

# **Numerical Analysis of Air Flow Characteristics in Heating Ventilating and Air Conditioning (HVAC) Duct**

Aravinda D<sup>1</sup>, Karthik T<sup>2</sup>, Ningappa D Kunabeva<sup>3</sup>

Assistant Professor, Department of Mechanical Engineering, Dr. Ambedkar Institute of Technology, Bangalore, India<sup>1</sup>

Assistant Professor, Department of Mechanical Engineering, School of Engineering, Presidency University  
Bangalore, India<sup>2</sup>

Assistant Professor, Department of Mechanical Engineering, Rajarajeshwari College of Engineering, Bangalore, India<sup>3</sup>

**ABSTRACT:** HVAC system is widely used in residential, hotels, malls, hospital to provide thermal comfort and to maintain good indoor air quality. Design, optimization and numerical analysis of air flow characteristics in HVAC duct based on the optimization approach for the effective and efficient evaluation of HVAC duct layout, alternatives at the design stage to get the optimal duct layout results. It is concluded from the literature review that the numerical modeling based, duct layout modeling and simulation approach provides a conducive environment for evaluating the HVAC Duct system layout. Various aspects of HVAC duct system optimization are analyzed in several studies and the optimization of HVAC duct system layout is analyzed, the systematic method is developed for evaluating HVAC duct system layout for specific building load demand and climate conditions. Performance analysis of rectangular cross sectioned divergence long radius bend, V- shape cut of HVAC DUCT layout has been carried out. The detailed study of the architectural drawing has included and cooling load calculations is also carried out by Hourly Analysis Program (HAP) software and cross verified with using standards codes of ASHRAE handbook of fundamentals. The duct layout is to be designed for auditorium then it analyzed in the CFD Tool for optimization the duct. The CFD tool ANSYS Fluent 15 is used for performance analysis of the HVAC Duct layout. During analysis the effect of pressure drop, turbulence, recirculation zone near the bend, total pressure velocity streamline distribution flow efficiency of the duct layout is investigated. This helpful for install appropriate duct layout for buildings. To conclude, the presented systematic based optimization models significantly help to install the duct layout for the buildings.

**KEYWORDS:** HVAC, Duct, ASHRAE, CFD, HAP

## **I. INTRODUCTION**

Optimization is concerned with the minimization or maximization of the objective function depending on what someone is looking for. However, normally optimization problems are defined as minimization 'and if a criterion is subjected to maximization then the negation of the objective function is minimized. In optimization problems, the building simulation programs are increasingly being used to evaluate the objective function. The current study is based on the minimization of the objective function in which the overall HVAC duct system model is optimization by numerical algorithm. Such an interaction provides automatic selection of optimal configuration. However, it requires suitable design optimization along with optimization variables in terms of system design and configuration parameters.

The aspects of HVAC duct system configuration and sizing are strongly related with each other. The individual component models and the system of the developed overall HVAC duct system model must be accurately sized to fulfil the specified buildings load demands. However, the key difference between the configuration and the sizing issues is that the configuration of HVAC system is not predefined. Therefore, the aspect of system configuration is mainly focused on the decisions related to the type, the number, and the arrangement of the system components. Thus, after the selection of a suitable duct configuration, the sizing variables associated the particular configuration are identified. Meanwhile, the optimal configuration design is evaluated by the overall system performance that means

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 12, December 2017

solving the sizing problem is also required for a certain system configuration. As a result, in the current study, the applied approach for the optimization of HVAC duct system configurations can vary the both system configuration and system design parameters.

Heating Ventilation and Air-Conditioning (HVAC) systems control the indoor environment throughout the year to ensure comfortable conditions in homes, offices, and commercial facilities. Beside the fact that HVAC systems are making human life healthier and more productive but various products could also be produced faster, better, and more economically in an appropriately controlled environment. Almost each residential, institutional, commercial, and industrial building has a year-round controlled environment in the developed countries of the world.

The awareness of the importance of living place and indoor air quality has increased for both health and comfort. The growing requirements for closely controlled working environments in laboratories, hospitals, and industrial facilities have rapidly increased overall energy demands. Therefore, the HVAC systems designers are challenged as never before to come up with the most energy efficient systems. One way of addressing the challenge is by optimizing conventional setups and introducing efficient innovative systems utilizing renewable energy resources.

The heated or cooled air is delivered to the various rooms and spaces within the structure in distribution systems based on the forced air principle of delivery utilize systems of ducts. These air ducts are generally rectangular or round pipes made from a variety of different materials. When these ducts are accurately sized and the duct system currently designed the air will be delivered to the rooms and spaces with a minimum of resistance, the result being a more efficient operation with reduced operating costs. The purpose is to suggest methods for sizing ducts and designing an efficient duct system.

The duct has to be so sized that it is accommodated with in the available space. Like any other fluid passing through a pipe, air in passing through a duct suffers a pressure drop due to friction. Larger the quantity of air passing through a given cross sectional area of the duct (i.e. larger air velocity) greater will be the frictional loss and pressure drop.

The fan selected has to deliver the required quantity of air overcoming the resistance offered by the various components in the air distribution systems such as cooling coil/air-washer filters, supply –and return air outlets, dampers etc. plus the resistance offered by the duct system i.e. the fan has to work against a head. As this head increases, the fan will need higher power to deliver the required quantity of air against the system head or pressure. Further as velocity increases noise level due to the air in motion also increases. Thus, on these two count the velocity of air in the duct has to be kept at a reasonably low level with lower air velocities the size of the duct increases. At a given rate of flow of air the duct layout system should be balanced between the initial cost and the operating cost. The initial cost depends on the size of the duct, with a smaller duct layout the air velocity will increase and so pressure loss due to friction will also increase. The fan uses more power to overcome the head, thereby increasing the operating cost instead of evaluating the balance of initial cost and operating cost in every case designers depend upon past experience and generally accepted recommended air velocities in the ducts are employed.

Ducts are manufactured from a variety of different materials. The materials selected will depend on the use for which it is intended. It is every important that the duct materials be taken into consideration when designing a duct system because not every material is suitable for all conditions in which ducts are used.

It is possible to purchase ducts manufactured from the following materials.

## a. Plain Steel and Galvanized Sheet Steel

Plain steel and galvanized sheet steel ducts are available in thicknesses ranging from 0.0163 to 0.1419 inch. Ducts manufactured from this material are preferred for use in warm air gravity and forced circulation warm air heating systems.

## b. Aluminum

Aluminum ducts are available in thickness ranging from 0.012 -0.064 inches and are used in the same types of heating systems as steel ducts. Although aluminium duct is lighter than steel ones, they generally cost more. Aluminium ducts are frequently used in duct systems located on the outside duct work.

