STUDY AND ANALYSIS OF AIR FLOW THROUGH DUCT

Sulekha Walunj[1] , Shantanu Shirsath[2], Avinash Biradar[3], Asawari Salvi[4], Mansi Deore[5]

[1](Assistant Professor, Department of Mechanical Engineering, Terna Engineering College, Navi Mumbai, India)

[2,3,4,5](Students, Department of Mechanical Engineering, Terna Engineering College, Navi Mumbai, India)

***ABSTRACT*:** *Air conditioning systems are considered operating successfully based on the efficient supply of air in the air-conditioned space. Estimation of pressure losses in ducts is vital for the selection of duct size. The installation cost of duct in air conditioning system is about 20 to 30% of the total cost of equipment selection and power consumption of supply fans also adds to the running cost. Hence, it's important to minimize the capital and running cost of the duct system. Duct design is the design of enclosed passage for supplying conditioned air and then distributing the sir to specified areas according to the need. There are various advantages of efficiently designed and constructed interior ducts. One of the advantages is enhanced energy efficiency through the cancellation of duct leakage and reducing conductive heat gains/losses. This paper focuses on a study of design and improving air duct using computational fluid dynamics (CFD) analysis considering all the parameters related to air flow and its characteristics for enhancing duct system efficiency. The necessity of optimization of duct system is to improve supply air flow. It integrates theoretical and software tools to provide detailed comparative analysis of the advantages involved in selecting a particular shape (rectangular duct with Y-shaped bend and with 90-degree corner with sharp bend) of duct for a specified conditions. The focus of this paper will be on using CFD simulation tools to study velocity distribution of air in the duct at various sections, pressure difference at various outlets and distributions of air flow.*

Keywords- *Duct design, Duct system, HVAC, Pressure Distribution, Velocity Profile*

1. INTRODUCTION

An air conditioning duct is an enclosed passage or part that is used for the distribution of air throughout the air-conditioned space. Duct system also known as ductwork is a central part of Heating, Ventilation and Air Conditioning (HVAC) system. In duct systems, two set of ductworks are present, which is used to supply cool air and return air from air-conditioned space, also there is provision for general ventilation needs [1]. The purpose of the ducting system is to transfer the air from air handling unit to the area which needs to be air conditioned. The flow of air is due to the pressure difference created by the fan fitted in the air handling unit or the duct. The pressure difference which is responsible for the transport of air from one location to another reduces due to various fittings in the ducts (eg. Filters), bends present in the duct and also due to friction between air and duct surface [2]. When the working fluid (air) travels through the duct fittings the velocity and pressure profiles are disrupted (figure 1). The distortion of velocity profiles is caused by geometric conditions of the duct. There is rise of eddy zone in the fittings, which affect the shape of velocity profile (figure 1). After the fluid passes the duct fitting, the velocity profile tries to realign. The phenomenon mentioned above (distortion of the velocity profile) caused the local pressure loss [3].The amount of pressure loss in a system is directly proportional to the amount of energy consumed by the Fan in overcoming the loss in pressure. Thus, it becomes necessary to reduce the pressure losses in a ducting system.

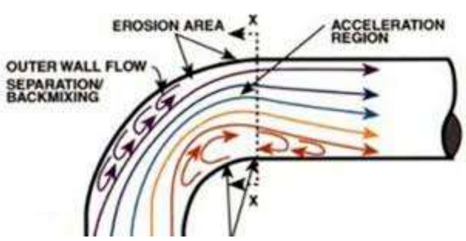


Figure 1: velocity profiles in a duct with an elbow [3]

For reducing the pressure losses in the duct system circular cross sectional ducts can be used instead of conventional rectangular ducts [4] however fabrication and installation of circular ducts is complicated and expensive. Also divergence can be provided for changing the direction of air flow instead of sharp 90° bend [5].

2. AIM AND OBJECTIVE

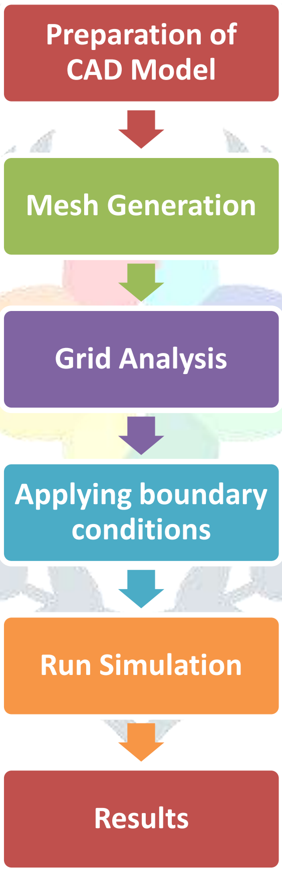
The objectives of effective duct design are to ensure comfort of occupants, proper air distribution, economical heating and cooling system operation and duct installation. The result of the duct design process will be a duct system (supply and return plenums, ducts, fittings, boots, grilles, and registers) that provides following functions.

- To determine size of duct so that the pressure drop across the air handling unit is within manufacturer and design specifications.
- To design duct for minimum pressure loss.
- To create vacuum-packed duct to provide proper air flow and to prevent air infiltration inside the air conditioned space or duct system from polluted zones.
- To design duct with stable supply and return air flows to maintain a neutral pressure in the air conditioned space.

3. METHODOLOGY

Duct design is done using equal friction method. The Equal Friction method makes assumptions for duct sizing by considering a constant pressure loss per unit of duct length. According to standards it is recommended to use 0.08 in. wg (~20 Pa) pressure loss per 100 ft (30 m) total length. This length is considered for the critical path, considering the longest branch in an air distribution system, and possibly has the highest sum of total pressure loss. However, short branches may have more elbows, fittings, and other flow restrictions casing greatest friction loss than the longest straight branch.

Flowchart of process followed



4. SIMULATION MODEL

The paper included the analysis of two types of bend viz. rectangular duct with Y shaped bend and with 90° corner with sharp bend and selection of one which encounters minimum distribution losses. The principle of Computational Fluid Dynamics was applied for fluid flow analysis, by using the following tool:-

Fluent – A commercial software package used to simulate the pressure drop inside the duct. The following sequence of steps were followed to obtain the desired results while performing the software analysis: Creating geometry in ANSYS > Meshing in ANSYS with Tool > Setting boundary types in ANSYS > Setting up a problem in FLUENT > Solving and obtaining solutions in FLUENT. Duct design is done using equal friction method. This CFD tool can be used for a whole building to analyze air pressure and velocity fluctuation for the duct.

4.1 DUCT MODEL

For designing of the system following steps are followed:

- Calculation of duct cross-section area
- Calculation of cooling load using Heat Load Software
- Selection of Air Conditioner Unit

Table 1: Specifications of duct system

Branch no.	CFM	Velocity (ft./min)	Area (m^2)	Aspect Ratio	A (in)	B (in)	Equivalent diameter (in)
1	480	1083	0.0412	1	8	8	8.75
2	960	1439	0.062	0.75	6	8	7.5

The first Model under study was the existing rectangular aluminum duct with 90 degree bend. The size and shape of the duct was designed as per the air quantity requirements. The solid model was created using SOLIDWORKS 2019 software. As shown below in fig 2 the model consists of one inlet and three outlet sections.

