Issue 1 vol 4 ISSN: 2321-8134

IJFEAT

INTERNATIONAL JOURNAL FOR ENGINEERING APPLICATIONS AND TECHNOLOGY

TITLE: EFFECT OF SUDDEN CONTRACTION IN RECTANGULAR AC DUCT FOLLWED BY PRESSURE, TEMPRATURE AND VELOCITY CHANGE BY PERFORMING CFD ANALYSIS

AJINKYA RAUT[1], SHUBHAM DHEKALE[2], SHUBHAM ZADE[3], PROF. N.D.SHIRGIRE[4]

[1]STUDENT, MECHANICAL DEPARTMENT, JDIET , MAHARASTRA,INDIA**, ajinkyarautme@gmail.com**
[2]STUDENT, MECHANICAL DEPARTMENT, JDIET , MAHARASTRA,INDIA, **shubhamdhekale93@gmail.com**
[3]STUDENT, MECHANICAL DEPARTMENT, JDIET , MAHARASTRA,INDIA**, shubham8237375733@gmail.com**
[4]Asst. Professor, MECHANICAL DEPARTMENT, JDIET , MAHARASTRA,INDIA, **nileshshirgire@yahoo.com**

ABSTRACT
Rectangular AC duct is commonly used and well known type of ducting system which is implemented for the transport of conditioned air. These are cost effective and sophisticated ducting systems. At many places we find the cross section of duct decreases gradually which may affect the air flow inside the duct. It may be done intentionally to control flow inside the duct. But if it is not, then the study of their effects on the flow must be done. In this paper, the CFD analysis of rectangular duct is performed which has sudden contraction in the cross section. The duct model is performed on the CAD Tool called CATIA and further converted into IGES format to import it into ANSYS Fluent software. Effect on pressure, temperature and velocity is recorded.
Keywords: AC Duct, CAD Tool, CFD Analysis, ANSYS Fluent, IGES

---------- *** ----------

1.INTRODUCTION

AC Ducts are of two types mainly, rectangular and circular. These are hollow metal sheet structures used to carry conditioned air from central unit to various locations/rooms.
These types of ducts are produced from galvanised steel sheets, Stainless steel sheets or other type of metal sheets are design to comply SMACNA and ASHRAE standards. These are designed to withstand at strong vibrations, pressure and loading condition

Figure 1.1: Rectangular Duct with sudden contraction and turns.

- The Advantages of Rectangular Air Ducts

Rectangular ducts are necessary whenever we have got a tight space that won't allow for the height of round ducts.

- Takes up less height
- Easily connects fan coil to main duct
- Better for low pressure systems

- The Disadvantages of Square Ductwork

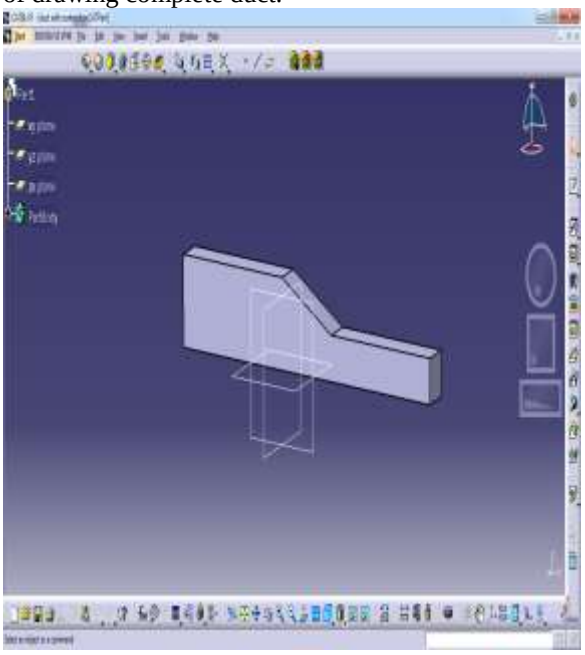
They allow for more noise to escape from the duct. Rectangular ducts also cost more to install initially.

- Higher noise pollution
- Costs more for initial installation
- **Table 1.1:** Common specifications of AC Duct.

Duct Wall Thicknesses	30 mm
Mean Air Velocity (Max.)	25.4 m/s
Design Pressure (Max.)	1000 Pa Positive 750 Pa Negative

Operating Temperature Limits	-20°C to +80°C

- 2. Geometry modeling by using CAD
- CATIA v5R19 is th software used for modeling of Rectangular AC duct. For this purpose several commands like pad, multi section solid etc are used. All the dimensions required for the modeling of duct are obtained from actual AC duct installed in a building. Here we have modeled only the sudden contraction section of rectangular duct.
- CATIA is one of the most popular and highly used CAD software for the design and development of engineering components. It consist of several modules like sketcher, part, assembly, drafting etc. We have used part module to develop duct model. Figure 2.1 shows the rectangular AC duct model which is developed in CATIA software. It is only half model of actual duct. It is only because we can perform symmetry conditions in CFD software. Hence no need of drawing complete duct.



- **Figure 2.1:** CAD model of rectangular AC duct.

Same model can be imported directly into ANSYS software by using some neutral file formats like step, IGES, parasolid. We have used IGES (.igs) format to import the duct into CFD tool. Further CFD Analysis is performed on the rectangular duct.

- **3. Rectangular Duct CFD Analysis.**
- We have used ANSYS 14.5 software to perform CFD Analysis on Rectangular duct. ANSYS Fluent is the module which is used for the performance of CFD analysis. the basic simulating conditions are applied to the model.
- ANSYS is the one of the leading CAE software which gives the solution of CAE problems by means of various types of analysis. There are several modules are available to perform analysis in ANSYS workbench
- window like static structural, steady state thermal, Modal, Fluid Flow (Fluent) etc. We can use our required type of tool for the analysis.
- **3.1 Boundary conditions and properties of fluid required.**
- To perform any type of analysis we need to apply proper boundary conditions. These boundary conditions will decide the accuracy of our analysis. Hence the proper input must be given for the better results.
- Following Boundary conditions are applied at the inlet of the duct.
- **Table 3.1**: Boundary conditions for performing CFD Analysis.

Property	**Value**
Inlet Velocity	15 m/s
Temperature	288 K
Pressure	1 Bar
Wall Thickness	1.5 mm
Atmospheric Temperature	300 K

We also need to apply the properties of Air for the CFD analysis. Following figure 3.1 shows the properties of air.

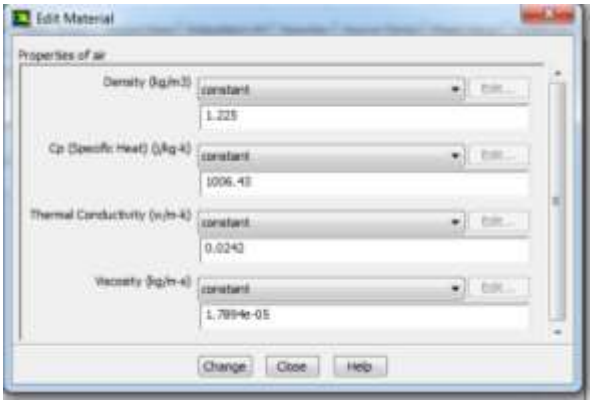


Figure 3.1: Properties of Air.

3.2 Meshing

This process is also called as discretization. In this process entire model is divided into small number of pieces called elements and they are connected together by means of points called nodes. Following figure 3.2 shows the meshed view of rectangular duct along with number of nodes and elements.

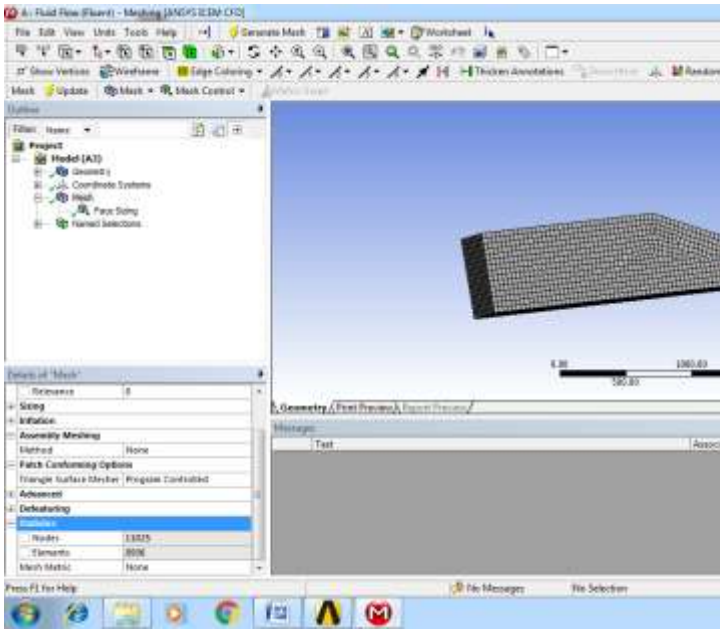


Figure 3.2: Meshed view with number of nodes and elements.

3.3 Performance of CFD Analysis

By applying all the boundary conditions and required properties, CFD analysis is to be performed. It involve following steps.

1) Importing of geometry into module.
2) Performing meshing operation on imported geometry.
3) Applying boundary conditions.
4) Applying air properties.
5) Applying velocity, pressure, temperature, hydraulic diameter etc.
6) Performance of solution intervals.
7) Result obtaining and saving.

3.4 Results obtained by performing CFD Analysis.

After performing all necessary steps for CFD analysis, we can obtain results. Following are the results which are obtained by performing CFD analysis. All these results are for the rectangular section duct with sudden contraction.

- **For Rectangular AC Duct:**

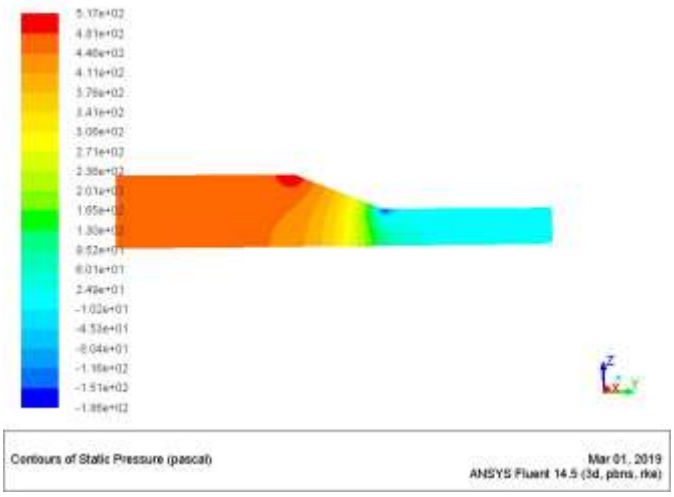


Figure 3.3: Pressure Counter obtained through CFD Analysis.

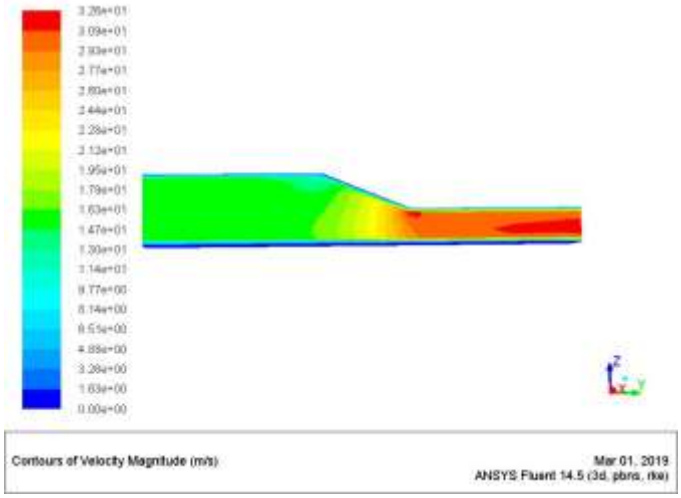


Figure 3.4: Velocity Counters obtained through CFD Analysis.

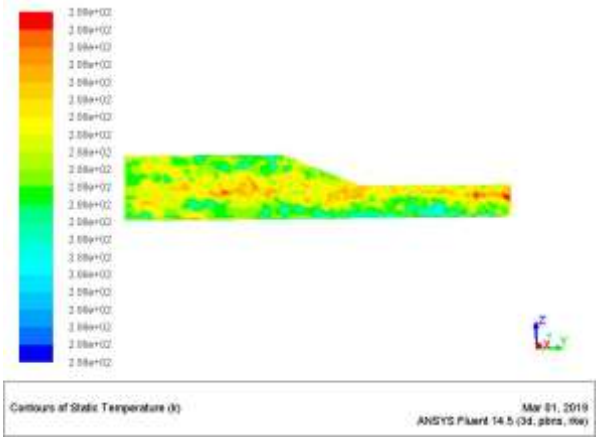


Figure 3.5: Temperature Counters obtained through CFD Analysis.

4. Result Comparison

Table 4.1: Results from CFD Tool at the inlet Duct

Sr. No.	Parameter	Obtained Values
1	Number of Nodes	11025
2	Number of Elements	8936
3	Maximum Velocity	15 m/s
4	Maximum Pressure	200 Pascals
5	Maximum Density	1kg/m^3
6	Maximum Temperature	288 k

Table 4.1 shows the values of all parameters which are applied as a boundary condition. No changes in the values as we are at inlet.

Table 4.2: Results from CFD Tool at the Middle of Duct

Sr. No.	Parameter	Obtained Values
1	Maximum Velocity	32 m/s
2	Maximum Pressure	130 Pascals
3	Maximum Density	1.23 kg/m^3
4	Maximum Temperature	290 K

Table 4.3: Results from CFD Tool at the End of Duct

Sr. No.	Parameter	Obtained Values
1	Maximum Velocity	32.6 m/s
2	Maximum Pressure	24.9 Pascals
3	Maximum Density	1.2 kg/m^3
4	Maximum Temperature	292 K

By observing all the values and the flow conditions the following conclusion is drawn.

5. Conclusion

Pressure drop accurse due to the sudden contraction in the duct. As the pressure drops, there is sudden increment in the velocity as shown in velocity counter diagram. No major changes were found in the temperature value. Hence the velocity and pressure values are the most affected values in the case of sudden contraction.

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