COMPARATIVE STUDY OF AC DUCT AND DIFFUSER IN AUTOMOBILES USING CFD

G.SUDHAKAR   *gsudhakarmgm@gmail.com*
S.ABDUL WAJID   *saikalwajid@gmail.com*
N.DAMODAR   *damodardamodar34@gmail.com*
P.OWAISE AHMED   *powaiseahmed@gmail.com*
Department of Mechanical Engineering, Dr.K.V.Subba Reddy Institute of Technology,
Kurnool-518218 (AP), India

Abstract
Automobile HVAC climate control system seems to be absolutely monetarily in automobile. It warms the cabin when it's cold outside and cools when it is warm outside. There is a clear debate that this system, takes up fuel and take away the efficiency. The directionality pressure drop and other parameters are the important design consideration of the climate control system. Thus, the basic structures available in modern cars have undergone plenty of modifications and right now we end up with distributors. A comparative study between duct and diffuser will be helpful to understand the exact flow physics in the system. Modern era is enjoying the advantage of computational method that saves lot of time and money over conventional experimental validation. CFD, Computational fluid dynamics is found to be an exceptional tool in the process of design nowadays. In our project CFD is used to understand it compared to advantages of duct and diffuser system which aids in good amount of understanding flow behaviors inside the cabin. Useful help in future modification to establish a optimum design to the climate control system in automobiles.

I.   INTRODUCTION

It is difficult to control the car interior environment by using heat insulating material, double insulating glass and so on. The air-conditioning system is the only way to regulate the environment. Air quality in car interior is one of the most important research targets in the field of indoor air quality because our modern personal transport is ever more reliant on various vehicles, and cabins are usually characterized by their limited, enclosed space and relatively high personal density, conditions which are readily associated with air quality issues. Unfortunately, there are many potential pollutant sources in a cabin, such as volatile organic compounds from the materials, contaminants from passengers and their preferences, and pollutants taken in through the ventilation system. Consequently, it is highly possible for indoor quality air problems to arise in such cabins, and may even be more harmful than usual indoor environments in buildings Therefore, it is necessary to ensure efficient ventilation and sufficient capacity to remove such pollutants from inside the cabin. Perimeter loads on automobiles are heavier than those in buildings. In addition, automobiles can hardly be provided with eaves to intercept solar radiation, or sufficient thermal insulating material and multi-layered glass for thermal insulation and air-tightness. These factors compel automobiles to control their interior environment by air conditioning. In order to ensure a comfortable interior atmosphere, it is necessary to design

automotive air conditioning with care and consideration. We have to develop a highly accurate method for predicting a car's interior environment, and then improve the air-conditioning system to ensure a comfortable environment. At present, experimentation and computational fluid dynamics (CFD) analysis are generally used to predict the atmosphere inside a car.

II.   CONCEPT OF THE PROJECT

It is difficult to control the car interior environment by using heat insulating material, double insulating glass and so on. The air-conditioning system is the only way to regulate the environment. Duct and diffuser used to deliver a specific volume of air to the panel and rear console registers of a car cabin. The most complex and time consuming step for the iterative process of modifying the part (through CAD) and preparing for evaluation (Meshing). First to analyze the directionality of flow of air from diffuser and pressure drop of the system. Based on the CFD analysis of the performance there is a possibility of optimizing the geometry of the duct and diffuser that can bring about reductions in both size and cost of the system by examining the critical dimensional requirements.CFD is a powerful tool that computes the flow field in the duct and enables the geometric optimization of components at a very early design stage

III.   BACKGROUND INFORMATION

SCOPE OF THE PROJECT
A comparative study between duct and diffuser will be helpful to understand the exact flow physics in the system. Modern era is enjoying the advantage of computational method that saves lot of time and money over conventional experimental validation. CFD, Computational fluid dynamics is found to be an exceptional tool in the process of design nowadays.
In our project CFD is used to understand it compared to advantages of duct and diffuser system which aids in good amount of understanding flow behaviors inside the cabin.
Useful help in future modification to establish a optimum design to the climate control system in automobiles.

IV.   STEPS IN PROJECT
- ➢ Plan
- ➢ Cad design
- ➢ Prototype
- ➢ Evaluation
- ➢ Analysis
- ➢ Final design
- ➢ Production

V.   COMPUTATIONAL FLUID DYNAMICS

INTRODUCTION TO COMPUTATIONAL FLUID DYNAMICS
"Computational fluid dynamics" (CFD) is the use of computers and numerical techniques to solve problems involving fluid flow. CFD has been successfully applied in a huge number of areas, including many of interest to civil engineers (highlighted below). Examples include:

➢ Aerodynamics of aircraft and automobiles;

➢ Hydrodynamics of ships;

➢ Engine flows – IC engines and jet engines;

➢ Turbo machinery – pumps and turbines;

➢ Heat Transfer – heating and cooling systems;

➢ Combustion;

➢ Process Engineering – mixing and reacting chemicals;

➢ Wind Power;

VI.   BASIC PRINCIPLES OF CFD

The approximation of a continuously-varying quantity in terms of values at a finite number of points is called discretization.
The fundamental elements of any CFD simulation are:
(1) The **fluid continuum is discretized**, i.e. field variables (u, v, w, p, etc) are approximated by their values at a finite number of nodes.
(2) The **equations of motion are discretized, i.e.** approximated in terms of values at nodes:
      Differential or integral equations $\rightarrow$ algebraic equations
         (Continuum)   (Discrete)
(3) The **system of algebraic equations is solved** to give values at the nodes.

VII. WORKING OF CFD CODE
PRE-PROCESSOR
Input of a flow problem to a CFD program:
➢ Definition of the geometry of the region of interest: the computational Domain.

➢ Grid generation: the sub-division of the domain into a number of smaller, Non overlapping sub-domains, i.e., a grid (or mesh) of cells (or control Volumes or elements)

➢ Selection of the physical and chemical phenomena that needs to be modeled.

➢ Definition of fluid properties

➢ Specification of appropriate boundary conditions at cells which coincide with or touch the domain boundary.

The accuracy of a CFD solution is governed by the number of cells in the grid. Optimal meshes are often non-uniform: finer in areas where large variations occurs from point to point and coarser in regions with relatively little change. Over 50% of the time spent in industry on a CFD project is devoted to the definition of the domain geometry and grid generation.

SOLVER

Three distinct streams of numerical solution techniques:

a) Finite Difference Method:

➢ Describe the unknown φ of the flow problem by means of point samples at the node points of a grid of co-ordinate lines.

➢ Truncated Taylor series expansions are often used to generate finite difference approximations of derivatives of φ in terms of point samples of φ at each grid point and its immediate neighbors.

➢ The derivatives appearing in the governing equations are replaced by finite differences yielding an algebraic equation for the values of φ at each grid point.

b) Finite Volume Method:

➢ It is a special finite difference formulation and is central to main commercial CFD codes. The numerical algorithms:

➢ Formal integration of the governing equations of fluid over all the (finite) control volumes of the solution domain.

➢ Convert the integral equations into a system of algebraic equations.

➢ Solution of the algebraic equations by an iterative method.

➢ The most popular solution procedures are the TDMA line-by-line solver of the algebraic equations and SIMPLE algorithm to ensure correct linkage between pressure and velocity.

c) Finite Element Method:

➢ Use simple piecewise functions (e.g. linear or quadratic) valid on elements to describe the local variation of unknown flow variable φ.

➢ The governing equation is precisely satisfied by the exact solution φ.

➢ If the piecewise approximating functions for φ are substituted into the equation it will not hold exactly and a residual is defined to measure the errors.

➢ The residuals are minimized in some sense by multiplying them by a set of weighting functions and integrating.

➢ A set of algebraic equations for the unknown coefficients of the approximating functions.