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 12, December 2017

Mukesh Didwania.et.al., [1]in this work the detailed review of commercial finite volume CFD computer code ANSYS 12.0 Fluent analysis flow over a 90° bend of Duct are explained. An effort is made to represent and summarize sophisticated Reynolds stress model are shown to better capture the anisotropy behaviour of the flow in contrast to the standard k-ε model that assumes isotropy in its original model formulation and also k-w SST model use to comparison.Santosh Kansal.et.al., [2] explains 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 CFD TOOLANSYSFluent 14.5 is used for performance analysis of the elliptical duct. During analysis cross section is varied as circular, rectangle and elliptical, and its effect on static pressure and total pressure of the duct.Vladimir Zmrhal.et.al., [3]in this work, the knowledge of the local loss coefficients is important for the accurate calculation of ventilation duct pressure loss. The pressure loss of ventilation duct is very forecasted, what causes the wrong design of the ventilating fan. A large number of local loss coefficients exist, but the published data are different. The local loss coefficient can be estimated experimentally by the measurement on the real model, or with using of numerical simulation. The paper presents the using of CFDsimulation for local loss coefficients of ventilationduct fittings (elbows and bends). The paper presents the advantages of the CFD simulation using for analysis of local pressure loss coefficient of the duct fittings.AA Rezwan.et.al., [4] this paper describes the air flow pattern inside a HVAC Duct used in commercial building. The duct chosen for this investigation are rectangular in shape with varying hydraulic diameter. A commercial building HVAC duct has been numerically modelled and simulated for characterizing the air flow pattern. It has been noticed that the pressure inside the duct has been gradually decreased along the duct length. Results can be modified for more complex system of the HVAC duct and can be helped in designing duct for any residential and commercial HVAC system.AnaPaulaP.et.al., [5] this review article describes the detail of Pressure loss or drop in ductworks is an important term to be considered in design of engineering systems such as power-plants, refineries, HVAC systems to reduce energy costs. Duct fittings are significant sources of pressure loss in fluid distribution systems. At the present work, a conventional 90° round elbow under turbulent incompressible airflow is studied. Mass, momentum, and k-ε turbulence model equations are solved employing the finite volume method. The SIMPLE algorithm is used for the pressure-velocity coupling. In order to validate the numerical tool, the elbow pressure loss coefficient is determined using the same conditions to compare with ASHRAE database.Asam A. Hirani.et.al., [6] in this paper, an author gives the details of a Plumbing system use pipe fittings to connect straight pipe for regulating or measuring fluid flow. Y shape fitting is one of the important components in the plumbing system. From the CFD analysis it was analyzed that the secondary flow caused by a fully developed straight pipe profile at the start of the bend increases to a maximum at a bend angle of about 60°. D. Ghate.et.al., [7] present paper gives the information about the experimental work done on the S-shaped ducts. The calculation of the distortion at the exit of S-shaped duct using Navier-Stokes solvers with turbulence models. Study for S-shaped duct Optimization with circular entry and exit sections using CFD solver. Raviraj Gurav.et.al., [8] have performed the use of air-conditioning for comfort purpose was considered a luxurious but now-a-day, it has been a necessity in extreme climatic conditions, such as extreme cold and hot in western countries. Window air conditioners are preferred for office rooms while large centralized units are installed for conditioning the auditorium, hospitals etc. Equal friction method is adopted to design the duct which is simple as compared with the other duct design methods.Dr.Neihad al khalidy[9]this paper gives a brief knowledge of design modification to a number of industrial ducts using Computational Fluid Dynamics (CFD) analysis considering all flow features relating to the duct system efficiency.

The above researchers have worked on the S-shaped, elbow- shaped duct layout depending upon the applications and no one researcher cannot do the cooling load calculation cross verification manually. From their study it is found that the duct layout of building has complex design and study reveals on the different software's such as Catia, CFD (ANSYS Fluent 15), Auto CAD, HAP. The researchers have concentrated only on the flow of outlets, neglecting the turbulent behaviour and eddies formation inside the duct layout and it increases the pressure drop.it is desired to carry out preliminary design calculation including CFD optimization for maximum comfort and Efficiency. And obtain Total pressure, velocity and Stream line distribution for the HVAC Duct divergence, consecutive bend, V-cut configuration So as to check the flow Efficiency and then Remodelling the HVAC Duct and suggest better duct layout in order to minimize total pressure loss and obtain a better velocity profile.

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 12, December 2017

## II. OBJECTIVES AND COMPUTATIONAL METHODOLOGY

### 2.1. OBJECTIVES

1. Work involves simulation of air flow characteristics in HVAC duct layout for different design alternatives.
2. To determine cooling load calculations by HAP software and cross verified by manually using ASHRAE Data base.
3. Design optimization of Duct layout on critical areas through CFD.

### 2.2. CHALLENGES

1. Remodelling CAD model of the Duct layout to perform pressure drop analysis.
2. Utilize applicable governing equations
3. Identify accurate turbulence model to simulate the air flow.
4. Develop Duct design alternatives to reduce the pressure drop.

### 2.3. METHODOLOGY

1. Detail study of Architectural drawing of proposed building (Auditorium) and to measure the dimensions of the Auditorium.
2. Design Input considerations or cooling load calculations using ASHRAE database
3. Using cooling load (CFM) the duct width and length is designed by Duct calculator and height of duct is calculated by Architectural drawing
4. The 2DCAD model DUCT is studied and preceded for modeling of 3D geometry in CATIA. The 3D model is imported in to ANSYS FLUENT CFD for pre-processing and solved in ANSYS FLUENT environment.
5. The boundary conditions are given as mass flow rate for the duct inlet, wall and exit as mass flow rate as per the analytical data provided.
6. The model is meshed with tetrahedral elements and the K-epsilon turbulence modeling is used.
7. The convergence value of  $10e^{-4}$  is given for the governing variables.
8. Check for flow profile uniformity and total pressure distribution throughout the duct.

