

➤ TEMPERATURE

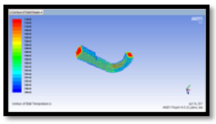


Fig 9.23 Temperature (Inlet velocity 37.5 m/s)

➤ HEAT TRANSFER COEFFICIENT

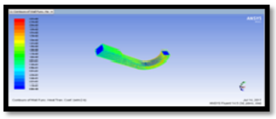


Fig 9.24 Heat Transfer Coefficient (Inlet velocity 37.5 m/s)

MASS FLOW RATE (Initial velocity 37.5 m/s)

Mass Flow Rate	(kg/s)
inlet	0.055124979
interior-____msbr	4.2853446
outlet	-0.055189796
wall-____msbr	0
Net	-6.4816326e-05

THERMAL ANALYSIS OF AC DUCT

Open work bench 14.5>select **steady state thermal** in analysis systems>select geometry>right click on the geometry>import geometry>select **IGES** file>open

MATERIALS GLASS FIBER

Thermal conductivity= 1.35 w/m-k

GALVANIZED IRON

Thermal conductivity= 76.5w/m-k

IMPORTED MODEL

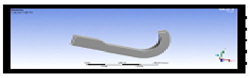


Fig 10.1 imported model

10.1 MESHED MODEL

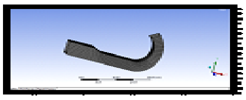


Fig 10.2 Meshed model

BOUNDARY CONDITIONS

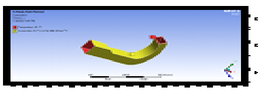


Fig 10.3 Boundary conditions

T =300K

Select steady state thermal >right click>insert>select convection> enter film coefficient value Select steady state thermal >right click>insert>select heat flux

Select steady state thermal >right click>solve

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

Solution>right click on solution>insert>select temperature

Heat transfer co-efficient values are taken from CFD analysis at different velocities

MATERIAL – GALVANIZED IRON

INLET VELOCITY - 7.5 m/s

➢ **Temperature Distribution**

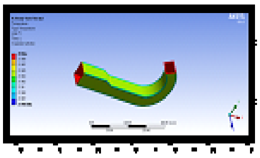


Fig 10.4 Temperature Distribution

➢ **Heat Flux**

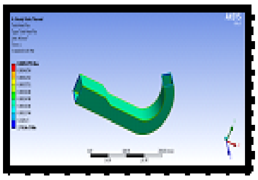


Fig 10.5 Heat Flux

INLET VELOCITY - 15 m/s

10.4.2 INLET VELOCITY - 15 m/s

➢ **Temperature Distribution**

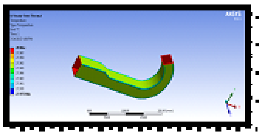


Fig 10.6 Temperature Distribution

➢ **Heat Flux**

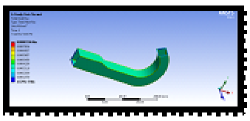


Fig 10.7 Heat Flux

INLET VELOCITY - 22.5 m/s

➢ **Temperature Distribution**

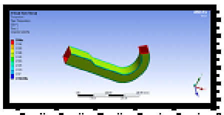


Fig 10.8 Temperature Distribution

➢ **Heat Flux**

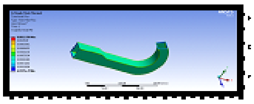


Fig 10.9 Heat Flux

INLET VELOCITY - 30m/s

➢ **Temperature Distribution**

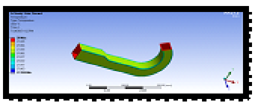


Fig 10.10 Temperature Distribution

➢ **Heat Flux**

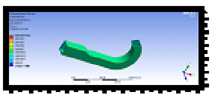


Fig 10.11 Heat Flux

10.4.5 INLET VELOCITY – 37.5 m/s

➢ **Temperature Distribution**

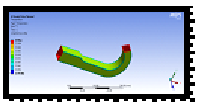
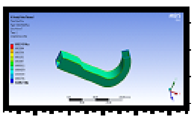


Fig 10.12 Temperature Distribution

➢ **Heat Flux**



MATERIAL – GLASS FIBER

INLET VELOCITY - 7.5 m/s

➢ **Temperature Distribution**

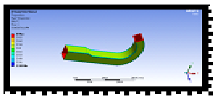


Fig 10.14 Temperature Distribution

➢ **Heat Flux**

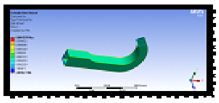


Fig 10.15 Heat Flux

INLET VELOCITY - 15 m/s

➢ **Temperature Distribution**

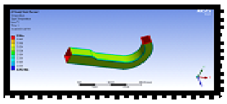


Fig 10.16 Temperature Distribution

➢ **Heat Flux**

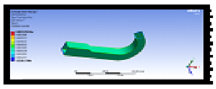


Fig 10.17 Heat Flux

INLET VELOCITY - 22.5 m/s

RESULTS TABLE

➢ **Temperature Distribution**

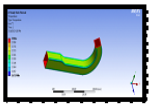


Fig 10.18 Temperature Distribution

➢ **Heat Flux**

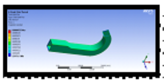


Fig 10.19 Heat Flux

10.5.4 INLET VELOCITY - 30 m/s

➢ **Temperature Distribution**

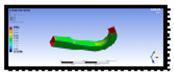


Fig 10.20 Temperature Distribution

➢ **Heat Flux**

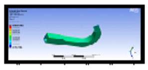


Fig 10.21 Heat Flux

10.5.5 INLET VELOCITY - 37.5 m/s

➢ **Temperature Distribution**

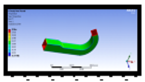


Fig 10.22 Temperature Distribution

➢ **Heat Flux**

CFD Analysis

Inlet velocity(m/s)	Pressure (Pa)	Velocity (m/sec)	Temperature (k)	Heat transfer coefficient (w/mk)	Mass flow rate(kg/sec)
7.5	7.45e+01	1.25e+01	3.10e+02	6.76e+01	0.000644
15	2.88e+01	2.51e+01	3.10e+02	1.13e+02	0.00002880
22.5	6.33e+02	3.77e+01	3.10e+02	1.53e+02	0.0000369
30	1.12e+03	5.03e+01	3.10e+02	1.91e+02	0.0000507
37.5	1.25e+03	6.29e+01	3.10e+02	2.27e+02	0.000064812

Table 10.2 Thermal Analysis

Material	Inlet velocity(m/s)	Temperature (k)		Heat flux(w/m)
		Min	Max	
Galvanized iron	7.5	27.925	28	0.00052255
	15	27.966	28	0.0008719
	22.5	27.958	28	0.0011788
	30	27.95	28	0.0014694
	37.5	27.303	28	0.0017439
Glass fiber	7.5	26.962	28	0.00045599
	15	26.719	28	0.00070458
	22.5	26.528	28	0.00089535
	30	26.373	28	0.0010553
	37.5		28	0.0011925

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

CONCLUSION

From the above result we got 306.759mpa stress
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 velocities7.5 ,15,22.5,30 and 37.5m/s. By observing the results, by increasing the velocity the pressure and velocity in the duct is increasing, outlet velocity 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 galvanized Iron at velocity 37.5m/sec, the heat transfer rate is more when galvanized iron 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.