International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 04 Issue: 09 | Sept -2020 ISSN: 2582-3930

ANALYSIS TO PREVENT OVERSIZING OF DUCTS

B. Phanindra Kumar[1], N. Ashok Kumar[2], P. Pavan[3], P. Aravind[4], M. Arbazz[5]
[1]*Assistant Professor, Dept. of Mechanical Engineering, Guru Nanak Institute of Technology, Hyderabad, India*
[2,3,4,5]*UG Student, Dept. of Mechanical Engineering, Guru Nanak Institute of Technology, Hyderabad, India*

***Abstract*: This project focus on study of design modifications to a duct using computational fluid dynamics (CFD). In the power plant the duct is the major part of the system. The duct system that is used to release the fuel gas from the boiler outlet to the economizer input port. The present review is intended to encompass the fluid flow analysis and heat transfer related work analysis and research. Curved ducts with various shapes and sizes play a very important role in many applications like heat transfer, refrigeration and air conditioning etc.**

Flow rate and heat transfer in curved duct is different from straight ducts because of secondary flow. Flow of fluids like air, water or nanofluid are investigated with variation of aspect ratio, dimension of duct, Reynolds number etc and obtained result in form of average nussult numbers, friction factor etc.

***Keywords*: Velocity, pressure, Ducts.**

1. Introduction

The duct is an essential part of the air conditioner that carries the air flow rate. The main purpose of ducting system is to transfer the fuel gas from the boiler to the other systems like economizer and air preheater. By using the duct we can reduce the pressure drop and equal flow of the gas to the other systems. The duct can be made by using the Galvanized steel, aluminum, and Polyurethane and phenol insulation panels, fiberglass duct board. In this project we will come to know that the stability of a duct and pressure drops and how they determine the available static pressure, which then leads to the total effective length of our duct system. Related to this project we will come to know that various pressures acting on the duct as well as on the of the inner walls of the duct..

Many of our homes and most offices and commercial facilities would not become fordable without control of the indoor environment. The "luxury label" attached to air conditioning in earlier decades has given way to appreciate it practicality in making our live healthier and more productive. Along with rapid development in improving human comfort came the realization that goods could be produced better, faster, and more economically in a properly controlled environment. AutoCAD is the AutoCAD software for mechanical, electrical, and plumbing designers and drafters. Creation and coordination of construction documents is more efficient with AutoCAD more intuitive systems drawing and design tools. AutoCAD also assessing our vision and enhance our efficiency because of its purpose-built software for MEP designers and drafters

2. Literature survey

1.Madhulika singh, sham alam said that "CFD Approach to Design and Optimization of Air, Flue Gas Ducting System" In this paper mainly focused on ducting system is used to transfer of air or gas from one place to another place and chamfering the required section to have the uniform distribution of the fuel gas through the duct. And also pre circulate the gas before entered into the duct. The main objective of this paper is by using the CFD analysis to provide the improved flow condition in air and fuel gas in the duct and also to reduce the pressure loss and low turbulence. **2.A.Aravind kumar**, works on "Analyzing the gas flow in CFD for various ducts". In this paper focused on to form a vibration less duct and good flow distribution the number of plates provided in the inlet of the ducting system. The number of plates has increased in each case for reduce the vibration and also provide the even distribution of flow. The main view of this paper is analyzing the new economizer in CFD analysis And Finally the existing design is compared with the new modified design.

3.S.Srikanth, Dr. Benny paul, on his paper named "CFD analysis of waste heat boiler".In this paper focused on to guide the flow properly through the RA duct with guide plates to eliminate the vortex generation and circulation induced which was the primary cause for vibrations developed in the duct assembly. The vibration developed reduced Turbulent flow of the gas. So the suitable solution was found that the guide plates are introduced to reduce the vibration. And also reduce the cost of the process.

