

Design and Optimization of a Duct in Different Working Conditions for Circular and Elliptical Section

A Gokul karthik¹, T.K. Aravindhan^{2*}, P. Girithararajan³, R. Jatin⁴

¹ Assistant professor, ^{2,3,4} UG students,

¹⁻⁴ Department of Mechanical engineering, Sri Shakthi institute of engineering and technology, Coimbatore, Tamil Nadu, India

Email: ²tkaravindhan1997@gmail.com

Abstract

The objective is to evaluate and compare the computational fluid dynamic analysis of duct under three types of working conditions namely home, industry and restaurant. In this, a fluent analysis is conducted on elliptical and circular cross section duct to obtain the variation of velocity and pressure magnitude at critical locations of duct. The computational fluid dynamic analysis is done and verified by simulations in ANSYS workbench. Results achieved from a mentioned analysis were used in optimization of the duct used for various applications like home, industry and restaurant. Geometry is analyzed for different working conditions under different velocity inputs.

Keywords: Duct, HVAC, Different working conditions, Circular and Elliptical section.

INTRODUCTION

HVAC

HVAC (heating, ventilating, and air conditioning; additionally heating, air flow, and air con) is the generation of indoor and vehicular environmental comfort. Its purpose is to offer thermal consolation and suitable indoor air first-class. HVAC is crucial inside the design of medium to big industrial and office buildings which includes skyscrapers, onboard vessels, and in marine environments along with aquariums.

Ventilation

Ventilating or ventilation is the procedure of "changing" or replacing air in any area to offer excessive indoor air exceptional which entails temperature control, oxygen replenishment, and removal of moisture, odors, smoke, warmness, dust, airborne bacteria, and carbon dioxide. Ventilation removes ugly smells and immoderate moisture, introduces outside air, keeps indoors building air circulating, and forestalls stagnation of the indoors air.

Air Conditioning

An air conditioning system, or a standalone air conditioner, presents cooling and humidity control for all or a part of a constructing. Air conditioned homes frequently have sealed home windows, because open home windows would work against the gadget supposed to hold steady indoor air situations. Outside, sparkling air is usually drawn into the gadget by way of a vent into the indoor warmth exchanger phase, creating positive air stress.

LITERATURE SURVEY

Santosh Kansal and Krishna Kumar Waghe described about performance analysis of elliptical cross sectioned bends of HVAC duct for less pressure drop. In this paper, a performance analysis of the elliptical cross sectioned bend of HVAC duct has been carried out. Static pressure loss in HVAC ducts system is most concern, and in this a theoretical analysis is done using computational fluid dynamics CFD [1].

**Mukesh Didwania, Lokesh Singh,
Ashish Malik and Mangal S Sisodiya**

Described about analysis of turbulent flow over a 90° bend of duct using in centralized AC plant by CFD code. This test case focuses on the use of approximate models such as the turbulence models to predict the physical characteristics of the turbulent flow around a 90° bend by increasing the bend length in duct, losses can be reduce [2].

Shweta Deshpande and Dr. R. S. Bindu presented FEA based frequency optimization of duct with variable cross section through structural modifications for avoiding resonance. In this paper, different methods to improve natural frequency of variable cross sectioned ducts are discussed briefly of structural modifications for frequency improvement [4].

METHODOLOGY

Modeling

The duct model has been entirely modeled by SOLIDWORKS software. First of all, sketch command of the solid works is opened. Then by using 2d commands sketch is created.

Transformation of Model

Then the model is converted in to the IGES format which is most suitable and easy access for any other software's. Using the IGES format we can import the duct model from SOLIDWORKS to ANSYS.

Mesh

After the complete structure is modeled, duct is meshed. The last step to be completed before meshing the model is to set the meshing controls. These areas can be remised with advance mesh control options. "Smart element sizing" is a meshing feature that creates initial element sizes for free meshing operation.

Loading

12 types of loading that can be applied in a computational fluid dynamics analysis include: Internal and external velocities.

DESIGN OF DUCTS

Introduction to Solid works

Solid works (originally Solid Works) is solid modeling CAD (computer-aided design) software that runs on Microsoft windows and is since 1997 produced by Dassault Systems SOLIDWORKS Corp., a subsidiary of Dassault Systems.