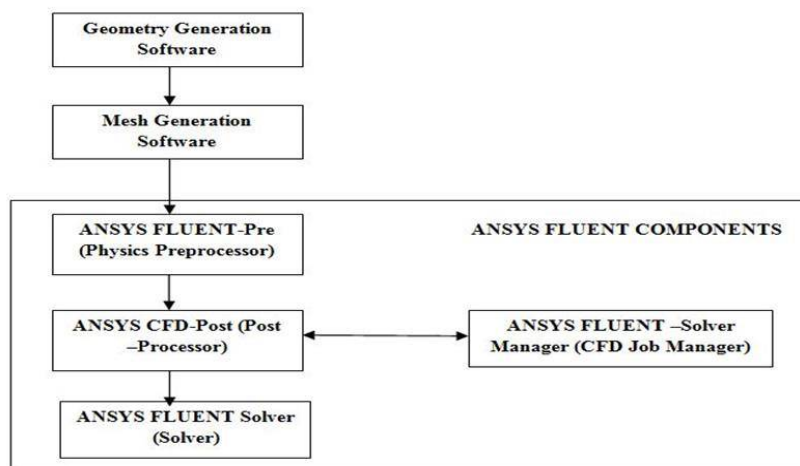


Fig.1:Flow Chart of ANSYS Fluent Components

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 12, December 2017

## III. DESIGN INPUT CONSIDERATIONS (COOLING LOAD MANUAL CALCULATION)

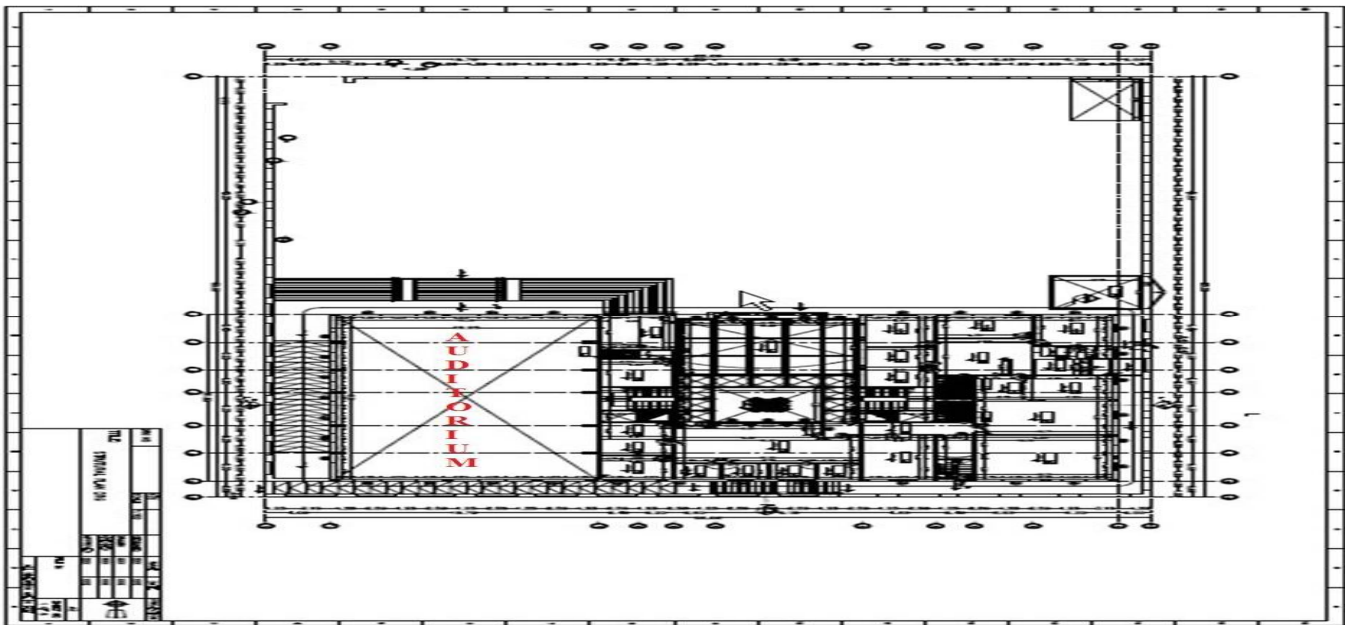


Fig.2: Detailed Study of Architectural Plans and to measure dimensions of the surface through which there will be an external Heat Gain, in an “Auditorium”

Table1: Details of Indoor and Outdoor Design Conditions Table 2: Envelope Loads for Space: “AUDITORIUM” Zone

Location	Dhahran Saudi Arabia
Latitude	26 <sup>0</sup> F
Longitude	50 <sup>0</sup> F
Elevation	80 ft
Dry Bulb Temperature	111 <sup>0</sup> F
Wet Bulb Temperature	85 <sup>0</sup> F
Daily Range	32 <sup>0</sup> F
Outdoor Design Temperature	128 <sup>0</sup> F
Indoor Design Temperature	78 <sup>0</sup> F

	Area Ft <sup>2</sup>	U-Value BTU/(h r-ft <sup>2</sup> -°f)	Shade Coeff	COOLING Trans BTU/hr	COOLING Solar BTU/hr
NW EXPOSURE					
WALL	1850	0.363	-	24176	-
WINDOW 2	424	0.540	0.586	12822	35531
DOOR	80	0.347	-	1944	-
NE EXPOSURE					
WALL	315	0.363	-	3202	-
DOOR	119	0.347	-	331	-
SW EXPOSURE					
WALL	1089	0.363	-	24905	-
WINDOW 1	227	0.540	0.459	38245	13857



## International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 12, December 2017

**Table 3: Design Cooling Load**

ZONE LOADS	COOLING DATA COOLING DB/WB 111 <sup>0</sup> F /85 <sup>0</sup> F		
	DETAILS	SENSIBLE(BTU/hr)	Latent(BTU/hr)
Window & Skylight Solar Loads	651 Ft <sup>2</sup>	49388	-
Wall Transmission	3254 Ft <sup>2</sup>	52283	-
Roof Transmission	0 ft <sup>2</sup>	0	-
Window Transmission	651 ft <sup>2</sup>	51067	-
Skylight Transmission	0ft <sup>2</sup>	0	-
Door Loads	199ft <sup>2</sup>	2275	-
Floor Transmission	5439 ft <sup>2</sup>	0	-
Partitions	1232ft <sup>2</sup>	22176	-
Ceiling	5616 ft <sup>2</sup>	60653	-
Overhead Lighting	3400 W	13913	-
Task Lighting	600W	1729	-
Electric Equipment	7840 W	25822	-
People	300	75000	60000
Infiltration	-	60000	-38000
Miscellaneous	-	0	0
<b>&gt;&gt;Total Zone Loads</b>	-	<b>414306</b>	<b>22000</b>
Zone Conditioning	-	414306	22000
Plenum Wall Load	30%	20743	-
Plenum Roof Load	70%	0	-
Plenum Lighting Load	30%	13718	-
Return Fan Load	22469 CFM	0	-
Ventilation Load	6240 CFM	0	0
Supply Fan Load	18695 CFM	2772	-
Space Fan Coil Fans	-	0	-
Duct Heat Gain /Loss	3%	9189	-
<b>&gt;&gt;Total System Loads</b>	-	<b>460728</b>	<b>22000</b>
Central Cooling Coil	-	460728	22000
Central Heating Coil	-	0	-
<b>&gt;&gt; Total Conditioning</b>	-	<b>460728</b>	<b>22000</b>

Conversion of Total Heat Load (Btu/hr) To CFM Using the Formula is as Follows

$$= \frac{\text{Total Heat load}}{1.08 \times (DB - WB)} = \frac{460728}{1.08 \times (111 - 85)}$$

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 12, December 2017

Cooling load = 16408 CFM

### 3.1. HOURLY ANALYSIS PROGRAM (HAP CALCULATIONS)

Hourly Analysis Program (HAP) is a computer tool which assists engineers in designing HVAC systems for commercial buildings. HAP is two tools. First it is a tool for estimating loads and designing systems. Second, it is a tool for simulating building energy use and calculating energy costs. HAP uses the ASHRAE-endorsed transfer function method for load calculations and detailed 8,760 hour by hour simulation techniques for the energy analysis. This program is released as two separate, but similar products. The “HAP System Design Load” program provides system design and load estimating features. The full “HAP” program provides the same system design capabilities plus energy analysis features.