4.J.J. Bezuidenhout, Y. Yang and J.J. Eksteen has investigated waste heat boiler in which the waste-heat boiler is used within the supplied flash smelting process as the main dust and energy recovery unit. Computational fluid dynamics (CFD) is done within a study to model the flow analysis and heat transfer distribution throughout the ducts. This study focus on geometric modifications and calculations to the boiler, which includes elevation for the ceiling, placement of flow- obstructing baffles and also to the radiation plates parallel within the flow system.

5.**M N Rahman** Y et. al(2018) The CFD simulation result for Minor Operation Theatre demonstrated differences in real measurement about 19% for air flow velocity and for the 0.04% temperature. The errors that occurred for the uncontrolled mesh density and walls of Minor Operations Theatres in actual Minor Operations Theatre which hard to be quantified for CFD analysis.

Based on the observation, better location of AC unit must be proposed for better distribution of air flow in Minor Operation Theatre and it reveals that ANSYS Fluent can be utilized for air simulation in Minor Operation Theatre.

3.HVAC – HOW TO SIZE AND DESIGN DUCTS

Air flow problems have plagued the HVAC industry for years. No matter how much money you spend on a high-quality HVAC system, the equipment won't work at its best without properly designed and installed ductwork. Ducts that are not well-designed result in discomfort, high energy costs, bad air quality, and increased noiselevels..

3.1DUCTWORK DESIGNPRINCIPLES

The following basic terminology is extensively used in this course.

cfm: volume of air flow; cubicfeet/minute

fpm: velocity or speed of air flow;feet/minute

sq. ft: duct size or cross-sectional area; squarefeet

Air volume in cfm can be calculated by multiplying the air velocity by the cross-sectional area of the duct in square feet.

cfm = fpm xArea

Given any two of these three quantities, the third can be readily determined:

fpm =cfm/area

Area =cfm/fpm.

3.2 Gauge and Absolute Pressures

Gauge pressure is indicated on the gauge; absolute pressure is the total of the indicated gauge pressure plus atmospheric pressure. The general equation for absolute pressureis:

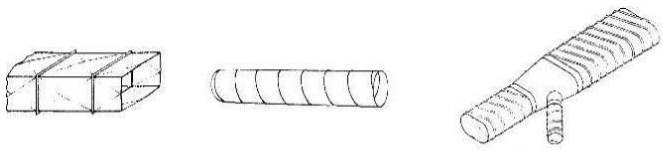
Gauge pressure + atmospheric pressure = absolute pressure

For example, if the gauge reads 10 psig then, using the above equation, the absolutepressure would be 24. 7psia:

10 psig + 14.7 psi = 24.7psia

Ordinary heating, ventilating, and air conditioning duct systems read air pressures at 0.4 psi or less, often much less. 1 psi equals 27.7 inches of water gauge; a common duct pressure of 0.25 inches water column is equal to (0.25 divided by 27.7 in-wc/psi) = 0.009 psi.

DUCTSHAPES



RoundDucts

The duct shape that is the most efficient (offers the least resistance) in conveying moving air is a round duct, because it has the greatest cross-sectional area and a minimum contact surface. In other words, it uses less material compared to square or rectangular ducts for the same volume of air handled.

An 18-inch diameter duct, for example, has the same air-carrying capacity as a 26" x 11" rectangular duct. The round duct has a cross-sectional area of 254.5 sq.-in and a perimeter of 4.7 ft., while the rectangular duct has a 286 sq.-in area and a perimeter of 6.2 ft. The rectangular duct thus has 32% more metal in it and would 17 cost proportionately more. Also the insulation, supports and labor are higher for rectangular ducts of similar capacity.

Advantages

Round shape results in lower pressure drops, thereby requiring less fan horsepower to move the air and, consequently, smallerequipment.