Due to high pressure variations and non-uniform velocity distribution in duct with 90 degree bend the model was modified and Y-bend section was tested. The parameters and boundary conditions for testing were same as that for duct with 90 degree bend.

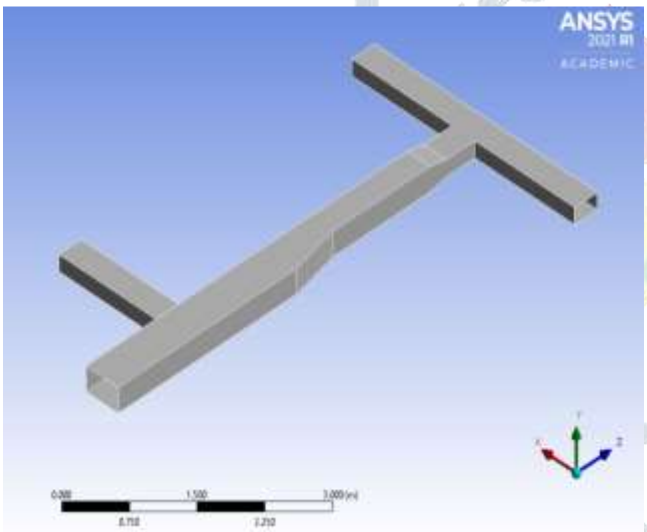


Fig 2. 90° bend CAD model

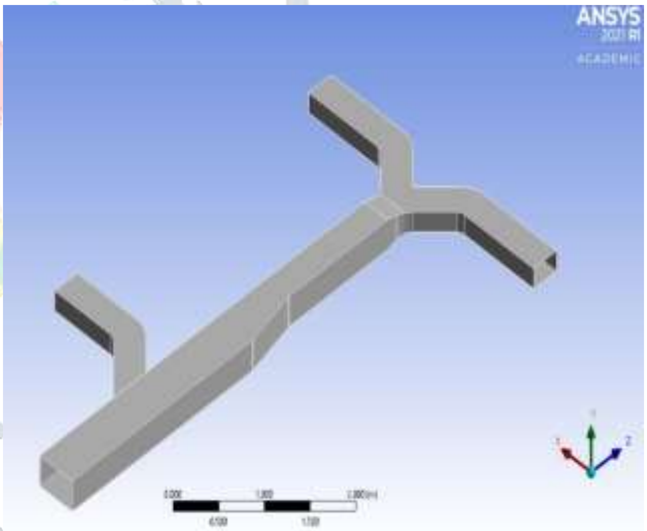


Fig 3. Y bend CAD model

4.2 MESH MODEL

4.2.1 Grid Independence test

The solution is said to be gird independent if changes made to the number of elements in a mesh model does not affect the value of the parameter calculated. Here as we increase the refinement of the mesh after a certain point there are no considerable changes in the solution. The point after which variation in results is not significant is the optimum point which gives optimum results.

Chart 1: gird test for 90° bend based on pressure

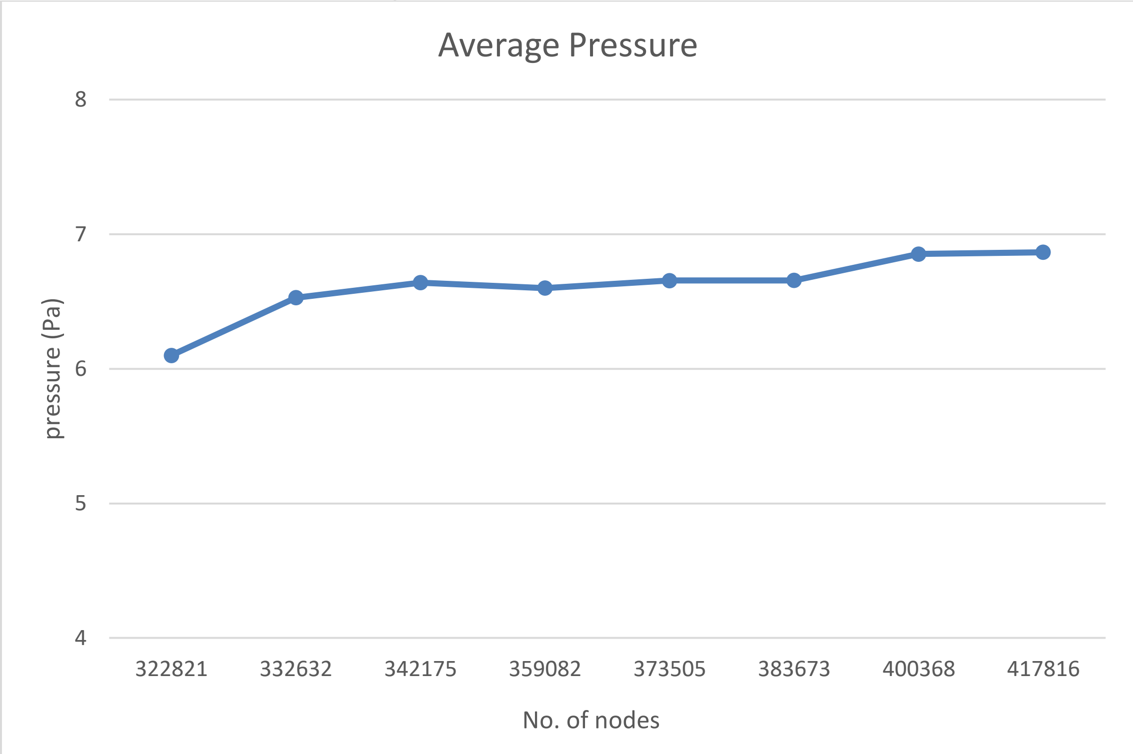
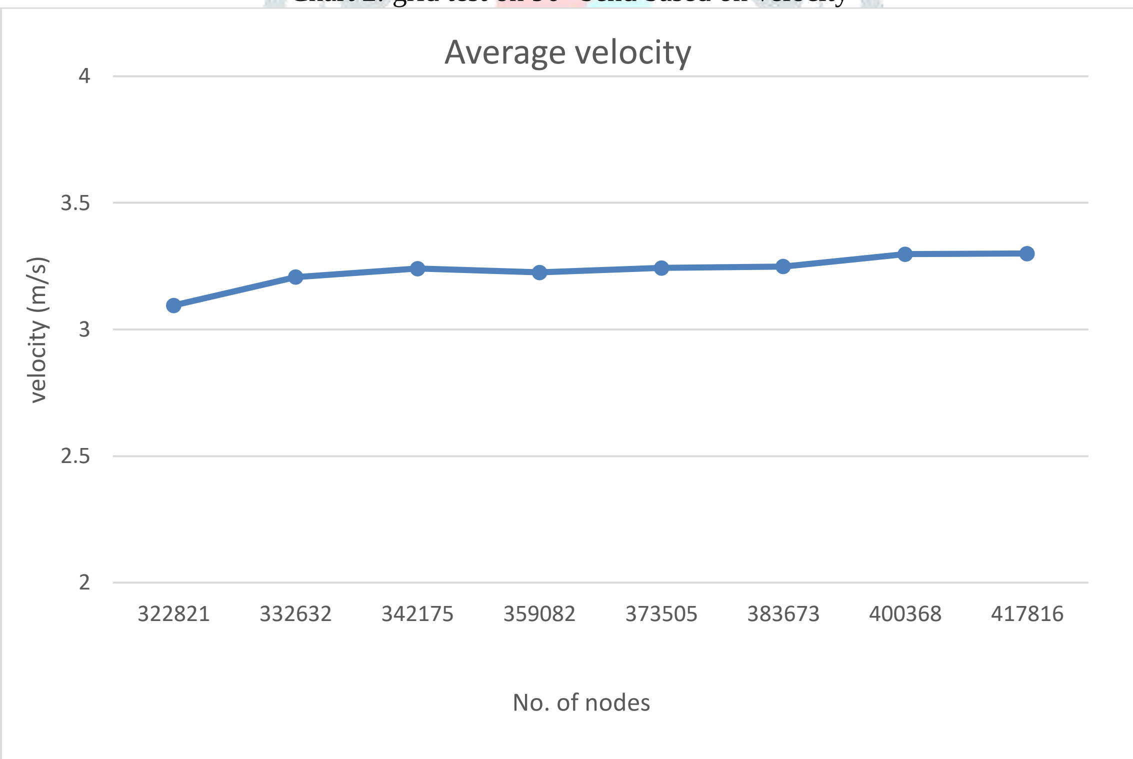


Chart 2: grid test on 90° bend based on velocity



From above charts viz., chart 1 and chart 2 it can be observed that there is no significant variation in the value of parameters after the number of nodes reach 342175. Therefore the element size is fixed to 25mm.

Chart 3: grid test on Y bend based on pressure

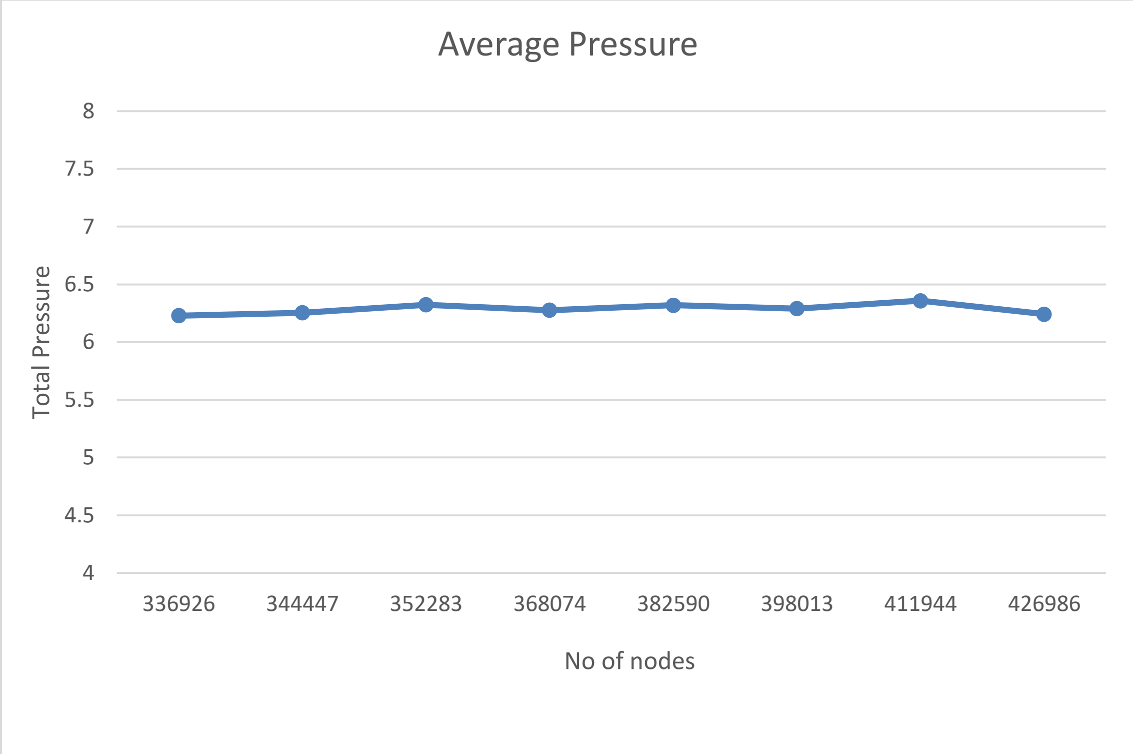
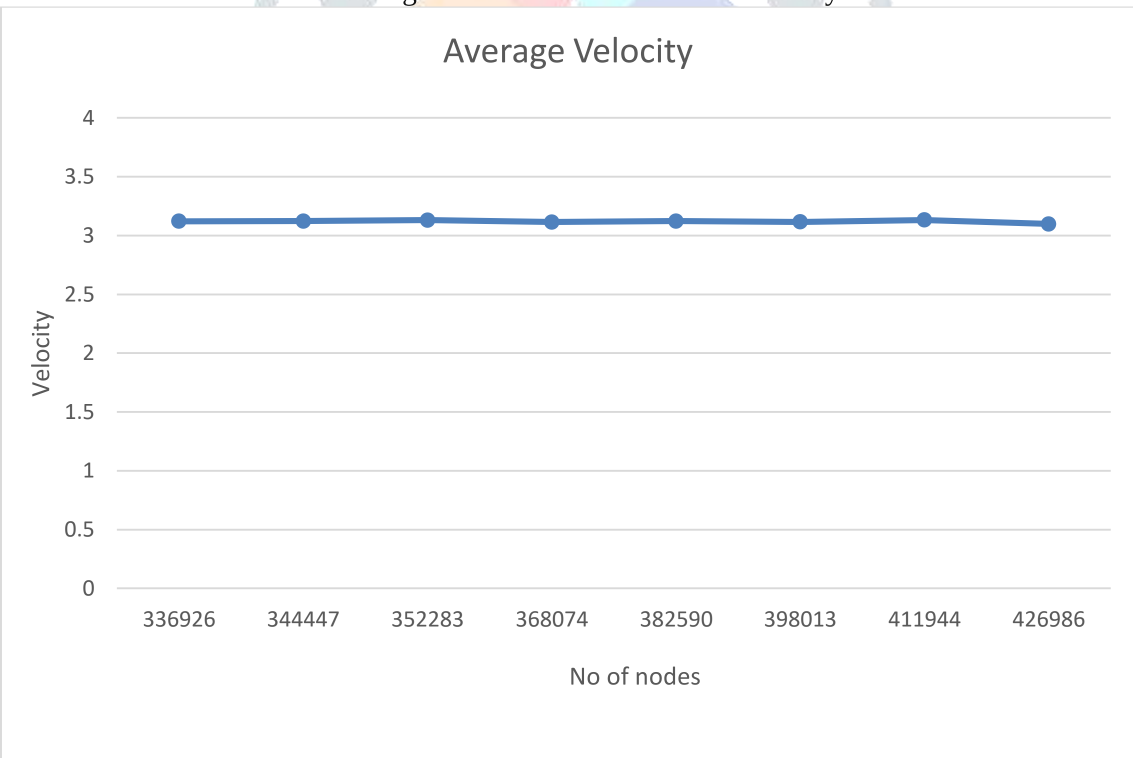


Chart 4: grid test on Y bend based on velocity



From above charts viz., chart 3 and chart 4 it can be observed that there is no significant variation in the value of parameters after the number of nodes reach 352283. Therefore the element size is fixed to 25mm.

The mesh model was prepared using ANSYS 2021 R1. Following are the details of the mesh model:

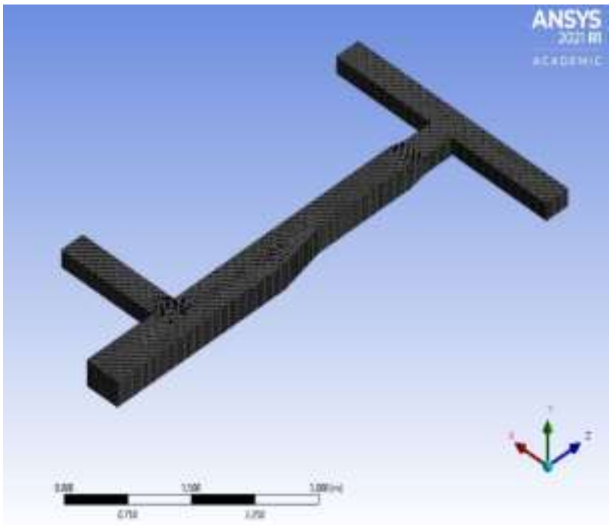


Fig 4. 90° bend Mesh model

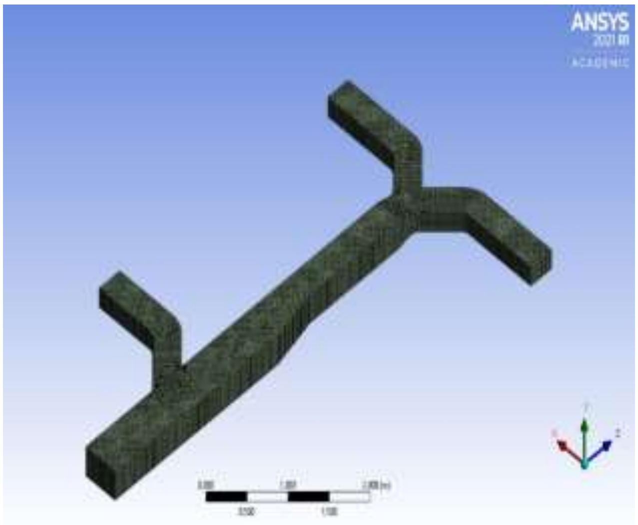


Fig 5. Y-bend Mesh model

Table 2: Mesh details for 90° bend

Object Name	*90° bend*
State	Solved
Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Element Order	Quadratic
Element Size	25.0 mm
Export Format	Standard
Export Preview Surface Mesh	No
Sizing	
Use Adaptive Sizing	No
Growth Rate	Default (1.2)
Max Size	Default (50.0 mm)
Mesh Defeaturing	Yes
Defeature Size	Default (0.125 mm)
Capture Curvature	Yes
Curvature Min Size	Default (0.25 mm)
Curvature Normal Angle	Default (18.0°)
Capture Proximity	No
Bounding Box Diagonal	7019.3 mm
Average Surface Area	6.1601e+005 mm²
Minimum Edge Length	79.796 mm
Quality	
Check Mesh Quality	Yes
Target Skewness	Default (0.900000)
Smoothing	Medium
Statistics	
Nodes	359080
Elements	83571
Method	Hex Dominant
Element Order	Use Global Setting
Free Face Mesh Type	Quad/Tri

Table 3: Mesh details for Y-bend

Object Name	*Y bend*
State	Solved
Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Element Order	Quadratic
Element Size	25.0 mm
Export Format	Standard
Export Preview Surface Mesh	No
Sizing	
Use Adaptive Sizing	No
Growth Rate	Default (1.2)
Max Size	Default (50.0 mm)
Mesh Defeaturing	Yes
Defeature Size	Default (0.125 mm)
Capture Curvature	Yes
Curvature Min Size	Default (0.25 mm)
Curvature Normal Angle	Default (18.0°)
Capture Proximity	No
Bounding Box Diagonal	7019.2 mm
Average Surface Area	3.6255e+005 mm²
Minimum Edge Length	8.5575e-002 mm
Quality	
Check Mesh Quality	Yes
Target Skewness	Default (0.900000)
Smoothing	Medium
Statistics	
Nodes	367892
Elements	86397
Method	Hex Dominant
Element Order	Use Global Setting
Free Face Mesh Type	Quad/Tri

4.2.2 Aspect ratio and Element quality

Aspect ratio: It is the measure of a mesh element's deviation from having all sides of equal length. A high aspect ratio occurs with long, thin elements. The meshing is done ensuring the aspect ratio to be closer to or equal to 1.

Element quality: During mesh creation it is made sure that the average element quality is above 0.9, to get accurate results.

4.2.2.1 Aspect ratio and element quality of duct with 90° bend

Chart 5: Aspect ratio

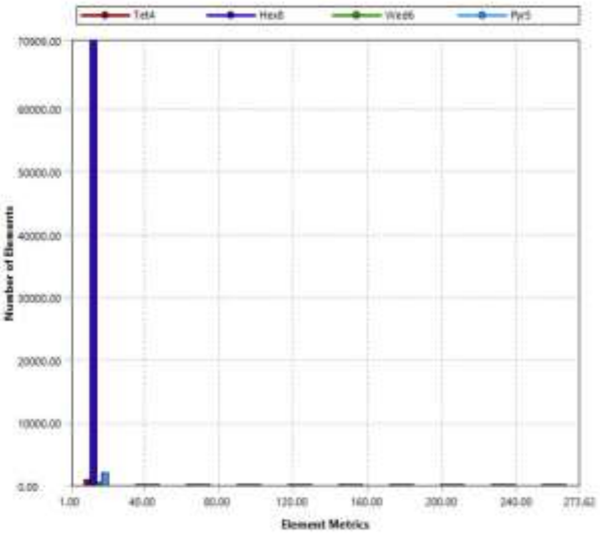
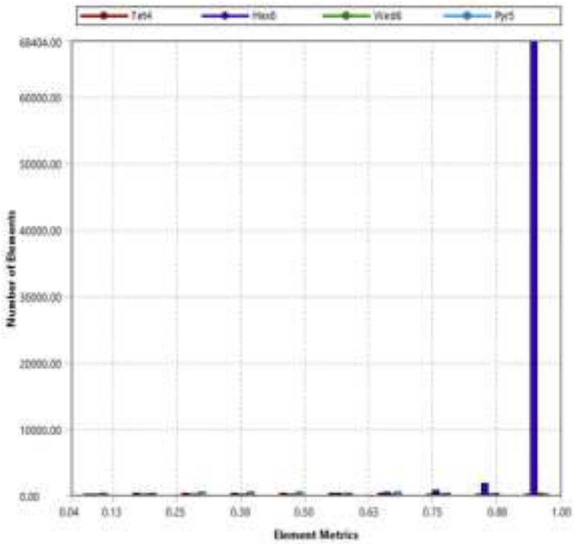


Chart 6: Element quality



4.2.2.2 Aspect ratio and element quality of duct with Y bend

Chart 7: Aspect ratio

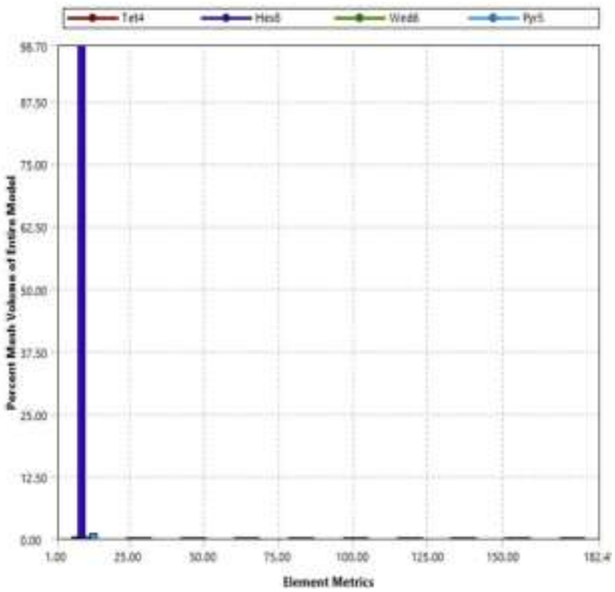
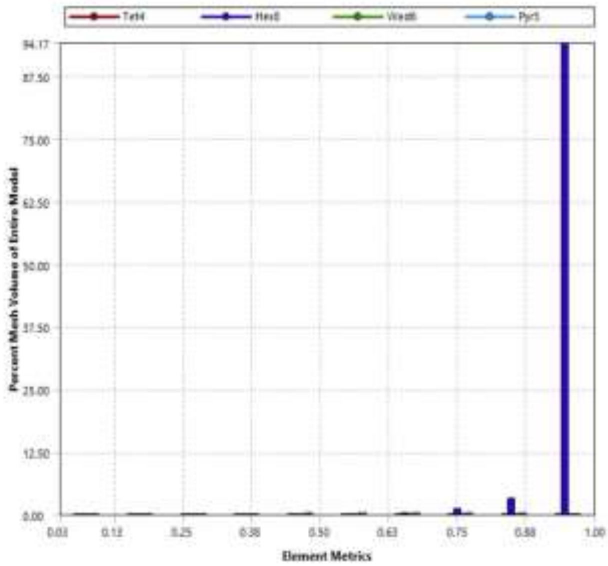


Chart 8: Element quality



4.3 GOVERNING EQUATIONS

4.3.2 k-omega model

The k-omega (k−ω) turbulence model is commonly used model to study the effect of turbulent flow conditions. It is from the Reynolds-averaged Navier-Stokes (RANS) family of turbulence models where the effects of turbulence are mathematically modeled.

The standard k−ω model is a low Reynolds model, i.e., it can be used for fluid flows with low Reynolds number where the boundary layer is moderately thick. This model shows excessive and early separations.

There exist different variations of the k-omega model such as **standard k-omega**, **baseline k-omega**, **k-omega SST**.

4.3.3 k-omega SST model

SST stands for shear stress transport. The SST model changes to a k−ϵ behavior in the free-stream, which circumvents the k−ω problem of being sensitive to the inlet free-stream turbulence properties.

The k−ω SST model delivers a better prediction of flow separation than most Reynolds-averaged Navier-Stokes models and also has advantage for its good behavior in adverse pressure gradients. It is the most commonly used model in the industry because of its high accuracy.

4.3.4 Mathematical Representation

The turbulent energy k is given by:

$$`k = \frac{3}{2}(U\,I)^2$$

Where U is the mean flow velocity and I is the turbulence intensity.

The turbulence intensity gives the level of turbulence and can be defined as follows:

$$I = \frac{u'}{U}$$

Where u′ is the root-mean-square of the turbulent velocity fluctuations given as:

$$u' = \sqrt{\frac{1}{3}\left({u'_x}^2 + {u'_y}^2 + {u'_z}^2\right)} = \sqrt{\frac{2}{3}k}$$

The mean velocity U can be calculated as follows

$$U = \sqrt{U_x^2 + U_y^2 + U_z^2}$$

The specific turbulent dissipation rate can be calculated using the following formula:

$$\omega = C_\mu^{\frac{3}{4}} \frac{k^{\frac{1}{2}}}{l}$$

Where Cμ is the turbulence model constant which usually takes the value 0.09, k is the turbulent energy, l is the turbulent length scale.

The turbulence length scale describes the size of large energy-containing eddies in a turbulent flow.

The turbulent viscosity νt is, thus, calculated as:

$$\nu t = \frac{k}{\omega}$$

4.4 Simulation parameters

Table 4: simulation properties

Fluid	Air
Time	Steady state analysis
Density	1.225 kg/m^3
Viscosity	1.7894e-05 kg/(m s)
Design pressure	1 atm
Viscous regime	Turbulent
Turbulence model	SST k- omega
Wall	No slip wall
Turbulent Intensity [%]	5
Turbulent Viscosity Ratio	10

4.5 Boundary Conditions

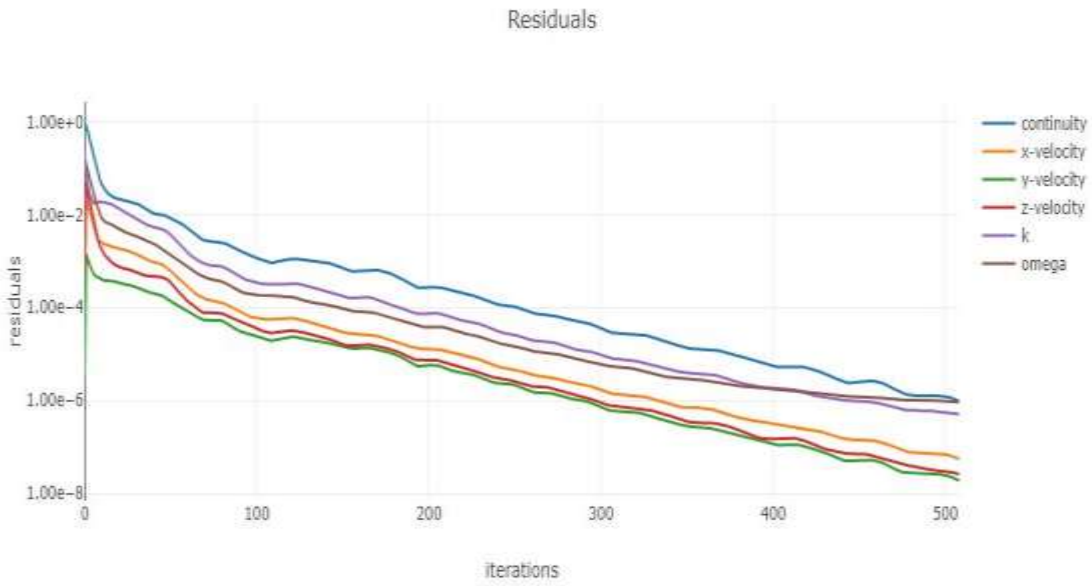
Table 5: Boundary Conditions

Zone	Boundary condition
Inlet	Vol. flow rate: 0.8076 m^3/sec
Outlet 1	Gauge pressure: 0 pa
Outlet 2	Gauge pressure: 0 pa
Outlet 2	Gauge pressure: 0 pa

4.6 Convergence Plots

4.6.2.1 Duct with 90^0 Bend

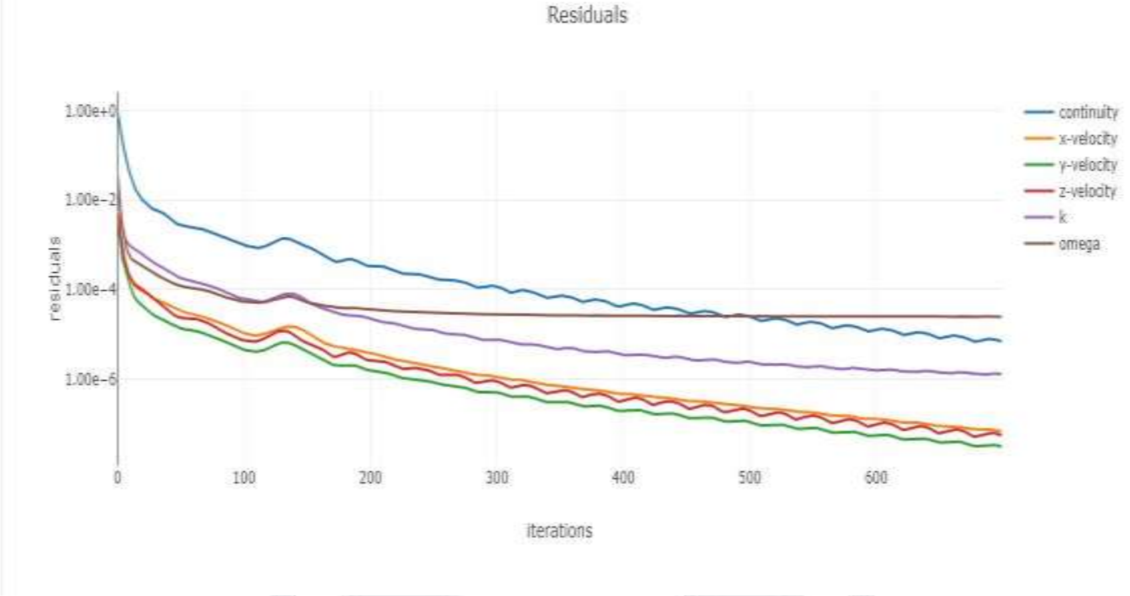
Chart 9: convergence plot for duct with 90 Bend



During the simulation target residual values were set to the power of 10^{-6}. Here as all the residuals reach the target value the solution is converged.

4.6.2.2 Duct with Y bend

Chart 10: convergence plot for duct with Y-Bend



Here as observed from the above graph the residual values have either attained a constant value or have reached the convergence criteria of 10^{-6}. Therefore it can be said that the solution is converged.

5 RESULTS AND DISCUSSION

5.2 Pressure analysis

5.1.1 Pressure Contour for 90° bend

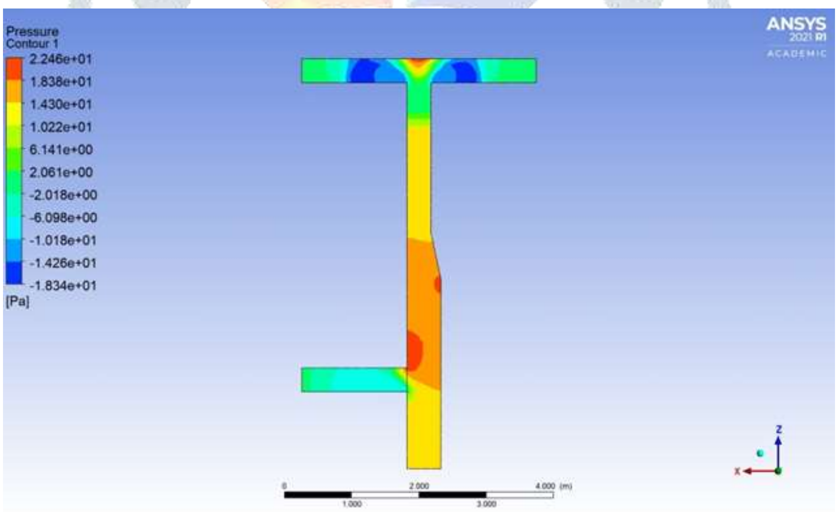
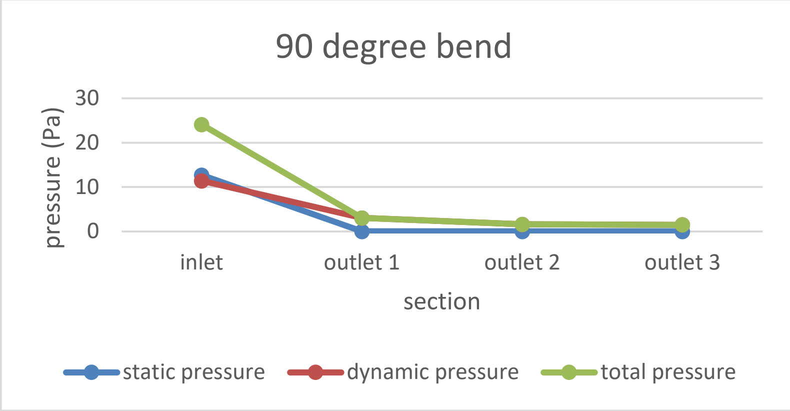


Fig 6: Pressure Contour

From fig 6 it can be observed that variable pressure zones are created throughout the duct. Near the first branch it can be observed that high pressure zone (red zone) is created due to sharp corner bend. This is due to sudden change in the direction of air and collision of air at the corner of the branch. Pressure energy is lost due to this kind of sharp bend. Also, there is low pressure cavity created at the bottom part of the first branch (light blue zone), this will create turbulence at the entry of the branch which will eventually increase the drop in pressure and contribute to the losses.

When air moves forward from the first branch toward the T- section the air strikes the wall of the duct at T-section and thus there is local increase in pressure in the region where air strikes the duct wall. Also, there is localized low pressure zone created at the entry of the T section on both branches (dark blue region) which will create turbulences and thus decrease the velocity of air coming out of the duct.

Chart 11: pressure across duct with 90^0 bend



From the graph it is clearly visible that there is a major loss in Total pressure as the air moves from inlet towards the last outlet viz outlet 3. A total pressure of 24 Pa is required to be given at the inlet to maintain the flow which drops down to 3 Pa at outlet 1 1.59 Pa at outlet 2 and 1.48 Pa at outlet 3

5.1.2 Pressure contour for Y bend

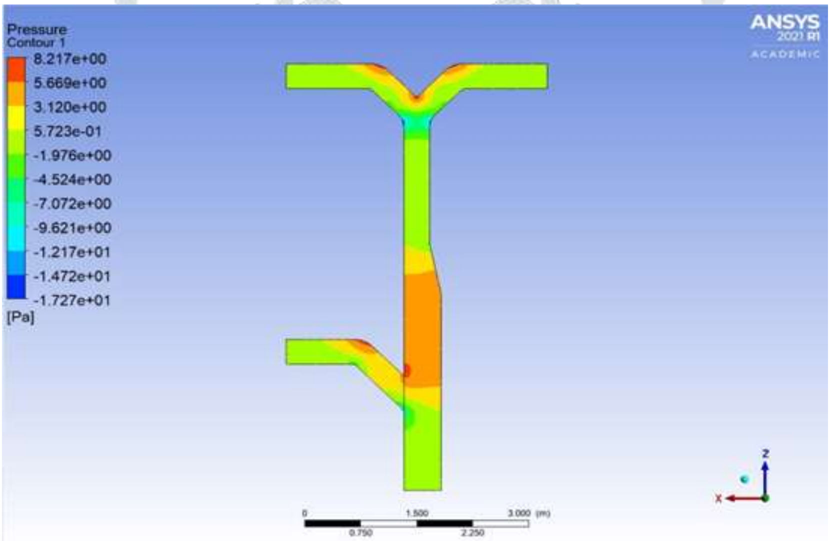
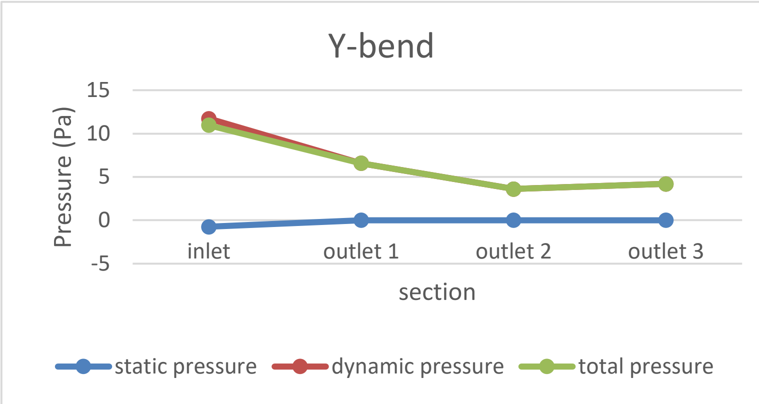


Fig 7: Pressure Contour

From fig 7 it is observed that there is comparatively uniform pressure variation across the duct when a Y-section bend and diverging bend is used for branching. As the air enters the first branch there is still high-pressure zone created as the air strikes the corner but it is comparatively less in this case. Also, the major problem of low pressure zone at the start of the branch is resolved due to diverging bend and there will not be any turbulence due to low pressure zone.

Also, when the air moves forward from the first branch towards the Y section there is very little effect of collision of air with the duct surface and thus very small high-pressure zone is created at the vertex of the Y section where the flow separates in two branches. Similar to the first divergence bend there is no creation of low-pressure zone at the entry of the branch and thus turbulence will be absent.

Chart 12: convergence plot for duct with Y-Bend



As seen from the above plot there is very less pressure drop as the air moves from inlet towards outlet section. The pressure required at the inlet to maintain the flowrate is also less i.e., 10.97 Pa (as compared to 24 Pa for 90^0 bend).

The pressure reduces to 6.58 Pa, 3.61 Pa and 4.19 Pa at outlet 1,2 and 3 respectively. The pressure drop is less in this case and the pressure values at the respective outlet sections are also larger as compared to duct with 90^0 bend.

5.3 Velocity contour

5.3.2 Velocity contour for 90° bend

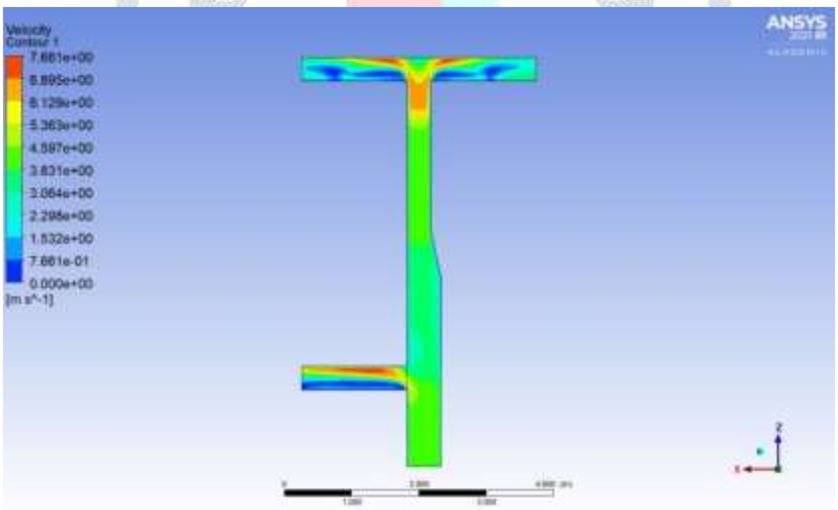


Fig 8: Velocity contour

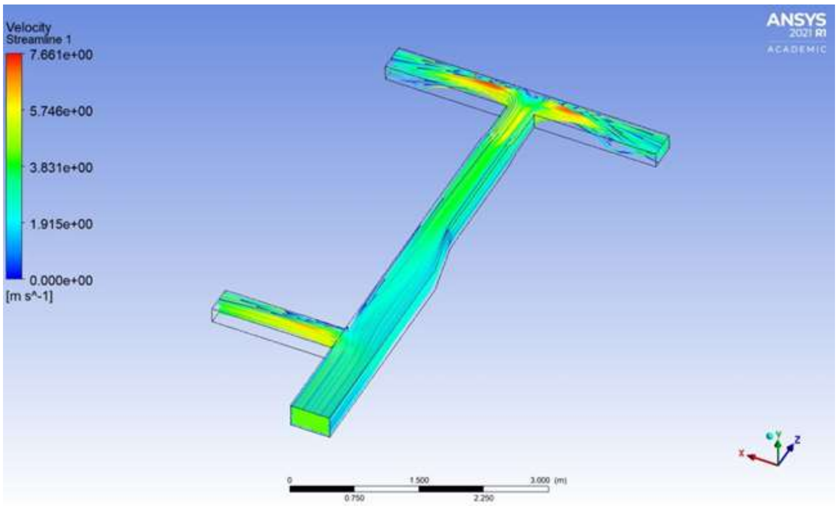


Fig 9: Velocity streamline for 90° bend

From the above fig 8 it can be observed that:

- There is non-uniform velocity distribution across the duct cross section.
- There is turbulence created at the entry of the branch due the low-pressure zones
- There is non uniform velocity at each outlet.

