

Umesh Chakotkar¹, Mohit Charjan², Sweta Bhongade³, Nilesh Shirgire⁴

¹Student, Mechanical Department, J.D.I.E.T. YAVATMAL, (M.S.), INDIA, chakotkarumesh@gmail.com

²Student, Mechanical Department, J.D.I.E.T. YAVATMAL, (M.S.), INDIA, mohit.charjan@gmail.com

³Student, Mechanical Department, J.D.I.E.T. YAVATMAL, (M.S.), INDIA, swetabhongade@gmail.com

⁴Assistant Professor, Mechanical Department, J.D.I.E.T. YAVATMAL, (M.S.), INDIA, nileshshirgire@yahoo.com

ABSTRACT

CFD Tools are the most versatile tools which are widely used for simulating the actual working condition of AC ducts to understand the behavior of air in working condition with pressure, velocity and temperature change. AC ducts carries a conditioned air which may be affected by duct surface while working due to change in temperature. Hence the efficiency and performance of duct may decrease as the surface affects the air temperature directly and pressure, velocity and density change indirectly. Hence it is found that the study of air behavior in AC duct is very important to know the nature of air flow in duct. There are several methods available to simulate air flow in AC duct. But the CFD analysis method is one of the best and simple method which gives us more approximate results. In this paper the change in velocity, pressure, density and temperature during working condition is obtained by using ANSYS Fluent CFD tool. Further conclusions are drawn based on these results.

Index Terms: CFD, ANSYS Fluent, AC Duct etc.

1. INTRODUCTION:

AC Ducts are the hollow sections which carry conditioned air. It is made up of sheet metal materials mostly. These are of either circular or rectangular in cross section. The main purpose of duct is to carry and transfer conditioned air from AC to rooms. A duct system is also called ductwork. Planning, sizing, optimizing, detailing, and finding the pressure losses through a duct system is named duct style.

Figure 1.1 shows the duct diagram used for this study. It has a circular cross section and conditioned air is generated at the rectangular section. Size and shape of duct varies as per the room size and location from central AC. Also the flow can be regulated with duct mountings. Duct must be sealed properly to avoid leakage.

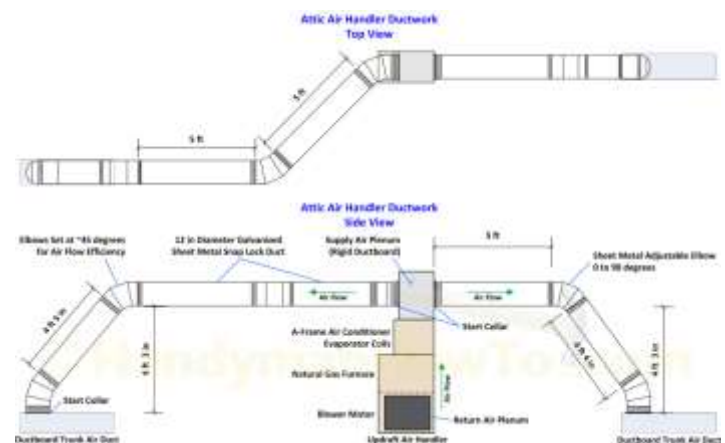


Fig. 1.1: Diagram of Duct used for this study.

Calculation of Duct performance is also an important part for which several methods are available. The performance of AC duct may go down as the duct contains leakage from various positions. Also the surface roughness inside the duct may affect the conditioned air temperature. Hence performance analysis of duct must be carried out periodically. This study also focused on the performance of duct with different cross

sections. Following are the various parameters which may include into a performance testing.

1.1 Performance on the basis of Velocity, Pressure and Density.

- Performance on the basis of Velocity:

Velocity at the outlet must be equal to or nearer to velocity at the inlet. Due to sudden turn in ducting may decrease the velocity and also affect the temperature of conditioned air. Hence the velocity must be nearer or equal to the inlet velocity.