- • Round shape also has less surface area and requires less insulation when externally wrapped.
- • Round ducts are available in longer lengths than rectangular ducts, thereby eliminating costly field joints. Spiral lock-seams add rigidity; therefore, spiral ducts can be fabricated using lighter gauges than longitudinal seam ducts. Spiral ducts leak less and can be more easily sealed compared to rectangular ducts.
- • The acoustic performance of round and oval ducts is superior because their curved surfaces allow less breakout noise. The low-frequency sound is well contained in roundducts.
- • Round ducts can help promote healthier indoor environments. Less surface area, no corners and better air flow reduce the chance of dirt and grime accumulating inside the duct and, therefore, becoming a breeding ground for bacterialgrowth.
- • While round air ducts have great advantages, there are some disadvantages to them. One of the most notable drawbacks of round air ducts is that they need more clear height for installation. If the net clear height of a furred space above a suspendedceiling is 14 inches, an 18-in diameter duct cannot be installed therein; however, its equivalent 26" x 11" rectangular duct will fit the space easily. A combination of a rectangular plenum and round branches sometimes is a good compromise.

1.6.2 RectangularDucts

Square or rectangular ducts fit better to building construction. They fit above ceilings and into walls, and they are much easier to install between joists and studs.

When rectangular ducts must be used due to space limitations, keep the width-to- height ratio (aspect ratio) low. A rectangular duct section with an aspect ratio close to 1 yields the most efficient rectangular duct shape in terms of conveying air. A duct with an aspect ratio above 4 is much less efficient in use of material and 18

experiences great pressure losses. Aspect ratios of 2 to 3 are ideal in trading off added duct cost of material and fan energy for headroomsavings

EquivalentDiameter By definition, equivalent diameter (**Deq**) is the diameter of a circular duct that will give the same pressure drop at the same air flow as the rectangular duct..

Equivalent Diameter Approach What this means is that all three ducts, 30" round, 16" x 51" rectangular, and 16" x 53" flat oval willhave

the same friction loss for a given cfm. The table below summarizes the equivalent diameter approach**.**

OvalDucts Flat oval ducts have smaller height requirements than round ducts and retain most of the advantages of the round ducts. However, fittings for flat oval ducts are difficult to fabricate or modify in the field. Other disadvantages include:
• Difficulty of handling and shipping larger sizes;
• Tendency of these ducts to become rounder under pressure;and,
• In large aspect ratios, difficulties of assembling oval slip joints

Equivalent Cross-sectional AreaApproach
Let's see what happens when using an equivalent cross-sectional area approach. Diameter of duct = 30 inches or 2.5 ft. Cross-sectional area of the 30-inch diameter duct = 3.14* 2.5 * 2.5/4 = 4.91 sq.-ft.
Cross-section area of rectangular duct:
A = a * b
Fixing minor axis "b" as 16 inches;
Major axis "a" will be = 4.91* 144/16 = 44.2 inches.
Cross-sectional area of oval duct:
Fixing minor axis "b" as 16 inches; Major axis "a" will be = 47.6 inches

4. METHODOLOGY

The ducting system for a single-family house is designed on the basis of using Revit software which is commonly used for the is building information modelling software for architects, landscape architects, structural engineers, mechanical, electrical, and plumbing (MEP) engineers, designers and contractors. The original software was developed by Charles River Software, founded in 1997, renamed Revit Technology Corporation in 2000, and acquired by Autodesk in 2002. The software allows users to design a building and structure and its components in 3D, annotate the model with 2D drafting elements, and access building information from the building model's database. Revit is 4D building information modeling capable with tools to plan and track various stages in the building's life cycle, from concept to construction and later maintenance and/ordemolition.