Tools Used

Create the working directory-First create the working directory to save the all model in one folder File – set working directory – selects the required folder – ok.

Sketch- This command is used to create the new sketch like circle, line, rectangle, ellipse, etc. The solid works window select the sketch icon and select the plane or surface want to sketch.

Circle- This command is used to create the circle. Create circle by picking the center point and a point on the circle from right click the middle mouse button – ok

Ellipse- This command is used to create the ellipse. Create ellipse by picking the center point and a minor radius point and major radius point. Pick the center for the ellipse – pick the minor radius of ellipse point and pick the major radius of the ellipse- click the middle mouse button – ok

Line- This command is used to create the line. Create the line by start point and endpoint. Select the line icon from the right Tool chest – click the start point of the line – click the end point of the circle – enter

Arc- This command is used to create the arc. Create the arc by using three points. The three points are start point, end point and center point of the arc. Click the end point of the arc and click the middle point of the arc –enter. The dimension of the arc is modified by double click on the arc then the dimension will appear in the pop up box.

Create the Hexagon– The hexagon is created by insert foreign data icon in the Right Tool chest Insert foreign data from palette into active object- scroll down to see the hexagon- double-click **hexagon**- Place the hexagon on the sketch by picking a position – with the left mouse button,

drag and drop the center of the hexagon at the origin – modify Scale value to the required size – click enter [3].

Rectangle- This command is used to create the rectangle and square. Click the rectangle icon in the right Tool chest – click the lower left point of the rectangle and higher right corner of the rectangle.

Extrude- This command is used to create the material (to make 3D object from 2Dsketch) from the sketched entities. The entities may be circle, line or rectangle, etc., enter the extrude length and press the middle mouse button to finish the extrude command.

Revolve- This command is used to create the material from taking the one axis and sketched entities. The axis is the center of the revolved part. The revolve angle should be between 0 degree to 360 degree. Select the revolve icon from the right tool chest then select the sketched part and axis of the object in the graphical window.

Sweep Features

The order of operation is to first create a trajectory and then a section. A trajectory is a path along which a section is swept. The trajectory for a sweep feature can be sketched or selected the section is swept along the sketched or selected trajectory.

SOLIDWORKS DRAWINGS

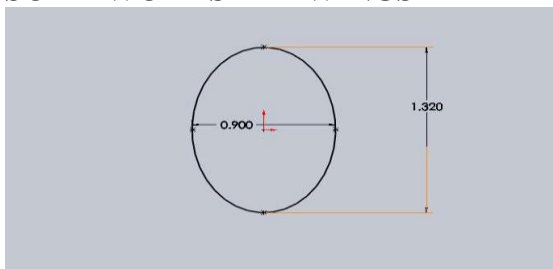


Figure 1 (a): Creating 2d sketch for elliptical duct.

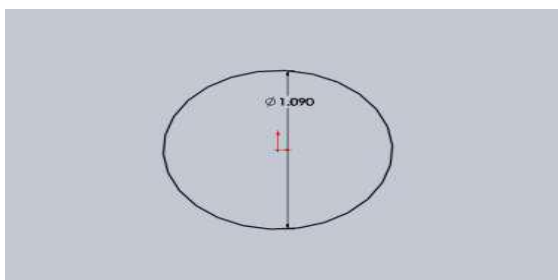


Figure 1 (b): Creating 2d sketch for Circular duct.

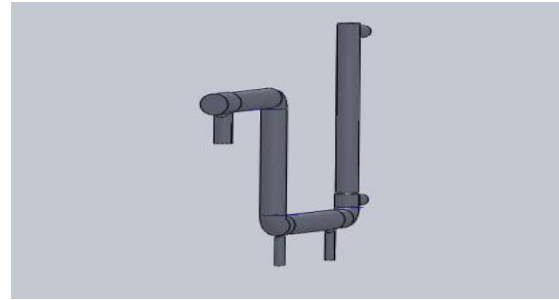


Figure 1 (c): Final view of elliptical duct.