<b>Air System Information</b>			
Air System Name .....	1	Number of zones .....	1
Equipment Class .....	SPLT AHU	Floor Area .....	1700.0 ft <sup>2</sup>
Air System Type .....	SZCAV	Location .....	Dhahran, Saudi Arabia
<b>Sizing Calculation Information</b>			
Zone and Space Sizing Method:			
Zone CFM .....	Sum of space airflow rates	Calculation Months .....	Jan to Dec
Space CFM .....	Individual peak space loads	Sizing Data .....	Calculated
<b>Central Cooling Coil Sizing Data</b>			
Total coil load .....	54.8 Tons	Load occurs at .....	Aug 1300
Total coil load .....	657.7 MBH	OA DB / WB .....	108.4 / 70.2 °F
Sensible coil load .....	625.6 MBH	Entering DB / WB .....	98.0 / 70.5 °F
Coil CFM at Aug 1300 .....	14945 CFM	Leaving DB / WB .....	59.2 / 56.7 °F
Max block CFM .....	14945 CFM	Coil ADP .....	54.9 °F
Sum of peak zone CFM .....	14945 CFM	Bypass Factor .....	0.100
Sensible heat ratio .....	0.951	Resulting RH .....	52 %
ft <sup>2</sup> /Ton .....	31.0	Design supply temp. ....	58.0 °F
BTU/(hr-ft <sup>2</sup> ) .....	386.9	Zone T-stat Check .....	1 of 1 OK
Water flow @ 10.0 °F rise .....	N/A	Max zone temperature deviation .....	0.0 °F
<b>Central Heating Coil Sizing Data</b>			
No central heating coil loads occurred during this calculation.			
<b>Preheat Coil Sizing Data</b>			
Max coil load .....	487.5 MBH	Load occurs at .....	Des Htg
Coil CFM at Des Htg .....	14945 CFM	Ent. DB / Lvg DB .....	67.7 / 98.0 °F
Max coil CFM .....	14945 CFM		
Water flow @ 20.0 °F drop .....	N/A		
<b>Supply Fan Sizing Data</b>			
Actual max CFM .....	14945 CFM	Fan motor BHP .....	0.00 BHP
Standard CFM .....	14915 CFM	Fan motor kW .....	0.00 kW
Actual max CFM/ft <sup>2</sup> .....	8.79 CFM/ft <sup>2</sup>	Fan static .....	0.00 in wg
<b>Outdoor Ventilation Air Data</b>			
Design airflow CFM .....	1102 CFM	CFM/person .....	11.02 CFM/person
CFM/ft <sup>2</sup> .....	0.65 CFM/ft <sup>2</sup>		

The above calculated cooling load is very high and manufacturing of air handling unit is very expensive because of this reason the above calculated cooling load is divided by 2 and installing the two duct layout for proposed auditorium.

Therefore

$$\text{Cooling load} = \frac{16408}{2}$$

$$= 8208 \text{CFM}$$

$$= 3.775 \text{m}^3/\text{sec}$$

$$\text{Mass flow rate} = \rho AV$$





## International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 12, December 2017

calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. Even with simplified equations and high-speed supercomputers, only approximate solutions can be achieved in many cases. Ongoing research, however, may yield computer software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial validation of such software is often performed using a wind tunnel with the final validation coming in flight test.

k-ε turbulence model has been used for the present analysis. k represents turbulence kinetic energy and is defined as the variance of the fluctuations in velocity. It has dimensions of (L<sup>2</sup>T<sup>-2</sup>); for example, m<sup>2</sup>/s<sup>2</sup>. ε represents turbulence eddy dissipation, and has dimensions of k per unit time (L<sup>2</sup>T<sup>-3</sup>). The k-ε model introduces two new variables into the system of equations. The continuity equation is then,

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_j) = 0$$

Momentum Equation is,

$$\frac{\partial \rho U_i}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_i U_j) = - \frac{\partial p'}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu_{eff} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \right] + S_M$$

Where **S<sub>M</sub>** represents sum of body forces, **μ<sub>eff</sub>** represents turbulence effective viscosity and **p** is the modified pressure. The k-ε model, like the zero equation model, is based on the eddy viscosity concept, so that:

$$\mu_{eff} = \mu + \mu_t$$

Where **μ<sub>t</sub>** represents turbulence viscosity. The k-ε model assumes that the turbulence viscosity is linked to the turbulence kinetic energy and dissipation via the relation:

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon}$$

Where **C<sub>μ</sub>** is a constant. The values of **k** and **ε** come directly from the differential transport models for the turbulence kinetic energy and turbulence dissipation rate.

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_j k) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \rho \varepsilon + P_{kb}$$

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_j \varepsilon) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{\varepsilon}{k} (C_{\varepsilon 1} P_k - C_{\varepsilon 2} \rho \varepsilon + C_{\varepsilon 1} P_{\varepsilon b})$$

Where **C<sub>ε1</sub>**, **C<sub>ε2</sub>**, **σ<sub>k</sub>** and **σ<sub>ε</sub>** are constants. **P<sub>k</sub>** and **P<sub>εb</sub>** represent the influence of the buoyancy forces, which are described below. **P<sub>k</sub>** is the turbulence production due to viscous forces, which is modeled using

$$P_k = \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j} - \frac{2}{3} \frac{\partial U_k}{\partial x_k} \left( 3 \mu_t \frac{\partial U_k}{\partial x_k} + \rho k \right)$$

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 12, December 2017

For incompressible flow,  $\frac{\partial U_k}{\partial x_k}$  is small and the second term on the right side of Equation does not taken significantly to the production. For compressible flow,  $\frac{\partial U_k}{\partial x_k}$  is only large in regions with high velocity divergence, such as at shocks.

### Advantages k-ε Turbulence Model

- Relatively simple to implement.
- Leads to stable calculations that converge relatively easily.
- Reasonable predictions for many flows.