- Performance on the basis of Pressure:

There should be no change in pressure due to ducting surface or sudden turn. Total work carried out by the air must be transferred to the outlet. Pressure variation may occur due to any obstacle in flow or sudden change in direction of flow. hence try to avoid unwanted turns in ducting.

- Performance on the basis of Density:

Amount of conditioned air content at the inlet must be same at the outlet. Mass flow rate in the entire ducting should be equally distributed in the flow path. density variation varies total mass content of air. hence it should be equal or nearer to the inlet value.

1.2 Effect of Change in Velocity, Pressure and Density.

Due to change in velocity the performance of AC duct varies. Also it affects the pressure and temperature of the conditioned air.

Air velocity in ducts is a critical factor in how well ducts do their job of efficiently and quietly moving the right amount of air from one place to another. Change in velocity will change pressure, density and temperature. Same phenomenon may occur with other parameters.

2. CAD MODEL PREPARATION:

PTC Creo, formerly called as Pro/ENGINEER and Wildfire, is a 3D CAD, CAM, CAE, and associative solid modelling app. It is only a suite of 10 collaborative applications that provide solid modelling, assembly modelling, 2D orthographic views, selected element analysis, direct and parametric modelling, sub-divisional and NURBS surface modelling, and NC and tooling functionality for mechanical designers. We have used this software for the preparation of AC duct.

Figure 2.1 shows the AC duct CAD model developed in Pro-E software. Part modelling commands are used for this task.

2.1 Dimensional Properties

Dimensions are obtained from Figure 2.1 which is the actual AC duct located in showroom/malls.

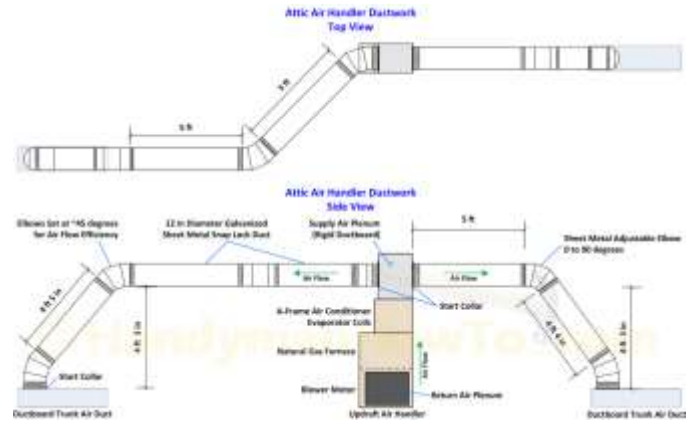


Figure 2.1: AC duct diagram.

2.2 Reverse Engineering Process.

Reverse engineering, also called as back engineering, it is the process by which a man-made object is deconstructed to find its designs, architecture, or to extract knowledge by the object; equal to scientific research, the major difference being that scientific research is a natural phenomenon.

As computer-aided design (CAD) has become a famous, reverse engineering has become a viable method to design a 3D virtual model of an old physical part for use in 3D CAD, CAM, CAE or other software. The reverse-engineering process made of measuring an object and then reconstructing it as an 3D model. The physical object also measured by 3D scanning technologies like CMMs, laser scanners, structured light digitizers, Industrial CT Scanning (computed tomography). The measured data single usually represented as a point cloud, lacks topological information and design intent.

2.3 CAD Tool

Pro-E was created by parametric Technology Corporation, Waltham, MA. Pro-E is a multi-programs software that can also be used for design, analysis and manufacturing. In this part of the course we will be dealing with the first capability of ProE which deals with the model creation and assembly of designs. But, ProE is linked to another package, MECHANICAL, which is able to perform structural, thermal and motion analysis. The structural analysis includes static stress, deformation, buckling and vibration. ProE and MECHANICA can be used for optimization and parametric studies. Connecting ProE to NC machines provide the manufacturing capabilities. Wide range of sheet metal forming, piping layout and model designs also handled.