4.1 Top view

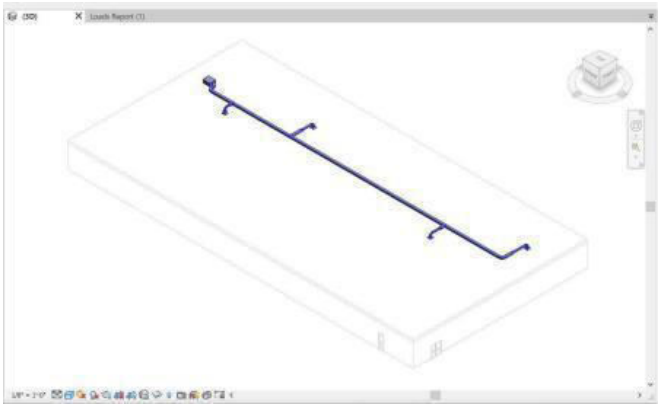


Fig. 4.1. Civil plan

This civil plan is design or build with the Rivet software that is provided with Base, Ceiling, Roofing and providing ducts and proper ventilation there by the heat load calculations.

4.2 wireframe view

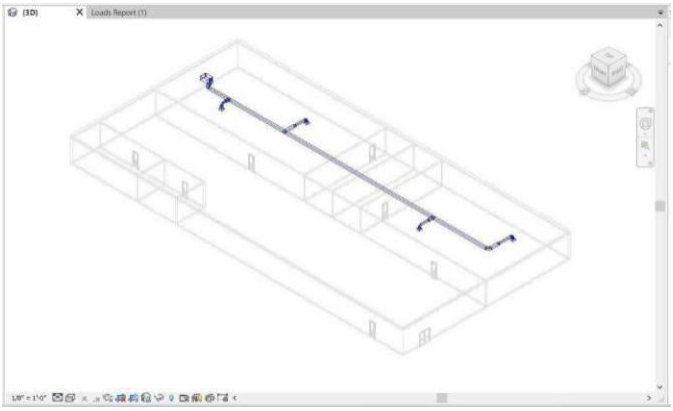


Fig. 4.2 Civil plan top view

4.3 Ducts 3D view

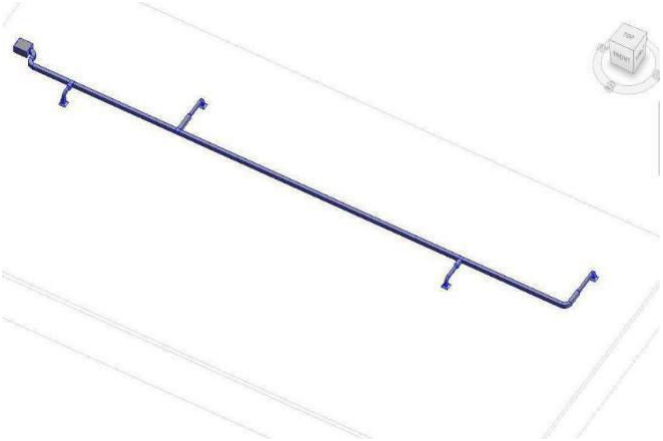


Fig. 4.3 Civil plan side view

4.4 planning

The paint and plastics manufacturing industry when manufacturing process undergoing the harmful toxic gases present in the building.so proper measures should take place for the removal of toxic gases in the plastic and paint industry.

5.Results and discussions

By this we conclude that CFD provides the power to quantify flow Improvement and to quickly predict behaviour of several designs and to prevent the oversizing of ducts by analysis of fluid flow. It gives the required output that includes the average velocity,pressure, turbulence, viscosity index Here we conclude that Computational fluid dynamics analysis provides the power to quantify flow improvement and to quickly predict the behaviour of several designs. The CFD analysis has offered a comprehensive range of output including velocity distribution, pressure profile and turbulencelevels.

Hence for the single family with the area of 7902 square feets and a volume of 63,212.57 cubic feet. the rooms with 4,269 sf and the volume of 34,155.18 cf will take the air flow is 716 cfm. Another room with the area of 3,632 sf and the volume is 29,057.39 of will take the air flow of 649 cfm. And the respective heat load calculations and colin calculations are viewed in the results and CFD analysis of a duct is done by the solid works software.