POST-PROCESSOR

The raw output of the solver is a set of numbers corresponding to the values of each field variable (u, v, w, p, etc.) at each point of the mesh. This huge quantity of numbers must be reduced to some meaningful subset and, usually, manipulated further to obtain the desired predictive quantities. For example, a set of surface pressures and cell-face areas is required to compute a drag coefficient or a set of velocities and areas to determine a flow rate. Commercial packages often provide post-

processing facilities to plot , interpolate or simply extract quantities from the output dataset. A key component of post-processing is being able to visualize complex flows – either to indicate important features of the flow or, unfortunately, sometimes to establish why a calculation is diverging.

VOLUME MESHING

ICEM contains three different types of meshing models that can be used to generate a volume mesh starting from a suitably prepared surface mesh. The four types of meshing models are:
- ➤ Tetrahedral - tetrahedral cell shape based core mesh
- ➤ Prism - prism cell shape based core mesh
- ➤ Hexahedral - hexahedral cell shape based core mesh

Volume Generation Steps

The grid generation process is highly automated in T-Grid. In most cases, you can use the Auto Mesh feature of T-Grid to create the volume mesh from the surface mesh. However, in some cases, the boundary mesh contains irregularities or highly skewed boundary faces that can lead to an unacceptable volume mesh or cause T-Grid to fail while generating the initial mesh. As a rule of thumb, check the boundary mesh before attempting to generate the volume mesh. This tutorial demonstrates the mesh generation procedure for a problem that has multiple

regions. It also demonstrates the procedure for generating a volume mesh using the automatic refinement feature of T-Grid.

In this tutorial, you will learn the volume mesh generation process which comprises the following steps:

1. Read and display the boundary mesh
2. Check the skewness distribution of the boundary mesh
3. Check for free and unused (isolated) nodes
4. Improve the boundary mesh
5. Repair the boundary mesh
6. Enable the meshing of multiple regions
7. Initialize the mesh
8. Refine the initial volume mesh
9. Further refine the mesh
10. Check the cell skewness
11. Check and save the volume mesh

Generate the volume mesh

Mesh ….> Auto Mesh …..>Init & Refine

Quality Check for Surface and Volume Cell Elements

Skewness is one of the primary quality measures for a mesh. Skewness determines how close to ideal (i.e., equilateral or equiangular) a face or cell.

Value of Surface Skewness	Cell Quality
1	degenerate
0.9-<1	bad (sliver)
0.75-0.9	poor
0.5-0.75	fair

0.25-0.5   good
>0-0.25   excellent
0equilateral

$$\text{Skewness} = \frac{\text{Optimal Cell Size} - \text{Cell Size}}{\text{Optimal Cell Size}}$$

Volume Skewness sets the minimum cell skewness value for which node smoothing will be attempted. T-Grid will try to move interior nodes of cells with skewness greater than this value. Recommended values are 0.8 for 3D, and 0.4 for 2D.

A new mesh can be created by clicking on the tool bar and selecting New > Surface Mesh or Volume mesh. It is shown figure3.13
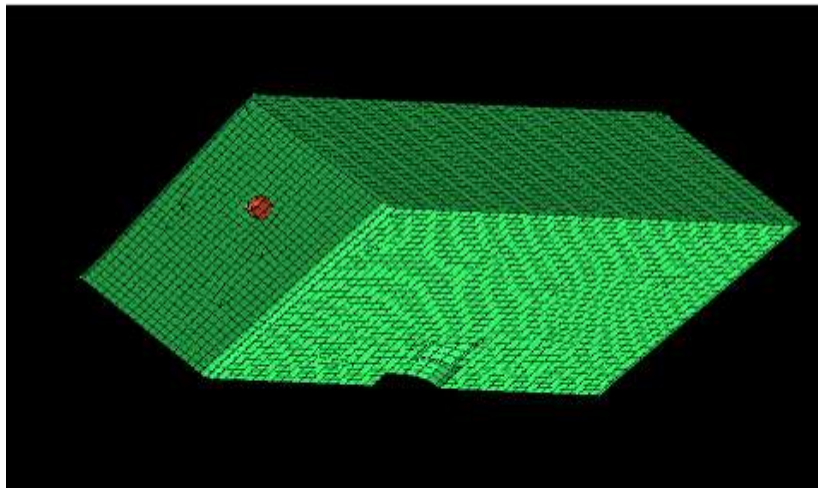
Figure 3.13 Surface Mesh

The model after volume mesh is shown in Figure 3.14.
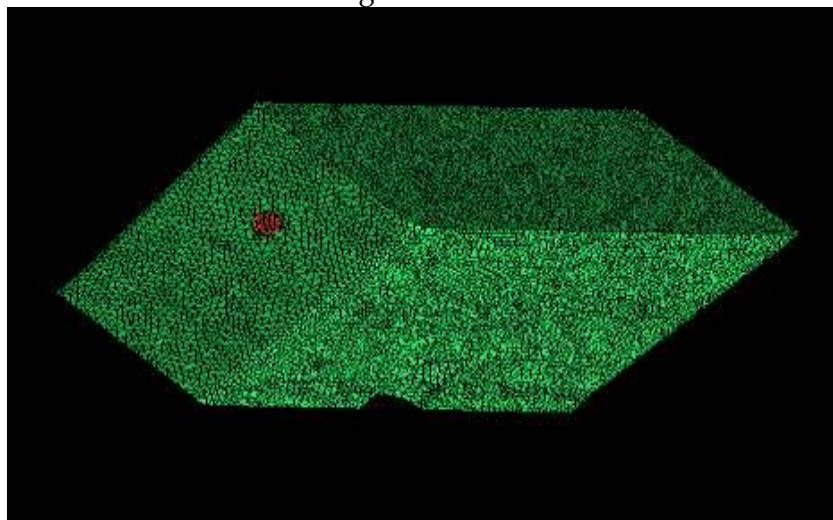
Figure 3.14 Volume Mesh

Meshing views
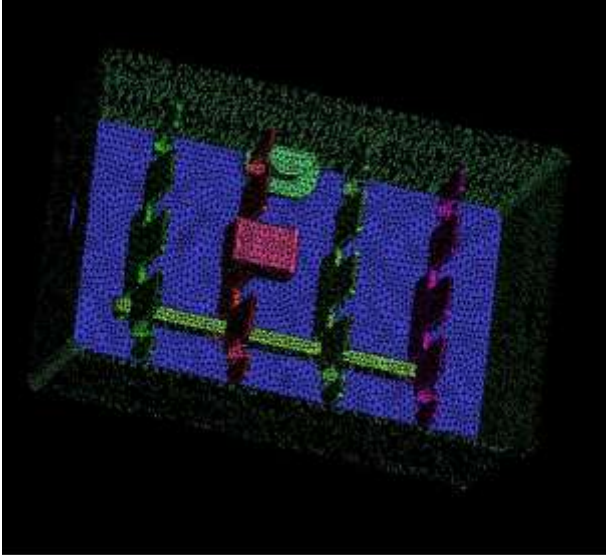
Figure 3.15 Meshing View 1

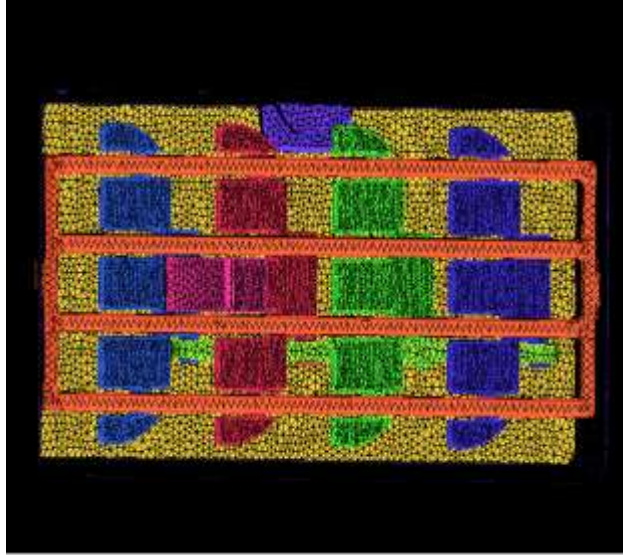Figure 3.16 Meshing View 2

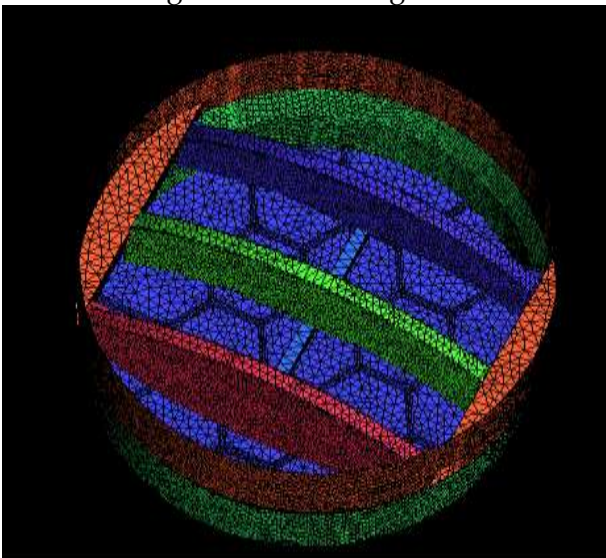Figure 3.17 Meshing View 3

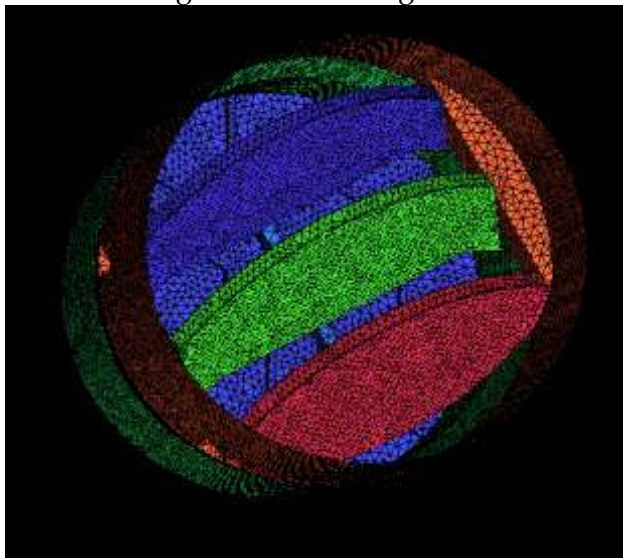Figure 3.18 Meshing View 4

➢ **Simulation**

To use ICEM CFD and FLUENT for solving flow problems, In the Physics Model Selection dialog:

➢ Three Dimensional model is selected in the Physics Models list.

➢ Stationary radio button in the Motion group box is selected.

➢ Implicit Unsteady in the Time group box is selected.

> ➢ Multiphase Mixture in the Material group box is selected.

> ➢ Volume of Fluid (VOF) in the Multiphase Model box is selected.

> ➢ Turbulent in the Viscous Regime group box is selected.

> ➢ K-Epsilon Turbulence in the Reynolds-Averaged Turbulence group box is selected.

> ➢ **What Is the Implicit Unsteady Model?**

The Implicit Unsteady model is the only unsteady model available with the Segregated Flow and Segregated Fluid Energy models. It uses the Implicit Unsteady solver. With the Coupled Flow and Coupled Energy models, the implicit unsteady approach is the alternative to the Explicit Unsteady one. The choice between these two approaches is based on the time scales of the phenomena of interest. When this model is activated, objects that offer a choice between Iteration or Time-Step for a trigger (such as Monitors or Scenes), can be set to update at each time-step.

VIII. MAIN RESULTS

ANALYSIS FOR DUCT

The Figure 4.1 shows the outlet velocity of the air from car ac duct with straight baffles. It shows almost the flow separation over the cabin. It spreaded all the direction in same velocity. The region shows blue color is the velocity at low level it considered zero velocity.

Contours of Velocity Magnitude (m/s)
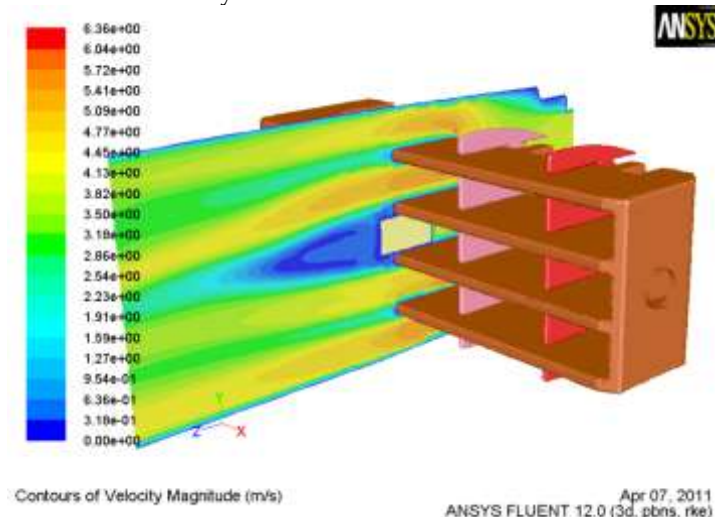
ANSYS FLUENT 12.0 (3d, pbns, rke)   Apr 07, 2011

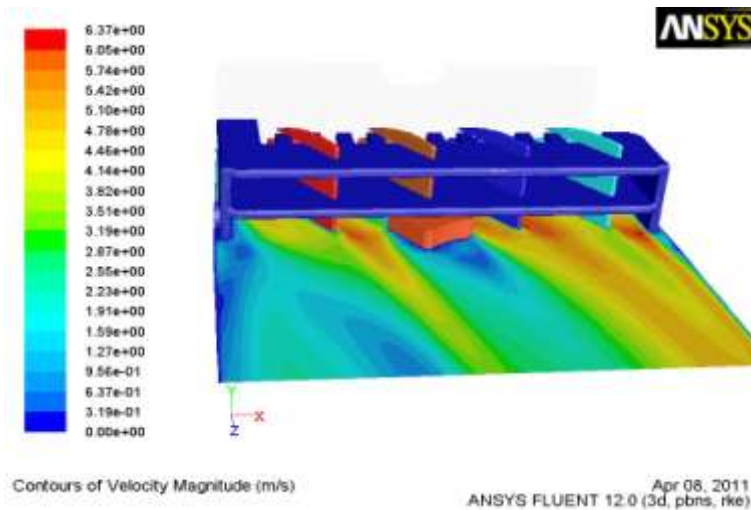Figure 4.1 Contours of Velocity Magnitude in Duct 1

Figure 4.2 Contours of Velocity Magnitude in Duct 2

The Figure 4.2 shows the outlet velocity of the air from car ac duct with right hand side baffles, slightly changed the angles for requirement of passenger. Its shows flow separation over the cabin. But little bit different from straight baffles, the corner side of the duct shows the flow velocity at low. The region shows blue color is the velocity at low level it considered zero velocity region.

Flow Paths

The figure 4.3 and figure 4.4 shows the path flow of air inside the duct with straight baffle arrangement. The vectors are shows how the fluids flow out of the duct with same velocity. Its shows stream line flow at the presence of knob there is a small amount of flow separation or variation.

The figure (4.5) and figure (4.6) shows the path flow of air inside the duct in right hand side baffle arrangement. The vectors are shows how the fluids flow out of the duct with same velocity. There is no high amount of fluid flow separation. Its shows stream line flow at the presence of knob there is a small amount of flow separation or variation.

There is no so much difference from these two models, because the fluids flow almost same as first one. All the fluids are trying to comes straight of the duct even though we have changed angles, this is the major disadvantages of the duct arrangement. There is another flow on left hand side, we have to change or moves the vertical baffles towards left-hand side. Both left and right hand flow direction must be equal in the same angle.
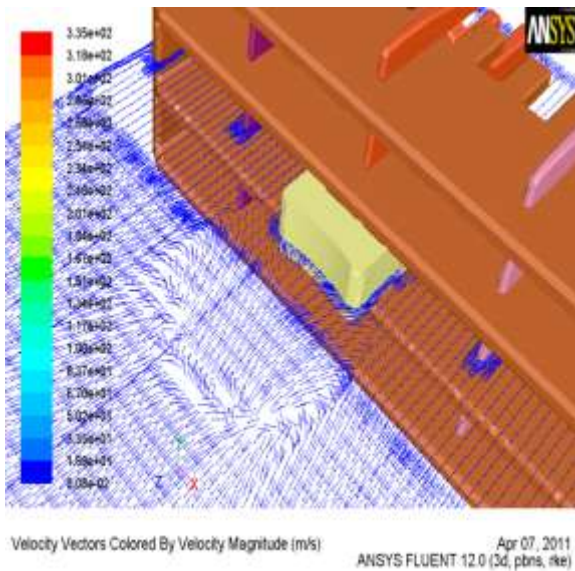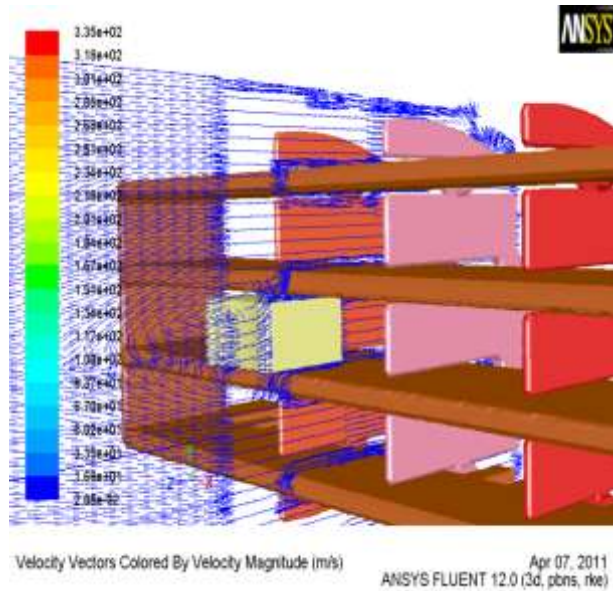
Figure 4.3 Flow Path Lines in Duct 1

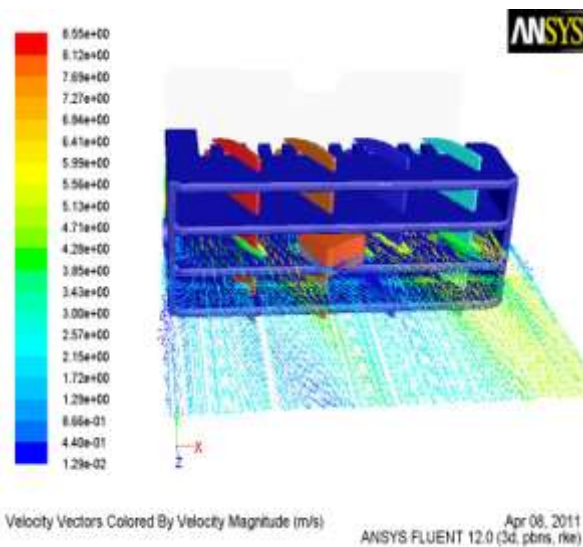Figure 4.4 Flow Path Lines in Duct 2

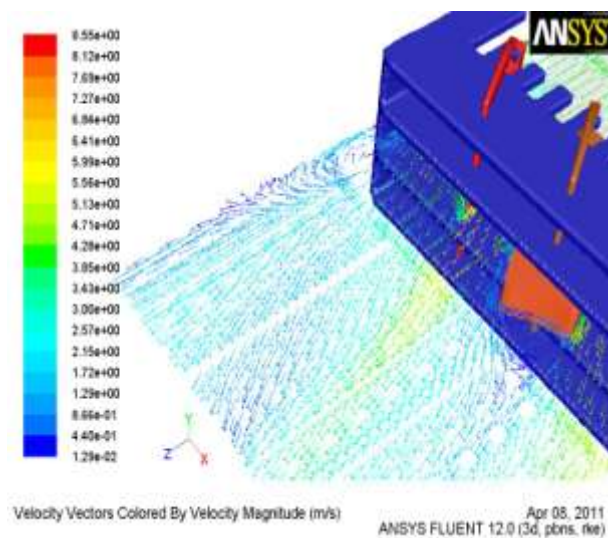Figure 4.5 Flow Path Lines in Duct 3

Figure 4.6 Flow Path Lines in Duct 4

IX. CONCLUSION

CFD simulation of automobiles duct and diffuser here resulted; the comparative results of duct and diffuser have been discussed.

In the ac duct arrangement the fluid flow goes straight over the cabin even though the angles or the baffle arrangement have changed. The directionality of the flow is not achievable. For upper flow; move to horizontal and vertical baffle section, because of these two coupled moments there is possible of flow will be highly turbulent. In the diffuser arrangement those problems can be avoided.

In the diffuser straight baffle arrangement can give the uniform velocity flow throughout the section, after changing the angles fluid flow at high velocity can be obtained. The directionality is achievable and very flexible because it is rotating element (360^0), there is possible of recirculation in the diffuser when angles are changed.

Comparatively diffuser is much better than the duct in automobiles, the directionality and the outlet velocity higher than the duct arrangement

REFERENCES

[1] Dindayal Mahto, Danish Ali Khan and Dilip Kumar Yadav, "Security Analysis of Elliptic Curve Cryptography and RSA", Proceedings of the World Congress on Engineering 2016 Vol I, WCE 2016, June 29 - July 1, 2016.

[2] Geetha G and Padmaja Jain, "Implementation of Matrix based Mapping Method Using Elliptic Curve Cryptography", International Journal of Computer Applications Technology and Research Vol. 3, Issue 5, 312 - 317, 2014.

[3] Naresh S. Badve, "Cryptography using Elliptic Curve with Matrix Scrambling", Journal for Research, Vol. 01, Issue 01, March 2015.

[4] Fatima Amounas, "On enhancement of Elliptic Curve Encryption of Amazigh Text using Graph Theory", International journal of Computer Science & Network Solutions, Vol. 4, No.3, pp. 1-10, 2016.

[5] D. Sravana Kumar, Ch. Suneetha and A. Chandrasekhar, "Encryption of Data Using Elliptic Curve Over Finite Fields", International Journal of Distributed and Parallel Systems, vol. 3, no. 1, 2012.

[6] William Stallings, "Cryptography and network security principles and practice", Prentice Hall, 5th Edition, 2011.

[7] Padma Bh, D.Chandravathi, P.Prapoorna Roja, "Encoding And Decoding of a Message in the Implementation of Elliptic Curve Cryptography using Koblitz's Method", International Journal on Computer Science and Engineering, Vol. 02, No. 05, pp. 1904-1907, 2010.

[8] O.Srinivasa Rao and S. Pallam Setty, "Efficient Mapping methods for Elliptic Curve Cryptosystems", International Journal of Engineering Science and Technology, Vol. 2(8), pp.3651-3656, 2010.

[9] B. Ravikumar, G. Naga Lakshmi, T. Surendra, A. Chandra Sekhar. "A Study on group theoretic Elliptic curves - An m implementation to Cryptography", International Journal of Science and Advanced Technology, Vol 1, No 6, 2011.

[10] V.Kamalakannana and S.Tamilselvan, "Security Enhancement of Text Message Based on Matrix Approach Using Elliptical Curve Cryptosystem", 2nd International Conference on Nanomaterials and Technologies, Procedia Materials Science 10, pp. 489-496, 2015.