2.4 Neutral File Format

Neutral file exchange used an intermediary neutral format in order to translate data between CAD systems. This method starts from a pre-processor embedded in the original CAD system, which generates the neutral file from the originating CAD format. The target CAD system post-processes the neutral file and converts it into the target native format. Some neutral formats are explained by standards organisations such

as IGES and STEP while others are proprietary but still widely used and are regarded as quasi industry standards.

IGES (Initial Graphics Exchange Specification) –This file format considers the product definition as a file of entities, with each entity being represented in an application-independent format. After the release of STEP (ISO 10303) in 1994, interest in future development of IGES declined, and Version 5.3 (1996) was the last published standard.

3. PARAMETERS FOR SIMULATING WORKING CONDITION:

Inlet Velocity, pressure at the inlet, temperature at inlet are the important parameters for simulating the working condition.

3.1 Initial velocity, pressure and temperature

- Inlet Velocity: The value of inlet velocity along with hydraulic diameter must be assigned for simulation. without this value we cannot proceed further.
- Inlet Pressure: It is the optional value which can be assigned by the user, if it is known. Atmospheric pressure value is the default value for this simulation.
- Inlet Temperature: This value is also optional which must be assigned by the user, in order to simulate actual working conditions.

3.2 Boundary Conditions

- Material Properties: The value of thermal conductivity, Coefficient of thermal expansion, density, young's modulus, specific heat is the required properties of metals.
- Fluid Properties.: Density, velocity, pressure, temperature, volume etc. are the important properties of fluid.
- Geometrical Properties: Cross section, bend, sudden contraction/enlargement etc. are the main geometrical properties.

4. CFD ANALYSIS OF AC DUCT:

Fluent software contains the broad, physical modelling capabilities required to model flow, turbulence, heat transfer and reactions for industrial applications. These range from air flow above an aircraft wing for combustion in a furnace, from bubble columns to oil platforms, from blood flow to semiconductor manufacturing and from clean room design to wastewater treatment plants. Fluent spans an high range, including special models, with capabilities to model in-cylinder combustion, aero-acoustics, turbomachinery and multiphase systems.

4.1 Meshing

Mesh generation is the practice of creating a polygonal or polyhedral mesh that approximates a geometric domain. The term "grid generation" is often used interchangeably. Typical uses are for rendering to a computer screen or for physical simulation like finite element analysis or computational fluid dynamics.

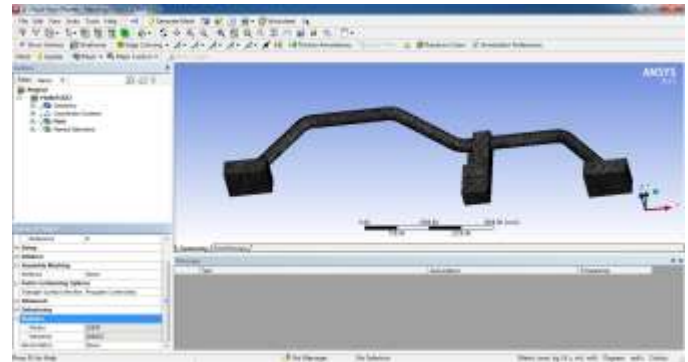


Figure 4.1: Meshing of AC Duct.

4.2 ANSYS Fluent as a CFD Tool

Computational fluid dynamics (CFD) is a tool having an amazing flexibility, accuracy and breadth of application. But serious CFD, the kind that gives insights to help you optimize your designs, can be out of reach unless you choose your software carefully. To get serious CFD results, we need serious software. ANSYS CFD goes beyond qualitative results to give accurate quantitative predictions of fluid interactions and trade-offs.