6.FUTURE SCOPE

In recent years, there has been growing interest by various engineering product companies to perform design simulation studies at different stages of product development to compete in the market. This has consistently resulted in increased requirement of skilled CFD resources and proving to be a very good career opportunity for engineers aspiring to make a career in the interesting domain of heat transfer and fluid flows. However there seems to be a widespread confusion in the student community as to what skills are desired by these industries for a fresher to qualify.

6.1CFD PROCEDURE

The CFD Process is carried out in following steps,

- Model generation
- Meshing
- Setup (boundary condition)
- Solution
- Result

FLUID FLOW ANALYSIS IN AIR DUCT FLOW WITH AND WITHOUT INTERNAL THREADS USING COMPUTATIONAL FLUID DYNAMICS (CFD)

Since cfd projects are are an extrapolation of advanced mathematics an are which we Indians are traditionally flavoured in , if the students are guided in the right way , cfd boom could be the next IT boom for india

Computational Fluid Dynamics (CFD) is a special kind of numerical analysis done to understand the patterns of fluid flow in various fluid machinery systems such as pumps, the flow over airplanes and their parts, and the flow inside devices such as internal combustion engines**.**

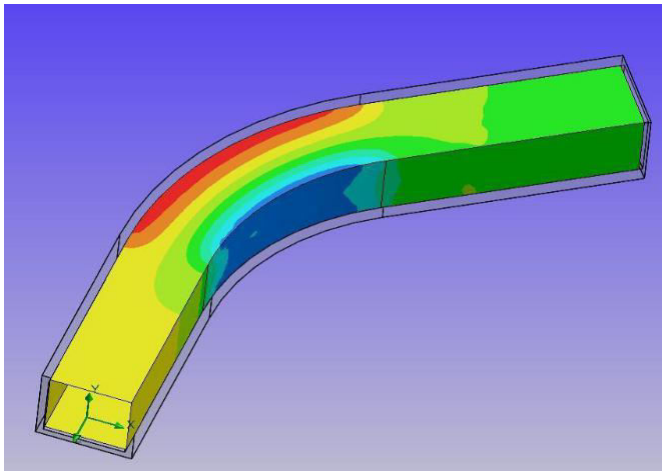


Fig 6.1 C in curve duct FD analysis of air flow

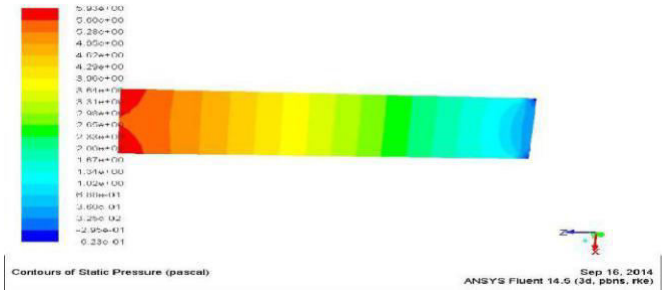


Fig 6.2 CFD analysis of heat in the duct

While numerical analysis is a broad umbrella term that describes various kinds of system modeling and analysis approaches in engineering, CFD as described above has more specific uses. Contrast this with functional analysis, where we model system behaviour using equations and then solve systems of equations using algebra, calculus, differential equations and other symbolic methods. Numerical analysis generally consists of matrix algebra operations applied in numerical optimization and for solving systems of equations to get approximate solutions to them.CFD is similar to the Finite Element Analysis (FEA), which is a more general purpose method for numerical analysis to solve structural, thermal, spectral and flux propagation problems by simplifying them. The fundamental simplification that approaches like FEA and CFD bring, is to discretise a continuum where phenomena (such as fluid flow) are being studied. Like FEA, CFD also involves extensive use of matrix algebra and solutions to systems of equations, but models the behaviour of fluids in the processA CFD based numerical simulation of flow velocity of hydrocyclone was conducted with different structural and operational parameters to investigate its distribution characteristics and influencing mechanism. The results show there exist several unsymmetrical envelopes of equal vertical velocities in both upward inner flows and downward outer flows in the hydrocyclone, and the cone angle and apex diameter have remarkable influence on the vertical location of the cone bottom of the envelope of zero vertical velocity. It is also found that the tangential velocity isolines exist in the horizontal planes located

in the effective separation region of hydrocyclone. The increase of feed pressure has almost no effect on the distribution characteristics of both vertical velocity and tangential velocity in hydrocyclone

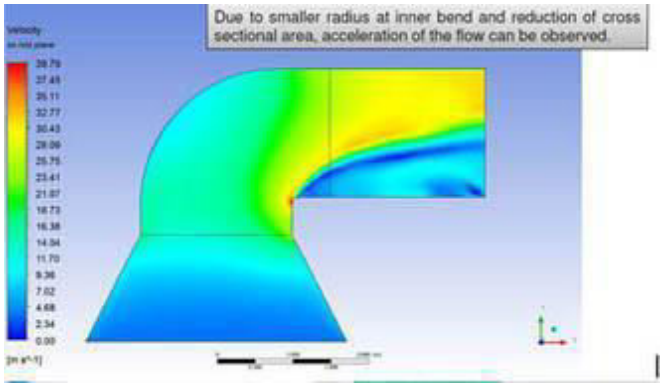


Figure 6.3 cfd analysis of air in the diffuser

REFERENCES

[1]Madhulika singh, sham alam on "CFD Approach to Design and Optimization of Air, Flue Gas Ducting System" Research Technology Management, (Sept- Oct), 25-32

[2] A.Aravind kumar, on "Analyzing the gas flow in CFD for various ducts" UNE

100713ASHRAE Standard 170, and pre-standard EN 16244, -2.

[3] S.Srikanth, Dr. Benny paul "Guidelines for the Fluid Dynamics Design of power plant Ducts," Electric Power Research Institute California, 1988.

[4]. Bezuidenhout, J.J. Yang, Y. and Eksteen , J.J. "CFD Analysis of a waste-heat boiler associated with flash smelting of base metal sulphides".

[5] Gonzalo Sánchez et. Al(2019) Design parameters of HVAC installations in high-performance hospital operating theatres were evaluated according to UNE 100713ASHRAE Standard 170, and pre-standard EN 16244, -2.

[6] Menke, Michael M. (1994), Improving CFD Decisions and Execution, Research Technology Management, (Sept- Oct), 25-32.

[7] S. Jayanti "Techniques for flow passage optimization in air and flue gas ducting in boilers", BHEL National conference on CFD Application ,Hyderabad, 17-18 Nov,2006.

[8] Bezuidenhout, J.J. Yang, Y. and Eksteen , J.J. "CFD Analysis of a waste-heat boiler associated with flash smelting of base metal sulphides".

[9] Hajee Mohammed, M. and Wilson,V.H. "Analysis of Boiler Super Heater Tubes from High Flue Gas Temperature"

[10] Blake Robert Eisner "CFD simulation with fine and a recenty developed turbulence model, the scale adaptive solution

11 Specifications, Boston, MA. Consortium for Energy Efficiency.

[12] Davis, R., P.Francisco,M. Kennedy, D. Baylon, B.Manclark. 2002. Enhanced Operations & Maintenance Procedures for Small Packaged Roof top HVAC Systems: Protocol Development, Field Review, and Measure Assessment. Submitted to Eugene Water and Electric Board. Seattle, WA .Ecotope ,Inc.

[13] Eley, 2002. Measure Analysis and Life-Cycle Cost. 2005California Building Energy Efficiency Standards, Part 1.Preparedfor the California Energy Commission. San Francisco, CA. Eley Associates, Inc. P400-02-011.