### 4.1. MODEL DESCRIPTION

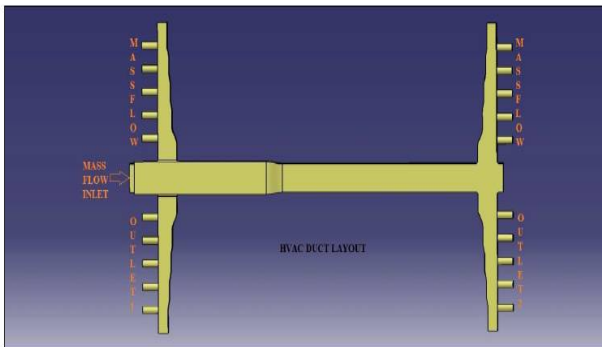


Fig.4: HVAC Duct Layout

CATIA Software is interactive application software used to generate three dimensional models with the help of computer, here in this project we are using mechanical surface design and sketcher feature to create HVAC Duct model as per the 2D details and the 3D model created in CATIA.

### 4.2. MESH GENERATION

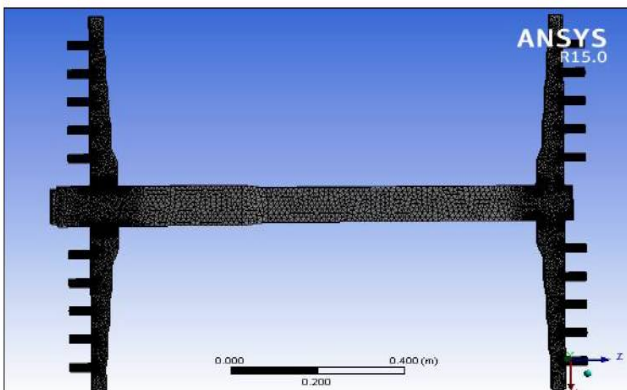


Fig.5: Meshing of HVAC Duct Layout

### Grid Independence Study

A grid independence study on unstructured meshes for the HVAC Duct was carried. The meshing was carried out by giving different size of the mesh element like 6mm, 5mm, 4mm, 3mm and 2mm and 1mm running the simulations for the HVAC Duct with different mesh numbers, comparing the results and getting a grid independent mesh size. It was found that the solution no longer changes for a size of the mesh element of 1 mm which gives a total number of elements as 8,81,250 and therefore this was kept constant for simulations.

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 12, December 2017

## 4.2.1. MESH DESCRIPTION

**Table 4: Mesh Orthogonality**

Statistics	
Nodes	181096
Elements	881250
Mesh Metric	Orthogonal Quality
Min	0.215749866332858
Max	0.995118013631006
Average	0.842355437283612
Standard Deviation	0.093224713339825

**Table 5: Skewness**

Statistics	
Nodes	181096
Elements	881250
Mesh Metric	Skewness
Min	6.61617201373366E-04
Max	0.93371117456172
Average	0.262435472800606
Standard Deviation	0.141080012853283

**Table 6: Aspect Ratio**

Nodes	181096
Elements	881250
Mesh Metric	Aspect Ratio
Min	1.1701
Max	11.209
Average	1.92404583239713
Standard Deviation	0.533061665828056

Value of Skewness	Cell Quality
1	Degenerate
0.9 — <1	Bad
0.75 — 0.9	Poor
0.5 — 0.75	Fair
0.25 — 0.5	Good
0 — 0.25	Excellent
0	Equilateral

## 4.3. BOUNDARY CONDITIONS

To solve the equation system, we also need boundary conditions. The typical boundary conditions in CFD are No-slip boundary condition axisymmetric boundary condition, Inlet, outlet boundary condition and Periodic boundary condition, interface boundary condition depending on the problem use the boundary condition. ANSYS Fluent consists 10 classes of boundary zone, types for the specification of flow inlets and exits: velocity inlet, pressure inlet, mass flow inlet, pressure outlet, pressure far-field, outflow, inlet vent, intake fan, outlet vent, and exhaust fan.

**Table7: Boundary conditions**

Fluid	Air
Time	Steady State Analysis
Density	1.2466 kg/m <sup>3</sup>
Mass Flow Rate	4.705 kg/sec
Design Pressure	Atm pressure
Design Temperature	300k
Viscous Regime	Turbulent
Turbulence model	Standard K-Epsilon
Wall	No-slip wall

## 4.4. SIMULATIONS

The simulations are performed in the FLUENT solver and each simulation run for 2500 iterations. Convergence is checked from the residuals to make sure that the solution is stabilized.

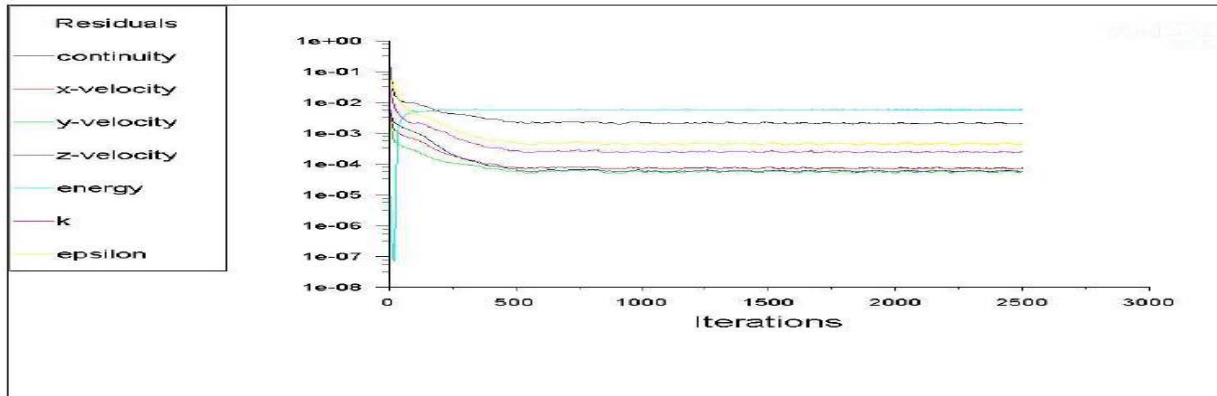


Fig.6: Convergence Plot

## V. RESULTS AND DISCUSSION

Analysis set up of all cases is carried out in FLUENT Solver. Result analysis is done using ANSYS Post processor. K-epsilon turbulence model is used for Analysis. Velocity and pressure plots are plotted for all the cases of study. Analysis results of the Base model and the Optimized designs are shown in this section, and comparison of the results of the base case and the different designs for optimization is presented.