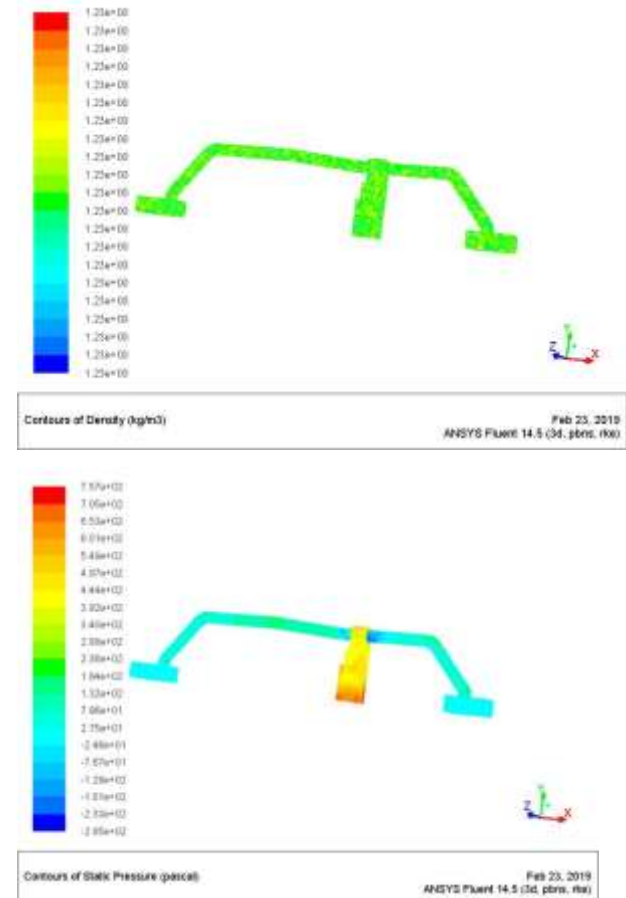


Figure 4.2: Density and pressure counter obtained through CFD Analysis.

5. RESULT FROM CFD TOOL:

Table 5.1: Results from CFD Tool at the inlet of Circular Duct

Sr. No.	Parameter	Value
1	Number of Nodes	21847
2	Number of Elements	104023
3	Maximum Velocity	32.5 m/s
4	Maximum Pressure	7570 Pascals
5	Maximum Density	1.23 kg/m ³
6	Maximum Temperature	300 k
7	Maximum Turbulence	56.3 m ² /s ²

Table 5.2: Results from CFD Tool at the Middle of Circular Duct

Sr. No.	Parameter	Value
1	Maximum Velocity	17.4 m/s
2	Maximum Pressure	10457 Pascals
3	Maximum Density	1.23 kg/m ³
4	Maximum Temperature	295 k
5	Maximum Turbulence	42.11 m ² /s ²

Table 5.3: Results from CFD Tool at the End of Right side Circular Duct

Sr. No.	Parameter	Value
1	Maximum Velocity	32.5 m/s
2	Maximum Pressure	7570 Pascals
3	Maximum Density	1.23 kg/m ³
4	Maximum Temperature	300 k
5	Maximum Turbulence	56.3 m ² /s ²

6. CONCLUSION:

- Velocity at the outlet is far greater than velocity at inlet.
- Pressure value decreases with increase in velocity.
- CFD Analysis gives more specific results for flow parameters.

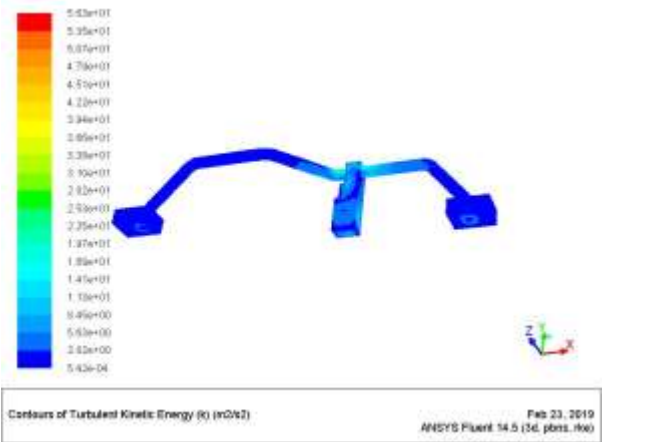
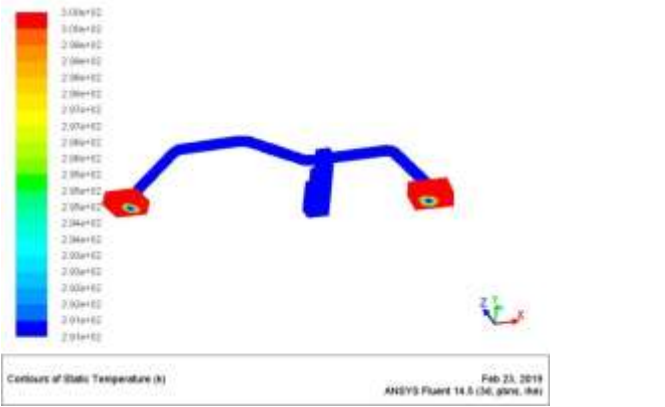


Figure 4.3: Temperature and Terbulance counter obtained through CFD Analysis.

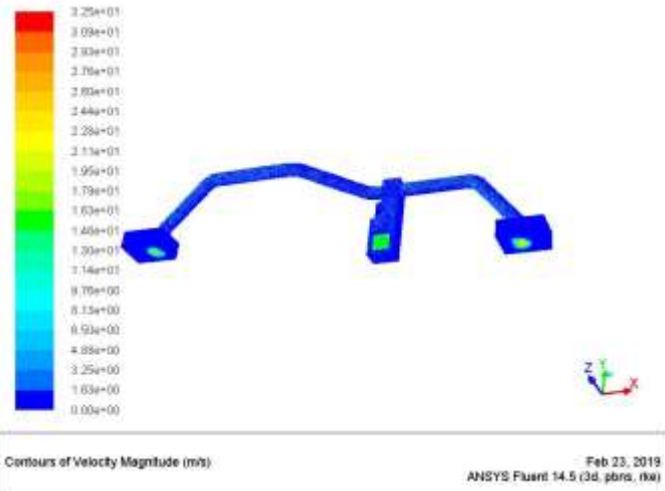


Figure 4.4: Velocity Counters obtained through CFD Analysis.

REFERENCES:

- [1] "Advanced Strategy Guideline: Air Distribution Basics and Duct Design", Arlan Burdick, IBACOS, Inc., December 2011
- [2] " Laminar Free Convection Heat Transfer through Horizontal Duet Connecting Two Fluid Reservoirs at Different Temperatures", A. Bejan and C. L. Tien
- [3] "Performance analysis of ducted marine propellers. Part II –Accelerating duct" R.ontempo*, M. MannaDipartimento di Ingegneria Industriale, Università degli Studi di Napoli Federico II, Via Claudio 21, 80125 Naples, Italya, Applied Ocean Research journal homepage: www.elsevier.com/locate/apor
- [4] **Riccardo Mereu***Politecnico di Milano Department of Energy Milan, Italy, "Preliminary Fluid Dynamic Analysis Of Turbulent Flat And Ribbed Square Duct Via Cfd Approach"
- [5] Bradley S. Hurak " Computational Fluid Dynamics Analysis of Air Flow and Temperature Distribution in Buildings"
- [6] "Stabilisation of buoyancy driven unstable vortex flow in mixed convection of air in a rectangular duct by tapering its top plate", W.S.Tseng
- [7] "Calculation method for a complete air conditioning study [case study of an industrial installation]", Charisis G. Vrellas
- [8] "Aircraft air conditioning heat exchanger and atmospheric fouling", S.J.Write, D.Dixon-Hardy and P.J.Heggs
- [9] "A review on CFD analysis in air conditioning system" Sumesh Pillai
- [10] "Pressure losses analysis in air duct flow using computational fluid dynamics", Ekpe Aniekan Essienubong
- [11] "Analysis of turbulent flow over a 90 degree bent of duct using a centralized AC plant by CFD code", Mukesh Didwania
- [12] "CFD modeling and validation of temperature and flow distribution in air conditioning space", Jahar Sarkar
- [13] "Comparative study of air conditioning duct and diffuser in automobile using CFD", G. Sudhakar
- [14] "An investigation of air flow and thermal comfort of modified conventional car cabin using CFD", S.Ibrahim