5.2.1 Velocity Contour for Y bend

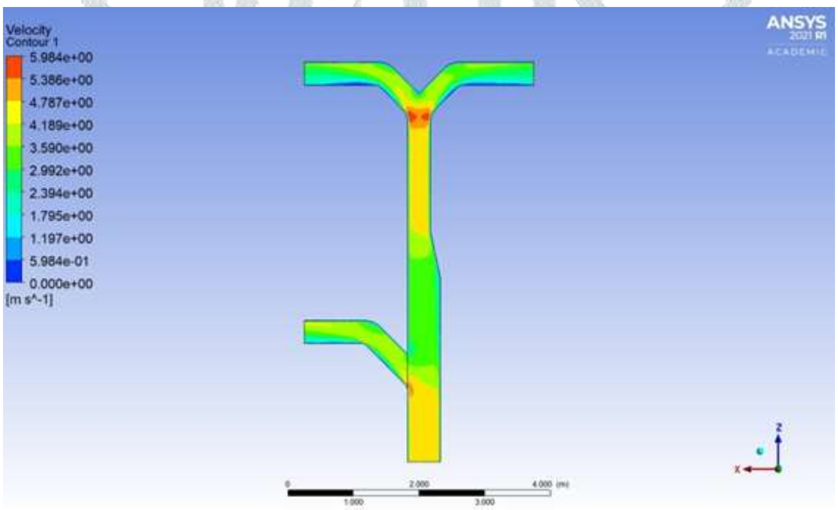


Fig 10: Velocity contour

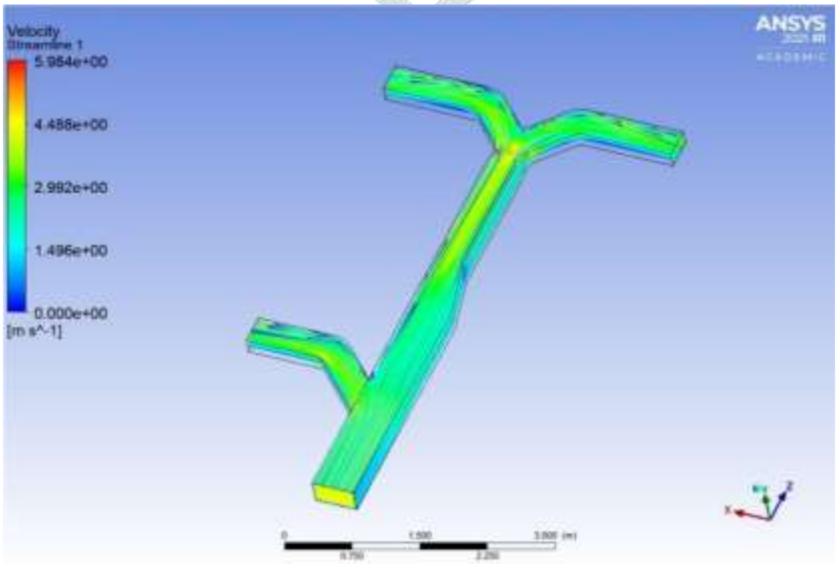


Fig 11: velocity streamline for Y-section bend

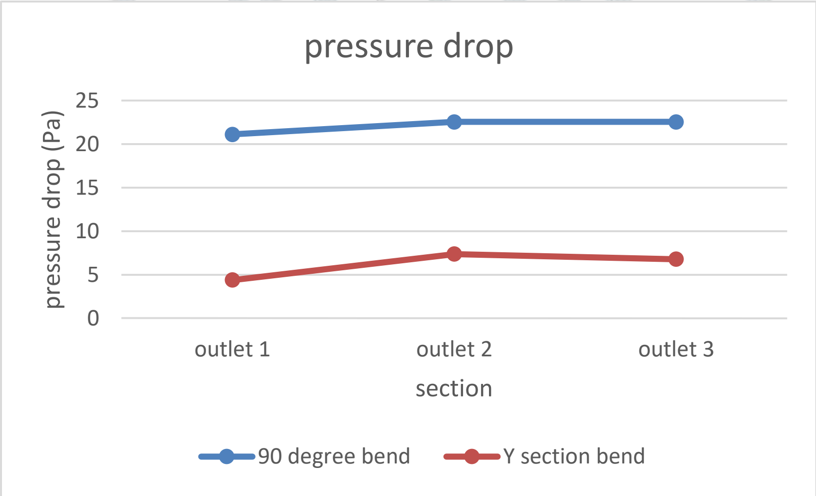
From fig 10 it is observed that:

- There is uniform velocity distribution throughout the duct and at the outlet.
- Turbulence is minimized as there is no low pressure zone created at the entry of the branch
- There is uniform velocity at each outlet

Table 6: Pressure distribution for ducts

Max Pressure (Pascal)	90° bend				Y bend			
	inlet	Out1	Out2	Out3	inlet	Out1	Out2	Out3
Static Pressure	12.71	0	0	0	-0.75	0	0	0
Dynamic Pressure	11.43	3.02	1.59	1.59	11.72	6.58	3.61	4.19
Total Pressure	24.14	3.02	1.59	1.59	10.97	6.58	3.61	4.19
Pressure Loss	-	21.12	22.55	22.55	-	4.39	7.36	6.78

Chart 13: pressure drop across ducts



From the above graph we can observe pressure drop across various sections of the duct. Thus we can conclude that pressure drop across duct with Y section bend is less as compared to the drop across duct with 90° bend.

6 CONCLUSION

- Using gradual bends instead of sharp bends for branching in ducting system reduces the pressure loss.
- 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 office room 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.
- In Y bend rectangular duct we observed minimum pressure losses as compared to the 90° sharp cornered bend and thus more uniform distribution of the supplied air.

REFERENCES

[1] Dr. Ghate., K,Sudhakar., and Majumdar,P.M., "3-D Duct Design Using Variable Fidelity Method" CASDE, Powai ,Mumbai, pp.1-11.
[2] Cai, J., and G. Thierauf. 1993. Discrete optimization of structures using an improved penalty function method. Eng.Opt.,Vol. 21, pp. 293-306.
[3] "Numerical simulation of local loss coefficients of ventilation duct fittings", Vladimir Zmrhal, Jan Schwarzer, Eleventh International IBPSA Conference | July 27-30, 2009

[4] "Design & Proportional Computational Assessment between a Conventional and Circular Duct System", JETIR July 2018, Volume 5, Issue 7.
[5] "Numerical analysis of air flow characteristics in heating ventilating and air conditioning (hvac) duct", Aravinda D , Karthik T , Ningappa D Kunabeva, IJIRSET, Volume:06, Issue: 12 | December 2017.
[6] ASHRAE. 2017. ASHRAE Handbook of Fundamentals. American Society of Heating Refrigerating and Air-conditioning Engineers, Atlanta, Georgia.
[7] ISHRAE 2017. ISHRAE Handbook. Indian Society of Heating Refrigerating and Air- conditioning Engineers, India.
[8] "Duct Designing In Air Conditioning System and Its Impact on System Performance", IRJET, Volume: 06 Issue: 07 | July 2019.
[9] "Design & Proportional Computational Assessment between a Conventional and Circular Duct System", JETIR July 2018, Volume 5, Issue 7.
[10] "Numerical Analysis of Air Flow Characteristics in Heating Ventilating and Air Conditioning (HVAC) Duct" ,IJIRSET, Volume:06 Issue:12 | December 2017.
[11] "A Study of Pressure Losses in Residential Air Distribution Systems", Authors: Abushakra, Bass Walker, Iain S. Sherman, Max H. | Publication Date 2002-07-01