### 5.1. BASE MODEL

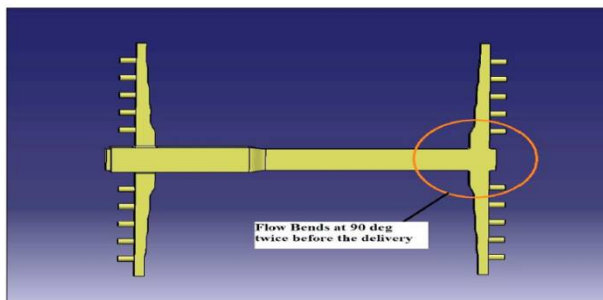


Fig.7: Base Model

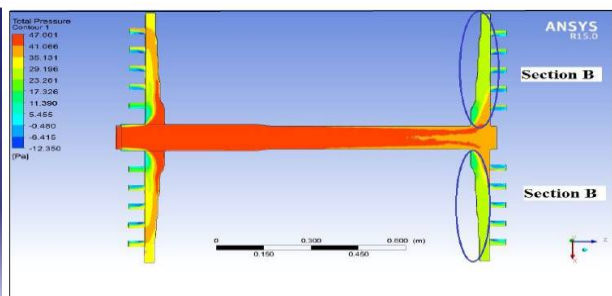


Fig.8: Total Pressure Distribution

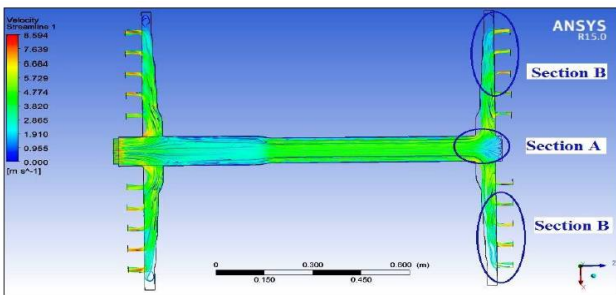


Fig.9: Stream Lines Plot

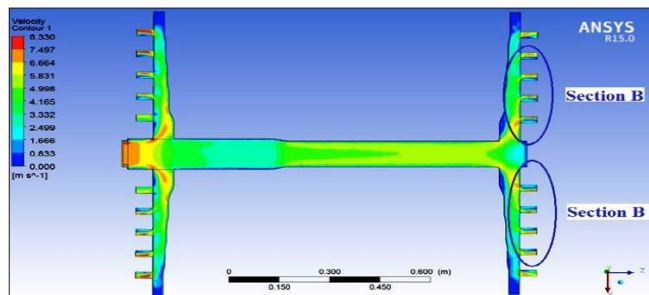


Fig.10: Velocity Plot

Due to the loss of mass flow, loss in the flow velocity and total pressure at section A of the duct can be seen. Huge pressure loss in section B as expected. The inlet total pressure of 45pa is reduced to 15pa towards the ends of

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 12, December 2017

section B as shown in Fig.8. The analysis of original design show cased circulation and corresponding vortex generation in the duct at section A and section B. so further modification were made in order to overcome these vortices. Consecutive bending will reduce the flow efficiency, there by leading to vortex generation which needs to be avoided s shown in Fig.9. Velocity at outlets at section B is in the range of 4m/s - 7m/s which is on higher end, this need to be reduced. Flow is not efficiently controlled, hence other alternatives to be consideredas shown in Fig.10.

## 5.2. OPTIMIZATION OF CRITICAL AREAS TO GIVE DIVERGENCE, SHARP CORNER AND V- SHAPE CUT

Optimization of critical areas to give divergence and V-shape cut is generated in the CATIA V5 R20 designing software. And meshed in ANSYSMesher and analyzed using the fluent solver. The results of total pressure distribution, velocity plot and stream lines plot are shown in the figure below:

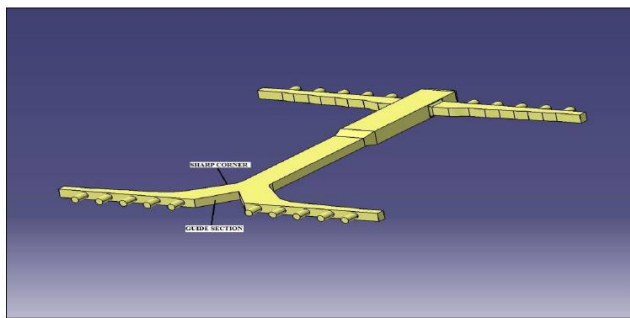


Fig.11: Criticalareastogive divergence, sharpcorner and V- shape cut

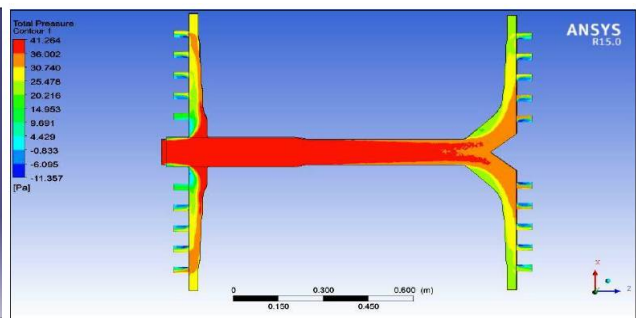


Fig.12: Total Pressure Distribution

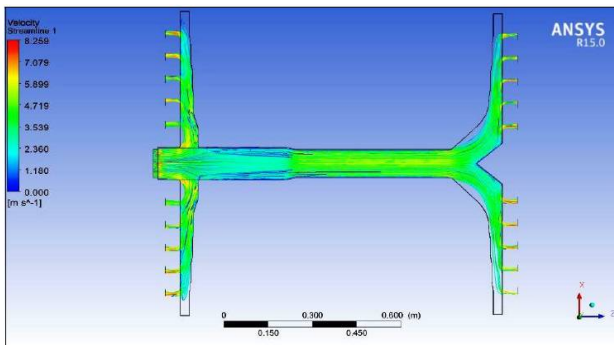


Fig.13: Stream Lines Plot

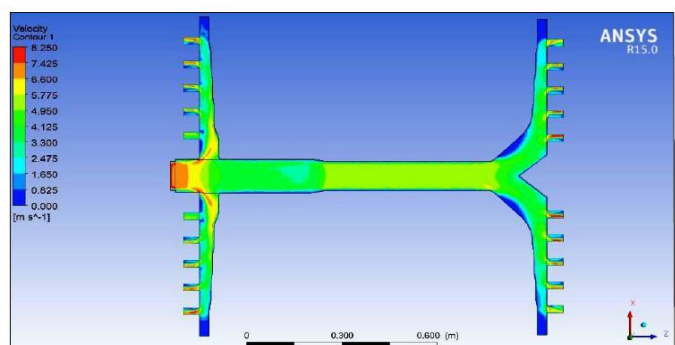


Fig.14: Velocity Plot

By giving the divergence and V-cut at the section A is to reduce the loss of mass flow, loss in the flow velocity and total pressure at section C of the duct can be minimized as compare with base model. Minimum pressure loss in section B as expected as shown in Fig.12. So, we have to remodeling the duct system.By giving the Divergence and V-cut, the analysis of original design show cased circulation and corresponding vortex generation in the duct at section A and section B is minimized. So further modification was made in order to overcome these vortices. Consecutive bending will reduce the flow efficiency, there by leading to vortex generation which needs to be avoided as shown in Fig.13. Velocity at outlets at section B is in the range of 4m/s - 7m/s which is on higher end, this need to be reduced. Flow is not efficiently controlled, hence other alternatives to be considered as shown in Fig.14.



# International Journal of Innovative Research in Science, Engineering and Technology

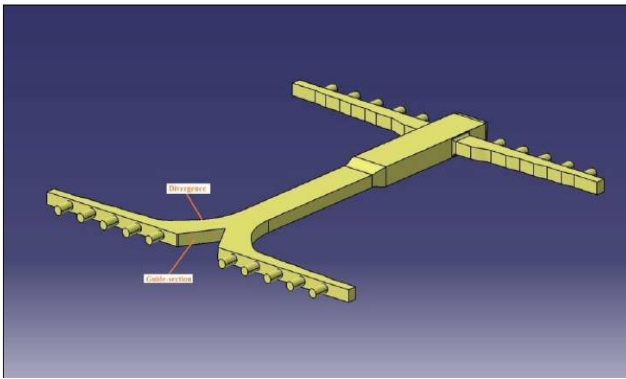
(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

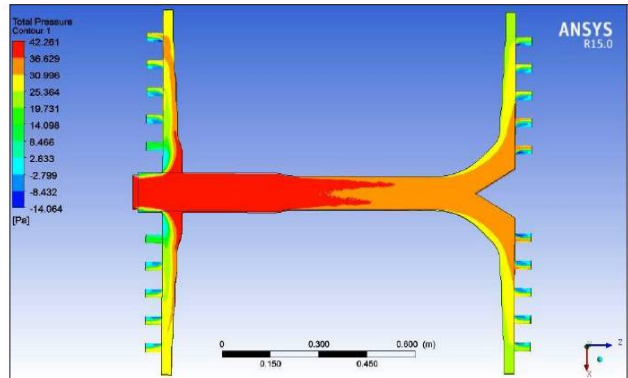
Vol. 6, Issue 12, December 2017

### 5.3. OPTIMIZATION OF CRITICAL AREAS TO GIVE DIVERGENCE, (V-SHAPE CUT) AND CONSECUTIVE BENDING OF THE CORNERS

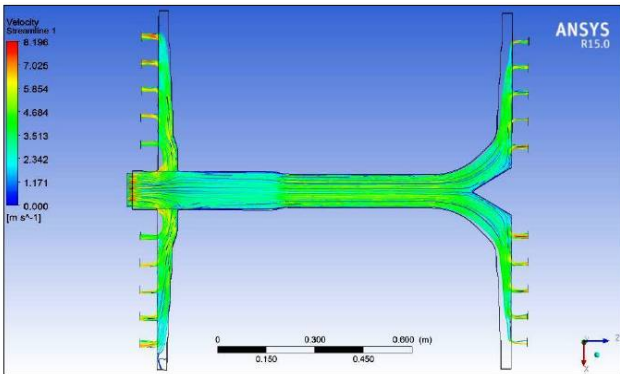
In the Case No 3 Optimization of Critical Areas to Give Divergence (V-Shape Cut) And Long Radius Bend of the Corners is generated in the CATIA V5R20 Designing Software. And Meshed in ANSYS Mesher And Analyzed Using the Fluent Solver. The Results of Total Pressure Distribution, Velocity Plot and Stream Lines Plot Are Shown in the Figure Below.



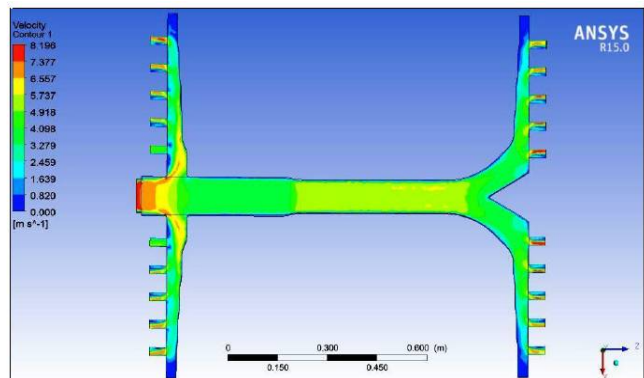
**Fig.15: Critical areas to give divergence and consecutive bending of the corners**



**Fig.16: Total Pressure Plot**



**Fig.17: Stream Lines Plot**



**Fig.18: Velocity Plot**

The total pressure remains uniform throughout, which indicates a minimum energy loss. The loss of total pressure occurred in Duct system is eliminated as shown in Fig.16. The analysis of remodeling HVAC duct is to eliminate the cased circulation and corresponding vortex generation in the duct at section A and section Bas shown in Fig.17. Velocity at outlets at section B is is uniform throughout, which indicates Flow is efficiently controlled as shown in Fig.18

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

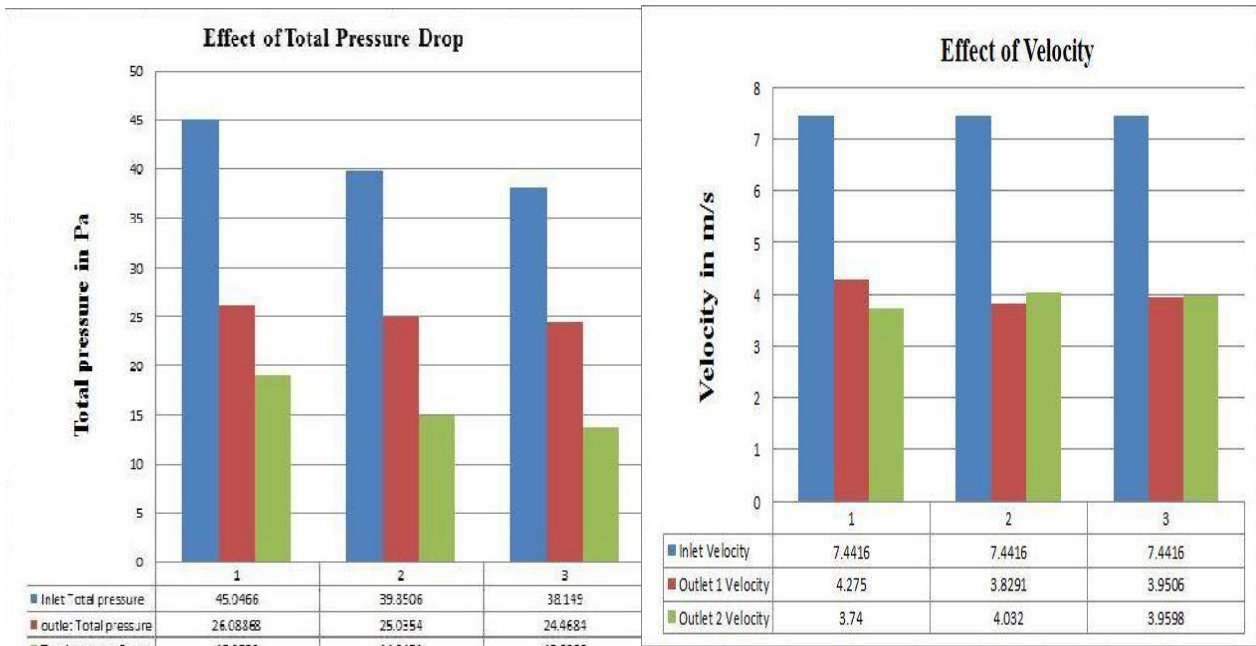
Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 12, December 2017

## 5.4. EFFECT OF TOTAL PRESSURE AND VELOCITY IN THE DUCT LAYOUT

Sl No	Model	Inlet Total Pressure Pa	Outlet 1 Total Pressure Pa	Outlet 2 Total Pressure Pa	Pressure Drop Pa
1	Base Model	45.04	14.30	11.78	18.95
2	Optimization Model 1	39.85	12.09	12.94	14.8
3	Optimization Model 2	38.14	11.34	13.12	13.68

Sl No	Model	Inlet velocity m/s	Outlet1 velocity m/s	Outlet2 velocity m/s
1	Base model	7.4416	4.27502	3.74
2	Optimization model 1	7.4416	3.8291	4
3	Optimization model 2	7.4416	3.94435	3.9644



## VI. CONCLUSION

The work demonstrates the possibilities of improving the performance of a HVAC duct by changing the design of the duct layout by giving Divergence, Consecutive bend, V shape cut to reduce the pressure drop, turbulence, recirculation zone near the bend and flow losses. As the flow becomes uniform for each outlet it equally distributes air in the Auditorium and it well conditioned which gave better performance of the HVAC duct Layout. The flow becomes uniform throughout the duct layout which indicates minimum energy loss. Due to the time limit only three different cases of the HVAC Duct layout were tried with different design and shapes. In the base model the inlet total pressure of 45pa is reduced to 15pa towards the ends of section B. Huge pressure loss in section B as expected, caused circulation and corresponding vortex generation in the duct at section A and section B, Velocity at outlets at section B is in the range of 4m/s - 7m/s which is on higher end, this loss overcome by giving a divergence consecutive bending, V shape cut of the section B. By giving the divergence consecutive bending V cut at the section A is to reduce loss of mass flow

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 12, December 2017

at duct layout can be minimized, cased circulation and corresponding vortex generation in the duct at section A and section B is minimized as compare with base model. Minimum pressure loss in section B as expected. So further modification was made in order to overcome these vortices. The total pressure remains uniform throughout, which indicates a minimum energy loss. The loss of total pressure and eliminate the cased circulation and corresponding vortex generation in the duct at section A and section B is eliminated. Velocity at outlets at section B is uniform throughout, which indicates Flow is efficiently controlled. It is found that the HVAC Duct layout gave satisfactory results with the flow of air is taking place uniformly and giving flow is efficiently controlled which is better compared to all other cases.

## REFERENCES

- [1]. MukeshDidwania, Lokesh Singh, Ashish Malik, Mangal S Sisodiya, Analysis of turbulent flow over a 90<sup>0</sup> bend of duct using in centralized A.C Plant by CFD code, e-ISSN:2278-1684, P-ISSN:2320-334X, volume 11, Issue 4 VER. I (Jul-Aug 2014).
- [2]. Santosh kansal, krishnakumarwaghe, Performance analysis of elliptical cross sectioned Bends of HVAC duct for less pressure drop, ISSN : 2347 - 4718 volume 1, issue 10, june-2014.
- [3]. Vladimir Zmrhal, Jan Schwarzer, Numerical simulation of local loss coefficients of ventilation duct fittings, July 27-30, 2009.
- [4]. A Rezwan, TMH Kabir, AKMH Arefin and MAR Sarkar, Numerical investigation of flow characteristics inside a HVAC duct, proceedings of the 6<sup>th</sup> international mechanical engineering conference & 14<sup>th</sup> annual paper meet (6imec&14apm) 28-29 September 2012, Dhaka, Bangladesh.
- [5]. Ana Paula P. dos Santos, Claudia R. Andrade, Edson L. Zapparoli, CFD Prediction of the Round Elbow Fitting Loss Coefficient, International Journal of Mechanical and Mechatronics Engineering Vol:8, No:4, 2014.
- [6]. Aslam A. Hirani, C. Udaya Kiran, CFD Simulation and Analysis of Fluid Flow Parameters within a Y-Shaped Branched Pipe, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), e-ISSN: 2278-1684, p-ISSN: 2320-334X, Volume 10, Issue 1 (Nov. - Dec. 2013), PP 31-34.
- [7]. D. Ghate, A. Isaacs, K. Sudhakar, P. M. Mujumdar and A. G. Marathe, 3d-Duct Design Using Variable Fidelity Method, Center for Aerospace Systems Design & Engineering (CASDE), Department of Aerospace Engineering, Indian Institute of Technology Bombay, Powai, Mumbai.
- [8]. Raviraj Gurav, Sanjay Gaikwad, Pritee Purohit, Duct Design using equal friction method & CFD, Assistant Professor, Army Institute of Technology, Pune.
- [9]. Dr. Neihad Al-Khalidy, Design optimization of industrial ducts using computational fluid mechanics, CSIRO, Melbourne, Australia 10-12 December 2013
- [10]. Cooling load calculations manual by American Society of heating, Refrigerating and Air Conditioning Engineers.
- [11]. A Bhatia, B E, HVAC made Easy: A guide to Heating and cooling load estimation.
- [12]. Shan K. Wang, Handbook of Air Conditioning and Refrigeration, McGraw-Hill New York San Francisco Washington, D.C. Auckland Bogotá
- [13]. Faye C Mc Quiston, Jerald D Parker: Heating, Ventilating, and Air Conditioning (Analysis And Design)
- [14]. James E Brumbaugh, HVAC Fundamentals 4<sup>th</sup> Edition.