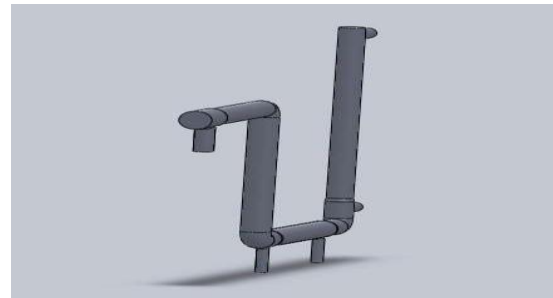


Figure 1 (d): Final view of circular duct.

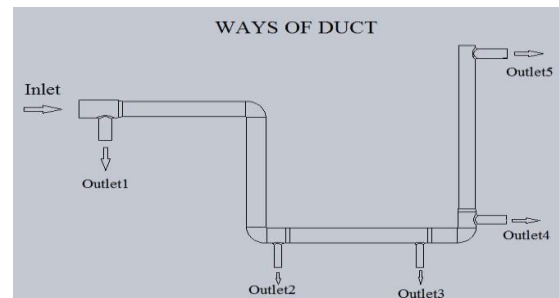


Figure 1 (e): Ways of duct.

SIMULATION

There are generally two types of analysis that are used in industry: 2-D modeling, and 3-D modeling. Capacities which may influence the framework to carry on directly or non-straightly. Straight frameworks are far less perplexing and for the most part don't consider plastic misshaping. Nonlinear frameworks do represent plastic distortion, and numerous additionally are equipped for testing a material right to crack [5].

Computational Fluid Dynamics

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, e.g. flight tests.

Simulation Details

Analysis Type- Internal Flow

Boundary condition

At home - 2m/s

At Industry - 6m/s

At Restaurant - 12m/s

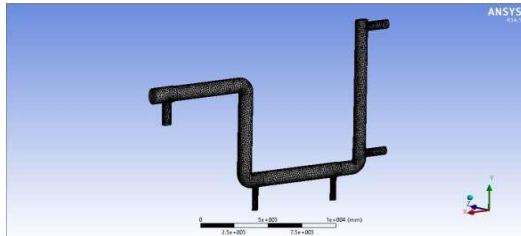


Figure 2 (a): Meshing of elliptical duct.

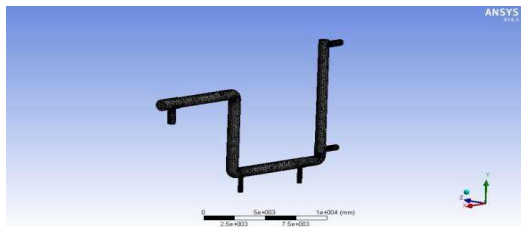


Figure 2 (b): Meshing of circular duct.

Step by Step Procedure

- The 3D model of the elliptical cross section duct is designed by using SOLIDWORKS software and it is converted as IGES format.
- The IGES (Initial Graphic Exchange Specification) format is suitable to import in the ANSYS Workbench for analyzing.
- Open the ANSYS workbench FLUENT (CFD)
- File – import external geometry file – generate
- Defaults – physical preference – mechanical
- Advanced – relevance center – fine
- Right click the mesh in tree view – generate mesh
- Project – convert to simulation – yes
- Right click on model – create named selection – inlet, outlet 1,2,3,4 and 5.
- New analysis – FLUENT (CFD)
- Fluent – double click

- Select the inlet surface of the duct – enter value for velocity.
- Double click the result icon in the tree.
- After solve the analysis – take the reading of pressure and velocity for elliptical cross section duct.
- The all results are taken in a picture – and save it to the required folder in the system.

CONCLUSIONS

Table 1: Results for circular and elliptical section ducts.

	Home		Industry		Restaurant	
Pressure	3.41	3.42	30.4	30.9	115.3	125
Velocity	4.3	4.3	13.2	13.6	25.8	27.5

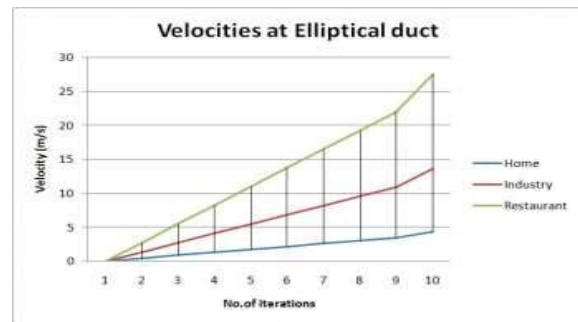


Figure 3: Graph for velocities at elliptical duct.

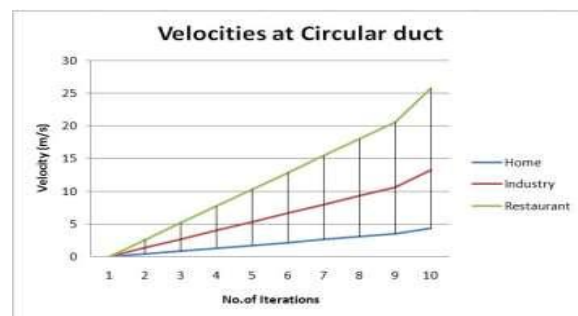


Figure 4: Graph for velocities at circular duct.

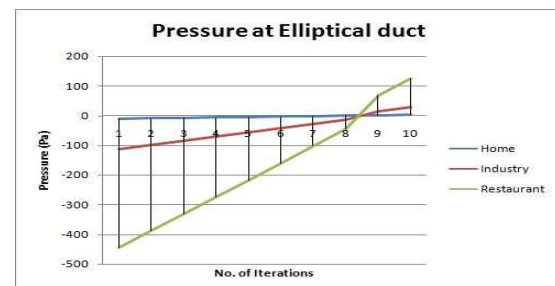


Figure 5: Graph for pressure at elliptical duct.

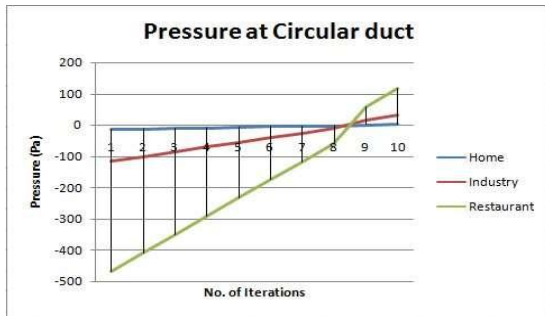


Figure 6: Graph for pressure at circular duct.

Here, I am giving brief introduction about duct and various authors' literature give an idea about duct design and various modifications. Elliptical cross section duct design is created by using the SOLIDWORKS software. Then the analysis will carried out in ANSYS workbench software.

In this project, various section duct is used under three types of working conditions namely home, industry and restaurant. The sections of duct are circular and elliptical. From the graphs duct in restaurant produce more pressure and velocity. While considering circular and elliptical ducts, elliptical duct produce high velocity and pressure. Compared with home and industry, industry produces more pressure and velocity. The high pressure at restaurant elliptical duct is 125.2Pa and circular duct is 115.3Pa. from computational fluid results the pressure drop is more in circular section duct. Therefore, elliptical cross section is good for duct which produces less pressure drop inside duct.

Doing this project we are study about the 3Dmodelling software (SOLIDWORKS) and study about the analyzing software (ANSYS workbench-14.5) to develop our

basic knowledge to know about the industrial design.

REFERENCES

1. Santosh Kansal, Krishna Kumar Waghe (June 2014), "Performance analysis of elliptical cross sectioned bends of HVAC duct for less pressure drop", *International Journal For Technological Research In Engineering*, Volume 1, Issue 10.
2. Mukesh Didwania, Lokesh Singh, Ashish Malik, Mangal S Sisodiya (Jul, Aug. 2014), "analysis of turbulent flow over a 90° bend of duct using in centralized plant by CFD code", *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)*, Volume 11, Issue 4.
3. 3. Neihad al-khalid (10-12 December 2003), "Design optimization of industrial ducts using computational fluid dynamics", *Third International Conference on CFD in the Minerals and Process Industries CSIRO*, Melbourne, Australia.
4. Shweta Deshpande RS Bindu, "FEA based frequency optimization of duct with variable cross section through structural modifications for avoiding resonance"
5. JJ Miao, TS Leu, JH Chou, SA Lin, CK Lin(1990), "Flow distortion in a circular-to-rectangular transition duct" *AIAA J.*, Volume 28, Issue 8.

Cite this article: