

MAJOR PROJECT REPORT
ON
“STUDY OF FLOW OVER AN AUTOMOBILE”

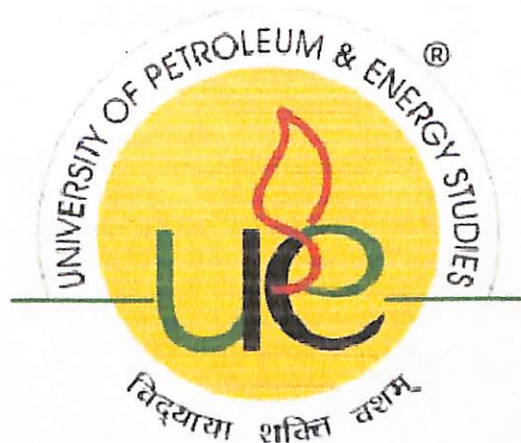
Submitted by:

ABHISAR MEHRA [R160209001]
OJAS CHOPRA [R160209043]
SHIVAM TANDON [R160209057]

COLLEGE OF ENGINEERING STUDIES

Under the guidance of

Mr. Ramesh Donga



Harnessing Energy through Knowledge

**IN PARTIAL FULFILLMENT OF THE DEGREE OF
BACHELOR OF TECHNOLOGY**

TO

**DEPARTMENT OF MECHANICAL ENGINEERING
UNIVERSITY OF PETROLEUM AND ENERGY STUDIES
DEHRADUN**

APRIL, 2013

THESIS COMPLETION CERTIFICATE

This is to certify that the major project report on “**STUDY THE FLOW OVER AN AUTOMPBILE**” by “**ABHISAR MEHRA, OJAS CHOPRA and SHIVAM TANDON**” in partial completion of requirements of the award of Degree of Bachelors of Technology is an original work carried out by them under our joint supervision and guidance.

It is certified that the work has not been submitted anywhere else for the award of any other diploma or degree of this or any other University.

Guide

Mr. Ramseh Donga

D. Ramseh Kumar
04/05/13.
(Name & Signature)

ACKNOWLEDGEMENT

It is with a sense of great satisfaction and pride that we are sitting down to pen down our major project thesis report. First and foremost we sincerely salute our esteemed Institution **University of Petroleum & Energy Studies, Dehradun** for giving this golden opportunity for fulfilling our warm dreams of becoming a bachelor.

The first and foremost person we would like to express our deep sense of gratitude and profound thanks to our guide **Mr. Ramesh Donga, Mechanical Department**, UPES Dehradun for his valuable advice, suggestions, insurmountable guidance which played a vital role in carrying out our project work successfully.

We would like to thank all faculty, staff, and students of Department of Mechanical Engineering and for rendering help during various stages of the project work.

Abhisar Mehra [R160209001]

Ojas Chopra [R160209043]

Shivam Tandon[R160209057]

ABSTRACT

This project deals with creating an understanding regarding the effect of aerodynamics on automobile using CFD software.

Aerodynamic forces form major part of total resistance. To calculate the aerodynamic forces (coefficient of lift and drag) a 3 dimensional CAD model was created using Solidworks of Maruti 800. Flow simulation of the surface profile have been performed using CFD techniques The analysis have been done using the FEA analysis.

3D analysis of surface profile of car has been done using GAMBIT and FLUENT. The results obtained from Fluent will help us to visualize the velocities, pressure contours, and turbulence intensities & pressure plot on the surface of cars.

TABLE OF CONTENTS

- List Of Figures
- List Of Graphs
- Nomenclature

CHAPTER 1. INTRODUCTION

- 1.1 vehicle aerodynamics
- 1.2 numerical prediction of aerodynamics performance

CHAPTER 2. PROJECT OBJECTIVE

- 2.1 Aims and objectives

CHAPTER 3. METHODOLOGY

- 3.1 Making a 3-D CAD model of a basic vehicle profile
- 3.2 Meshing of the CAD model in GAMBIT
 - 3.2.1 Importing the geometry
 - 3.2.2 Brick creation
 - 3.2.3 Aligning the model
 - 3.3.4 Deleting volume
 - 3.2.5 Meshing the geometry
 - 3.2.6 Boundary conditions
- 3.3 Analyzing in Fluent
 - 3.3.1 Reading the file
 - 3.3.2 Checking the grid and scaling the model
 - 3.3.3 Define models and materials
 - 3.3.4 Boundary conditions
 - 3.3.5 Define solution parameters
 - 3.3.6 Enable the plotting of residuals
 - 3.3.7 Start the iterations

CHAPTER 4. LITERATURE REVIEW

4.1 Automobile aerodynamics

4.1.1 Viscous boundary layer

4.1.2 Boundary layer separation

4.2 CFD (Computational fluid dynamics)

4.2.1 Discretization methods in CFD

4.2.2 How does CFD code works

4.2.3 Advantages of CFD

4.2.4 limitations of CFD

CHAPTER 5. RESULTS AND DISCUSSIONS

5.1 Static pressure plot

5.2 Velocity magnitude

5.3 Turbulent kinetic energy

5.4 Total pressure plot

CHAPTER 6. CONCLUSION

CHAPTER 7. REFERENCES

List of figures :

Figure Number	List of Figures	Page Number
3.1.1	Basic car dimension in solid works	11
3.1.2	Extrusion form mid-plane	12
3.1.3	Fillet command applied on side of model	12
3.2.1.1	Importing IGES file	13
3.2.1.2	Browsing the File	14
3.2.1.3	The imported view of geometry	14
3.2.2.1	Brick creation around the model	15
3.2.3.1	Aligning model to virtual road surface	15
3.2.4.1	Subtracting car volume from brick volume	16
3.2.5.1	Selecting face mesh for the car model by defining interval size	16
3.2.5.2	Volume mesh for remaining brick volume	17
3.2.6.1	Final created mesh	18
3.3.1	Reading the file	19
3.3.2.1	Checking grid and scaling	19
3.3.3.1	Defining model	20
3.3.3.2	Defining material	20
3.3.4.1	Defining boundary conditions	21
3.3.4.2	Setting operating conditions	21
3.3.5.1	Solution parameters	22
3.3.6.1	Plotting residue	22
3.3.7.1	Starting iteration	23
4.1.1	Pressure distribution on a car	25
4.1.1.1	Boundary layer on a flat plate	26
4.1.2.1	Boundary layer separation	27
5.1.1	Contours of static pressure	32
5.1.2	Static pressure vs car length	32
5.2.1	Contours of velocity magnitude	33
5.3.1	Contours of turbulent kinetic energy	33
5.4.1	Contours of total pressure	34
5.4.2	Total pressure vs car length	34

1. INTRODUCTION

1.1 Vehicle aerodynamics

Aerodynamics of vehicles is a complex subject because of interaction between air flow, vehicle body and the ground as complicated geometrical shapes are involved. Aerodynamics has its own importance as both vehicle stability and fuel consumption are affected by it. Reducing the aerodynamic drag improves the fuel economy and results can be seen for constant speed operation. Though the actual benefit for normal use is less, as reducing the drag does not significantly reduce the energy required for acceleration at normal speeds.

The development of vehicles with low drag shapes has been increasing rapidly because of past experience and better computational techniques. Reducing the drag is not only an aerodynamic consideration. The air flow affects the aerodynamic lift forces and the position of center of pressure. Though the presence of ground has only a little effect on drag forces, it has a significant effect on lift forces.

The comparison of drag forces data from different tests should be avoided, as the absolute values will be dependent on the details of experiments

1.2 Numerical prediction of aerodynamic performance

Traditional experimental approaches can be optimized by drawing on past experience- for e.g. , the degree of corner rounding, the bonnet and rear window slopes etc Though this approach still requires experimental data to determine aerodynamic characteristics of basic vehicle shape. One of the most high tech tools for predicting the aerodynamic performance in today's automotive industry is Computational Fluid Dynamics (CFD).

1.2.1 Computational fluid dynamics

In the beginning of 1960s, the researchers came across computational methods to solve their flow problems around complex geometries. This provided them with a powerful tool of computational fluid dynamics to analyze the flow within or outside the body. Computational fluid dynamics provides a third approach in the philosophical study and development of whole discipline of fluid mechanics. Throughout most of the twentieth century the study and practice of fluid dynamics involved the use of pure theory on one hand and pure experiment on other hand. However the advent of high speed digital computer combined with the development of accurate numerical algorithms for solving physical problems on these computers has revolutionized the way we study and practice fluid dynamics.

However to keep things in perspective, computational fluid dynamics provides a third approach to physical problems and nothing more than that. We have to still rely on experiment and theory to validate our results. The future advancement of fluid dynamics will rest upon the proper balance of all three approaches, with

computational fluid dynamics helping to interpret and understand the result of theory and experiment and vice versa.

The three basic governing equations of fluid dynamics applicable to CFD are: -

- Continuity equation- This states that the net outflow of mass from a control volume is equal to the time rate of decrement of mass inside the control volume. Mathematically it can be written as follows: -

$$\frac{\partial}{\partial t} \iiint_V \rho dV + \int_S \rho V \cdot dS = 0$$

The physical meaning of this equation is that mass cannot be created and nor be destroyed. The above equation is known as integral form of continuity equation. No assumption was made during deriving this equation except the fact that flow is continuum. Thus above equation is applicable to steady, unsteady, incompressible, compressible and any other type of flow.

- Momentum equation- The net force on a particle is equal to the time rate of change of linear momentum in an inertial reference frame. The momentum equation in x, y and z direction can be written as: -

$$\frac{\partial(\rho u)}{\partial t} + \nabla \cdot (\rho u V) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho f_x$$

$$\frac{\partial(\rho v)}{\partial t} + \nabla \cdot (\rho v V) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho f_y$$

$$\frac{\partial(\rho w)}{\partial t} + \nabla \cdot (\rho w V) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho f_z$$

- Energy equation- Rate of change of fluid inside the element is equal to the net flux of heat into the element and the amount of work done on the element due to body and surface forces.

$$\begin{aligned} & \rho \frac{D}{Dt} \left(e + \frac{V^2}{2} \right) \\ &= \rho q + \frac{\partial}{\partial x} \left(k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left(k \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left(k \frac{\partial T}{\partial z} \right) - \frac{\partial(u\tau)}{\partial x} - \frac{\partial(v\tau)}{\partial y} - \frac{\partial(w\tau)}{\partial z} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \\ & \quad \frac{\partial \tau_{zx}}{\partial z} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho f \cdot V \end{aligned} \quad (3.12)$$

The approach in computational fluid dynamics is to separate the flow into 2 parts

- (1) The main stream, where viscous effects are negligible
- (2) The boundary layer, where viscous effects are significant.

The steps involved in the computational fluid dynamics are as follows: -

- Pre-processor
- Solver
- Post-processor

The pre-processor involves the following steps

- Geometry acquisition/Geometry making- This is the first step in simulating any flow problem in CFD. The geometry on which the simulation has to be done is either to be made or acquired from other source.
- Discretization- This step involves the division of fluid domain and the geometry into small elements or volumes with the help of meshes. The mesh can be structured as well as unstructured.
- Boundary condition- This step involves the dividing of whole fluid domain into different physical phases like inlet, outlet, wall, symmetry etc.

The solver involves the process of simulating the flow around the meshed domain to acquire the results.

Post-processor is used to visualize the results.

2. PROJECT OBJECTIVE

2.1 . Aims and objectives:

The aim of the project is to study the flow of air an automobile. The advantages and disadvantages of various shapes of the vehicle bodies is studied. literature is verified using SOLIDWORKS, GAMBIT and FLUENT software.

A single geometry of the basic car profile will be studied .

The objective of the project is:-

- To study the aerodynamic characteristics of the vehicle profile.
- To conduct analysis using GAMBIT and FLUENT over the car profile.

3. METHODOLOGY

These are the following basic steps followed :

- Making a 3-D CAD model of a basic vehicle (Maruti 800) profile in SOLIDWORKS
- Meshing the cad model in GAMBIT.
- Studying the flow in FLUENT.

3.1 Making a 3-D CAD model of a basic vehicle profile

A basic car profile was made in SOLIDWORKS 11. For this first of all the sketch of the side view of the car body was made as shown in *fig.3.1*

Basic curvatures were given in the sketch itself instead of giving extrude command in the extruded model as it may have created problem during meshing in GAMBIT.

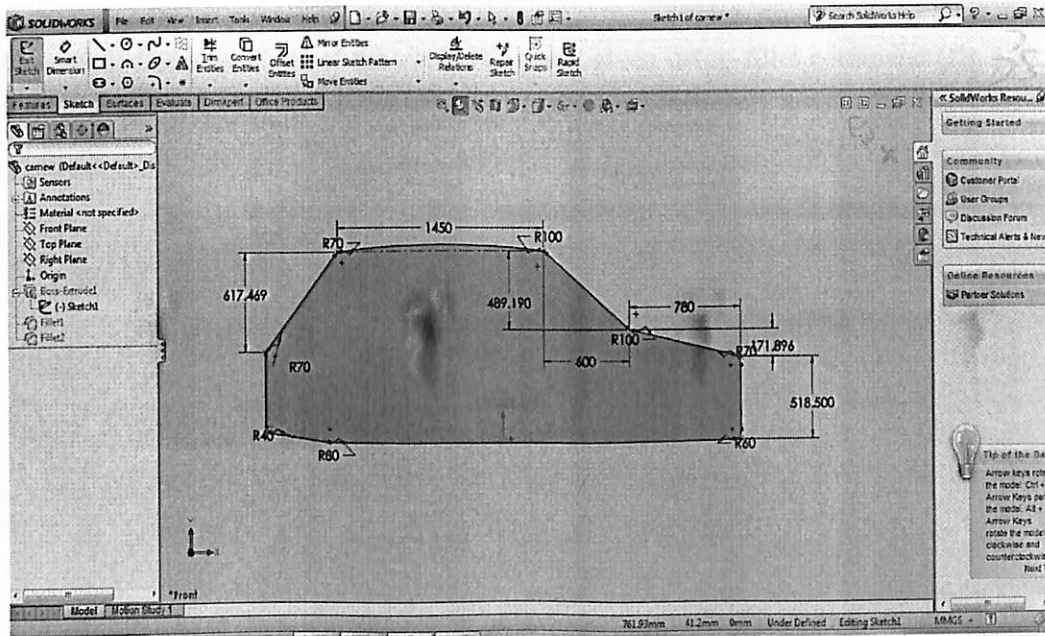


Fig.3.1.1: Basic car dimensions in solidworks.

The sketch made was then extruded from mid plane to the width of the car (refer *fig. 3.2*)

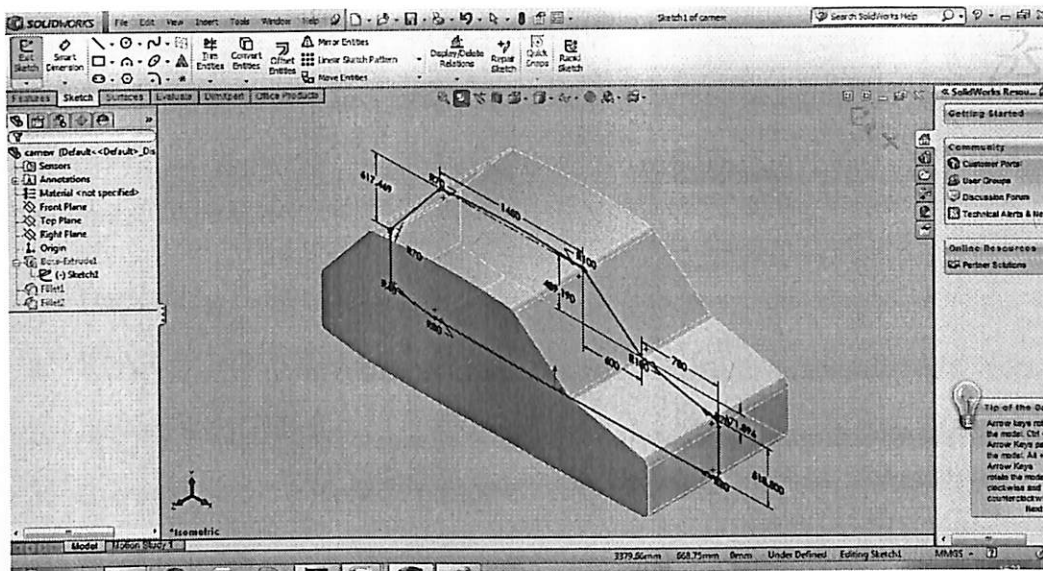


Fig.3.1.2: Extrusion from mid plane.

Curvature from the sides of the car profile was given using fillet command (fig.3.3), completing the 3-D model of the car.

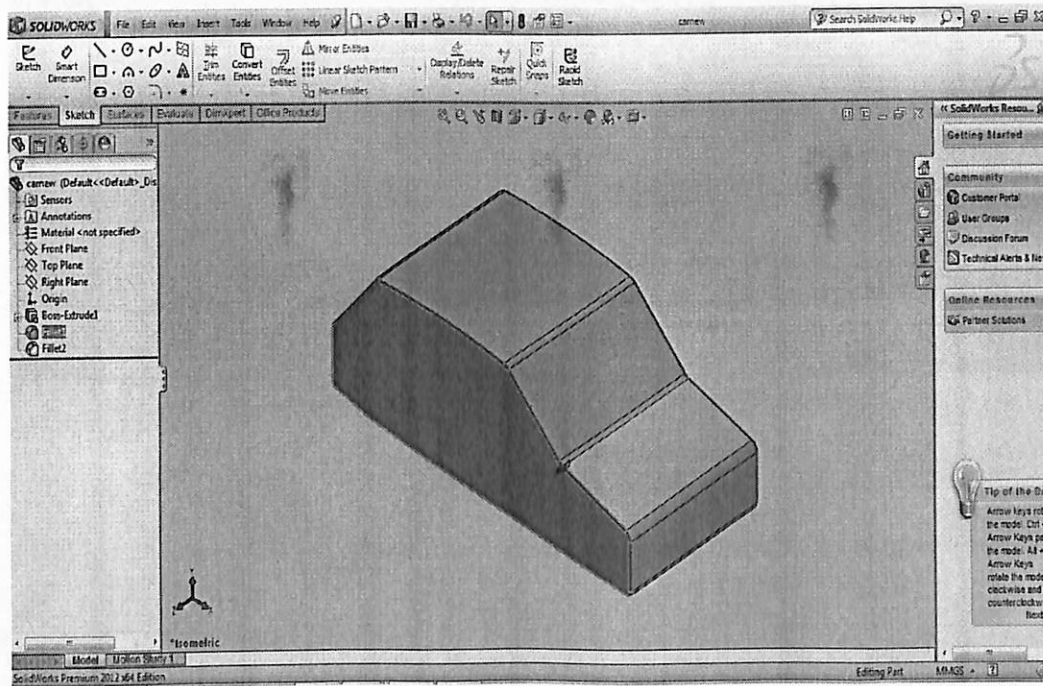


Fig 3.1.3: Fillet command applied on side of the model

3.2 Meshing of the CAD model in GAMBIT

Gambit is used to create and mesh the geometry. We have modelled the car in Solidworks and imported the geometry as IGES file for meshing. These were the general steps followed:

3.2.1 Importing the geometry

- START GAMBIT 2.4.6.
- Import the IGES file.
- Browse the file and click on the Heal Geometry.

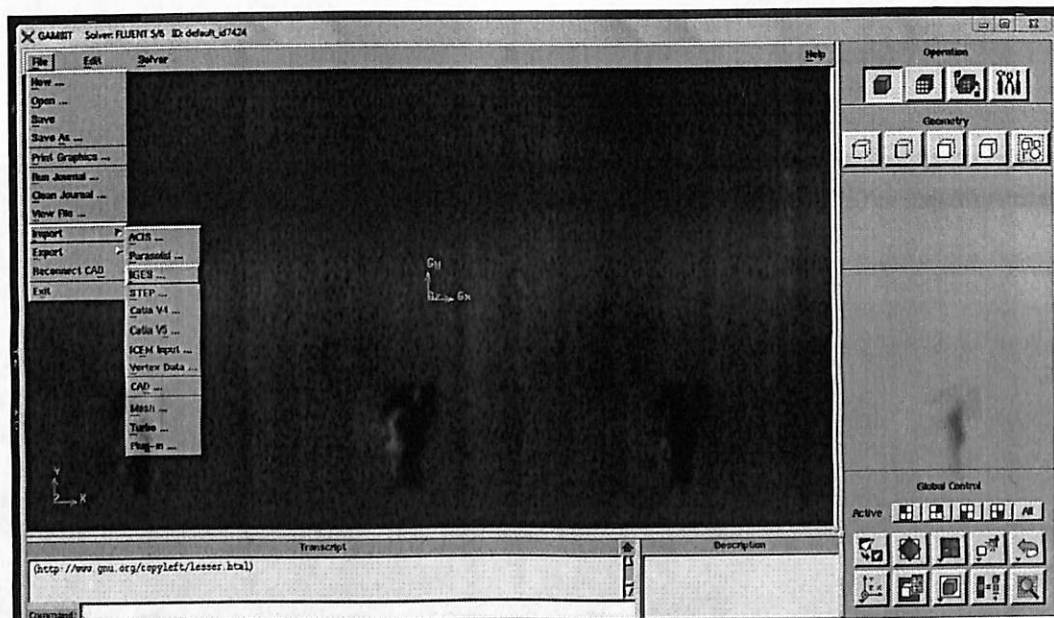


Fig 3.2.1.1: Importing the IGES file

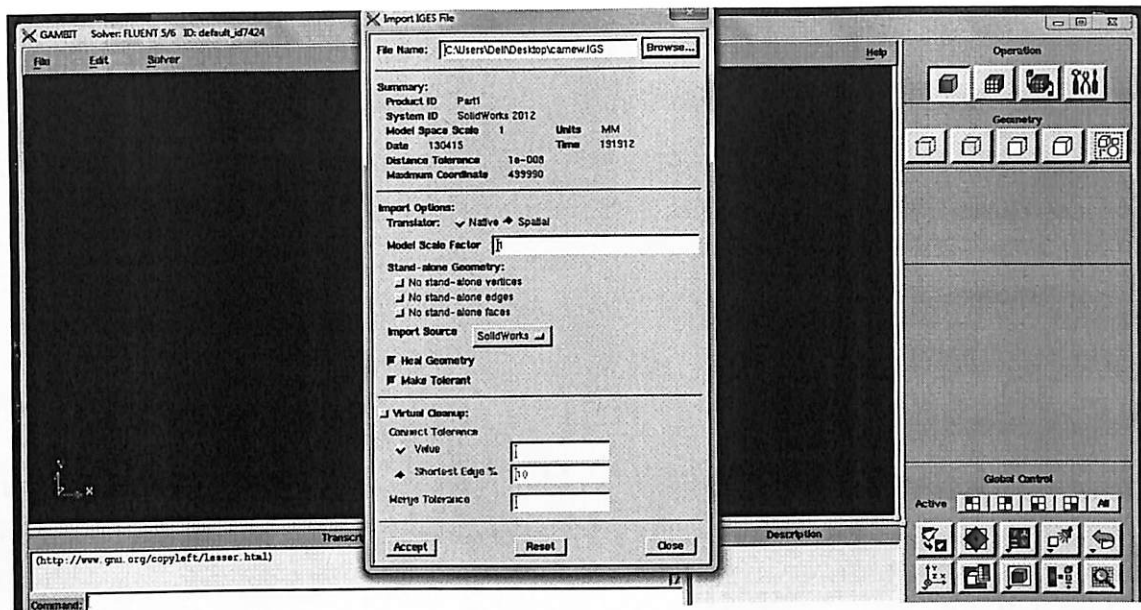


Fig 3.2.1.2: Browsing the file

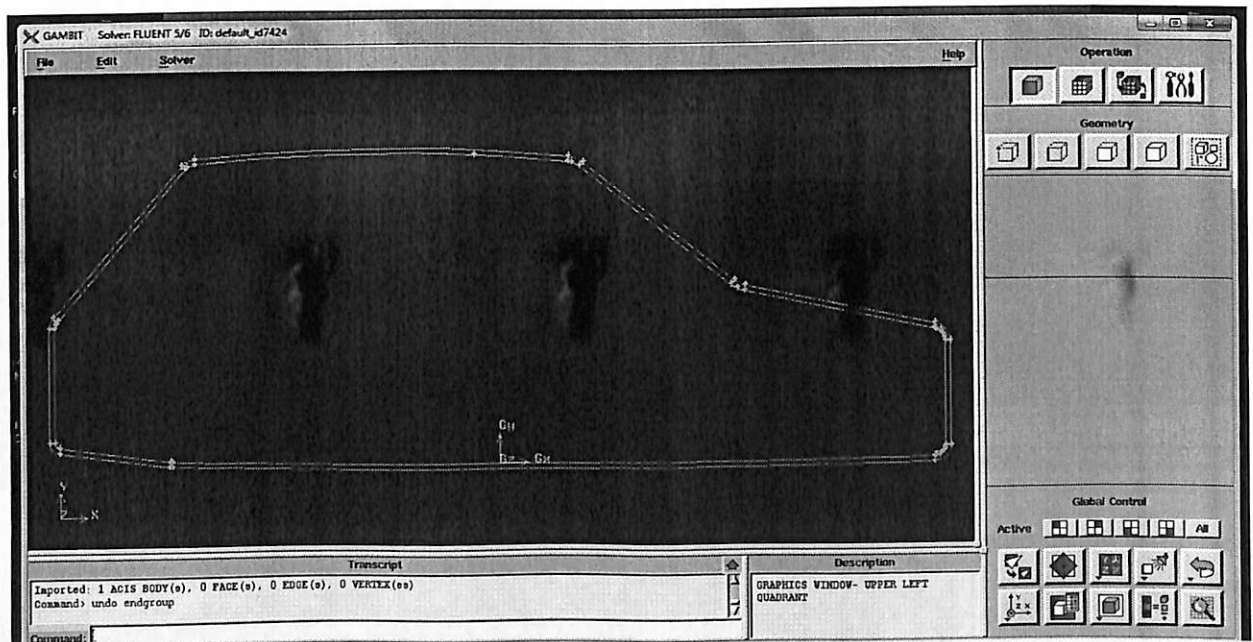


Fig 3.2.1.3: The Imported view of the Geometry

3.2.2 Brick creation

The aim of the brick creation is that it helps in analysis like a virtual wind tunnel with defined boundary conditions.

CLICK ON GEOMETRY → VOLUME → Create Real Brick

Width = 8 * (Length of the model)

Height = 4 * (Height of the model)

Depth = 3 * (Depth of the model)

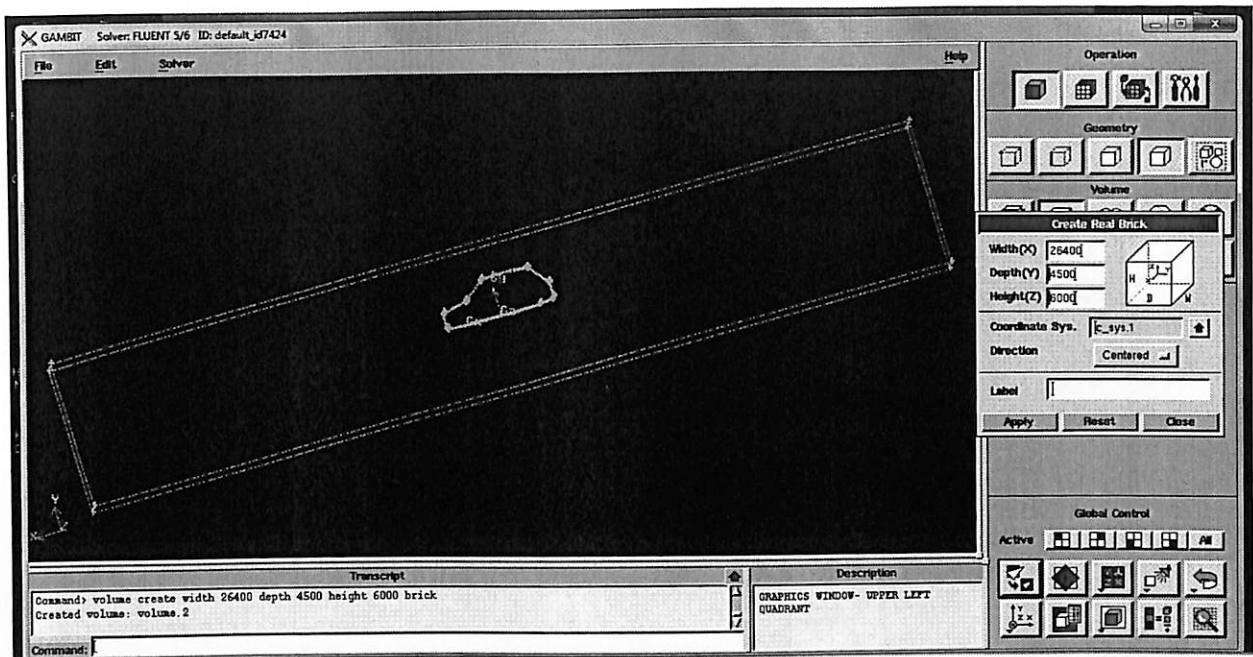


Fig 3.2.2.1.: Brick creation around the Model

3.2.3 Aligning the Model

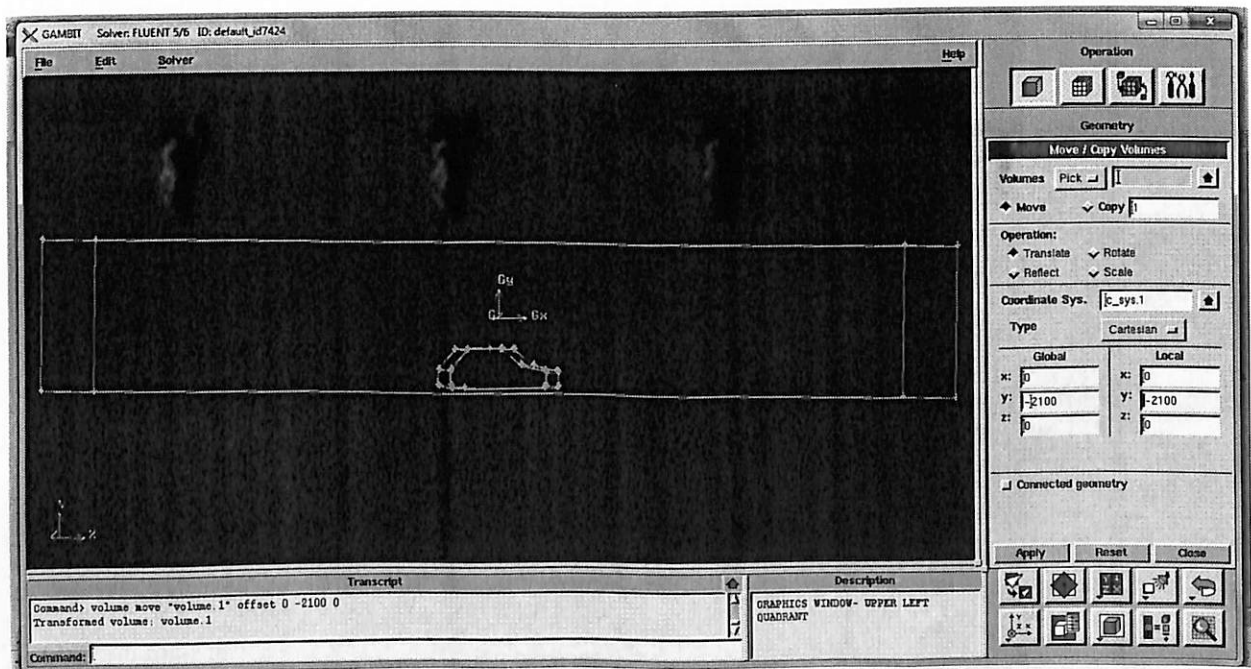


Fig3.2.3.1: Aligning the model to the virtual road surface.

3.2.4 Deleting Volume

GEOMETRY → VOLUME → SUBTRACT VOLUME

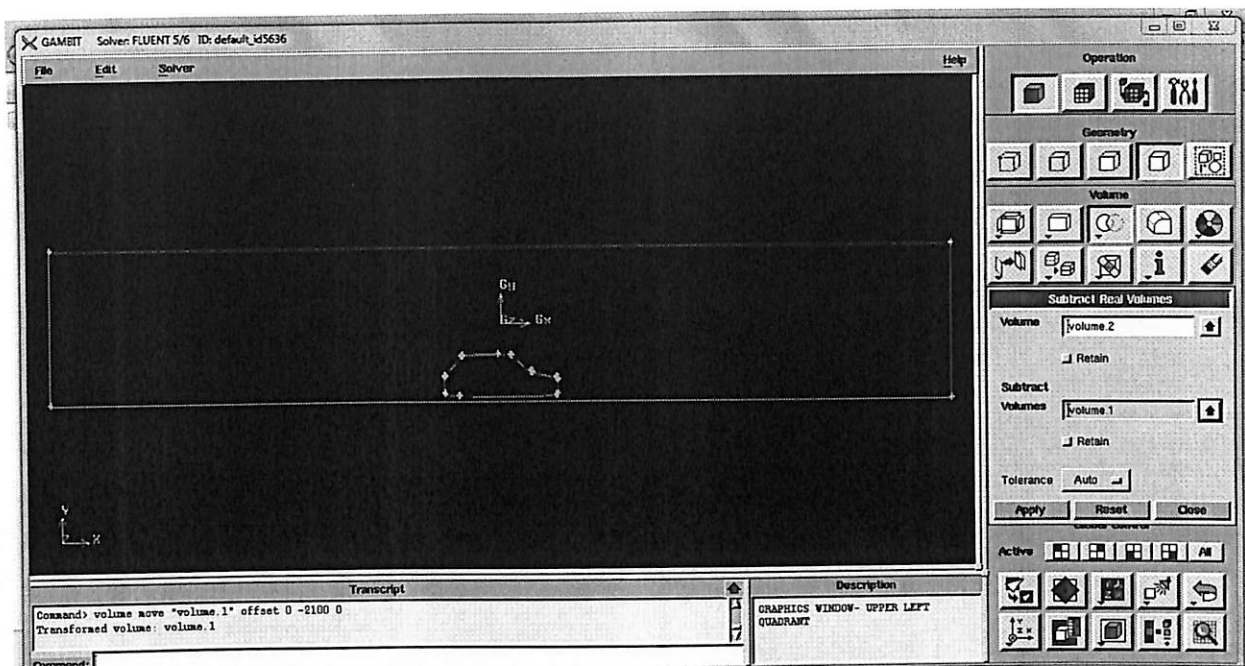


Fig 3.2.4.1: Subtracting the Car Volume from Brick volume.

3.2.5 Meshing the geometry

a) MESH → FACE

Pick all the faces of the car and change the element type to TRI.

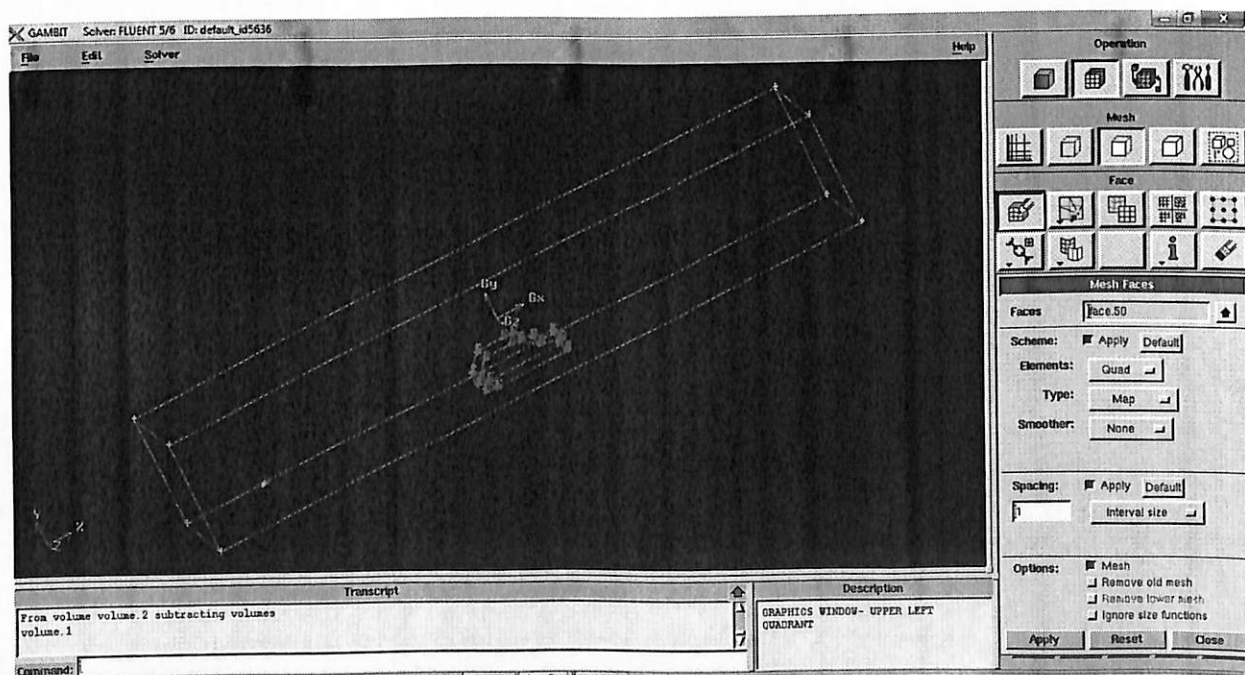


Fig 3.2.5.1: Selecting the Face mesh for the car model by defining the Interval Size

b) MESH → VOLUME

Volume mesh for the volume around the car.

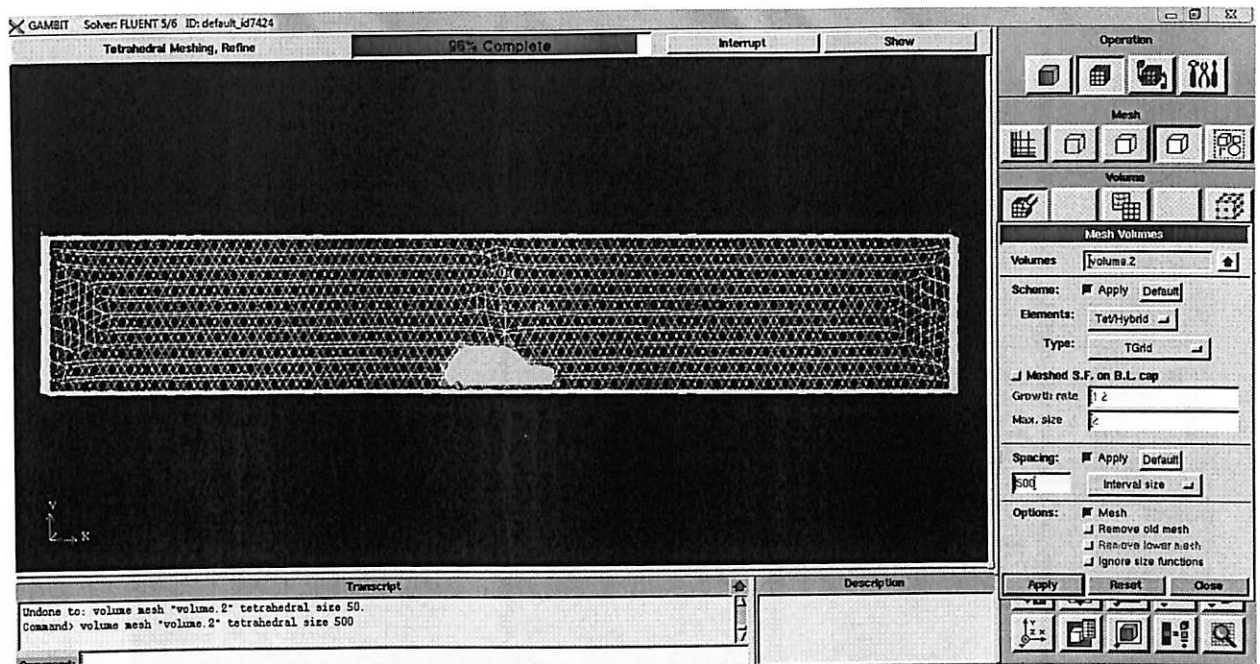


Fig 3.2.5.2: Volume mesh for the remaining brick volume

3.2.6 Boundary conditions

ZONES → Specify Boundary Types

Boundary	Conditions
<i>Inlet</i>	<i>Velocity Inlet</i>
<i>Outlet</i>	<i>Pressure outlet</i>
<i>Vehicle Surface</i>	<i>Stationary Wall</i>
<i>Road</i>	<i>Moving Wall</i>
<i>Top</i>	<i>Symmetry</i>

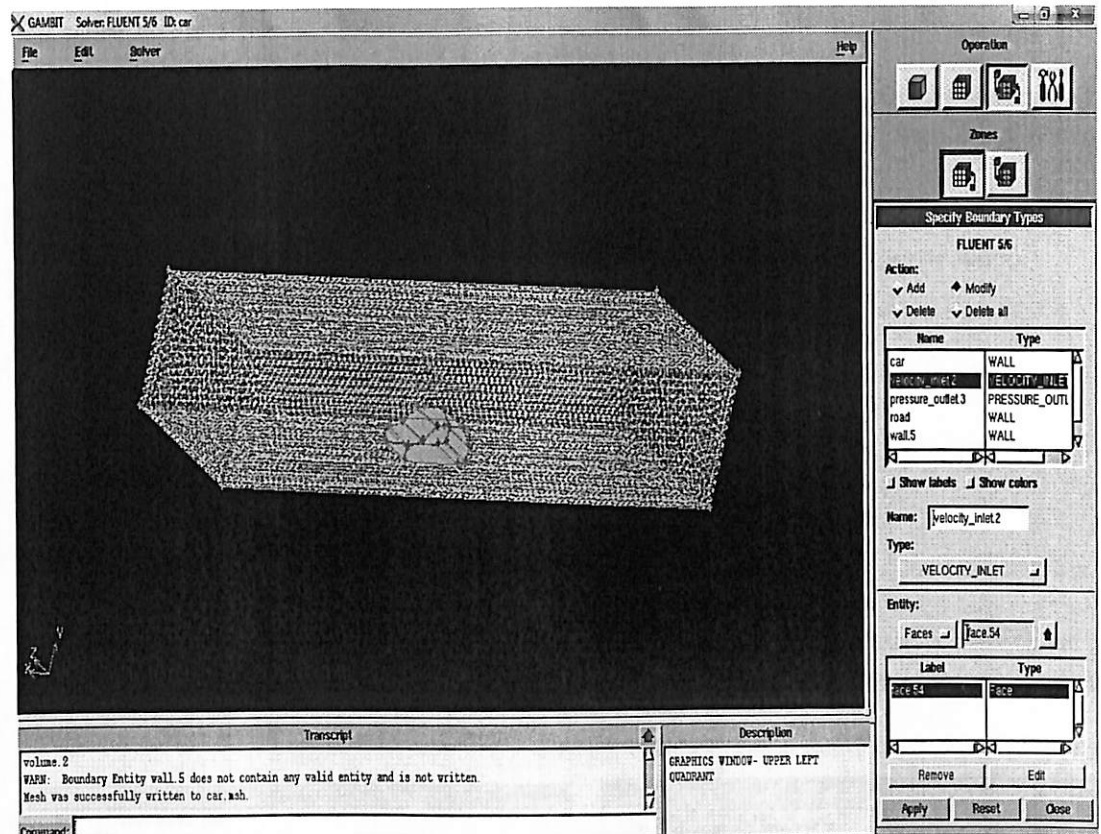


Fig 3.2.6.1: Final created mesh

3.3 Analyzing in Fluent

Fluent is the best CFD tool for the analysis of flow and heat transfer problems especially in process industries. It is a commercial CFD software developed in 'C' language.

The aim is to study the flow of air circulation over a car. A virtual wind tunnel setup can be set to analyze the flow around the vehicle other aspect that demonstrates the CFD advantages are the visualization and the accuracy provided.

The air enters perpendicular to the plane facing the front of the car at the velocity of 25 m/s. The operating pressure is assumed to be 1 atm = 101325 Pa.

The standard $k - \epsilon$ turbulence model is chosen with coefficients $C_\mu = 0.09$, $C_1 - \epsilon = 1.44$, $C_2 - \epsilon = 1.92$ and Prandtl Number = 1.

3.3.1 Reading the file

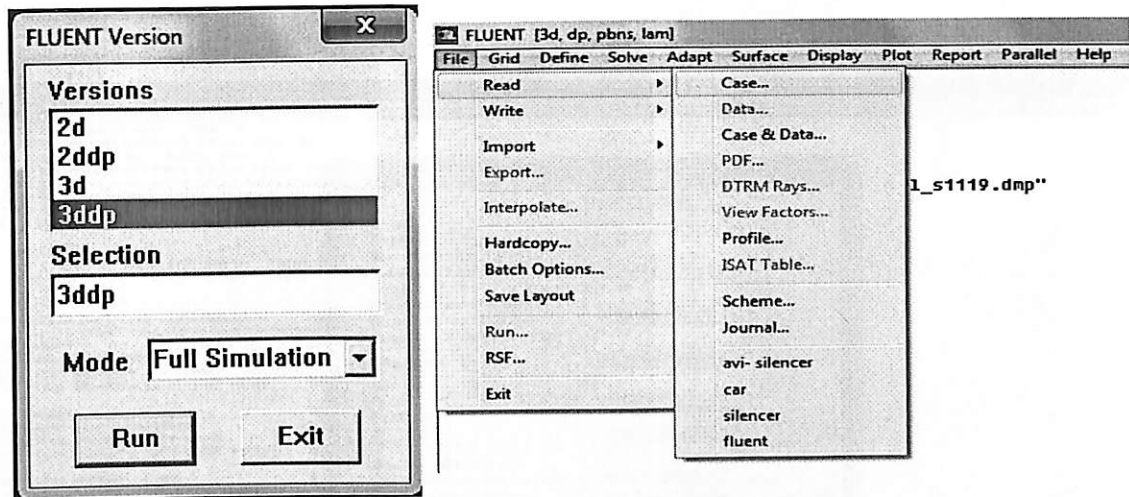


Fig 3.3.1.1: Reading the fill

3.3.2 Checking the grid and scaling the model

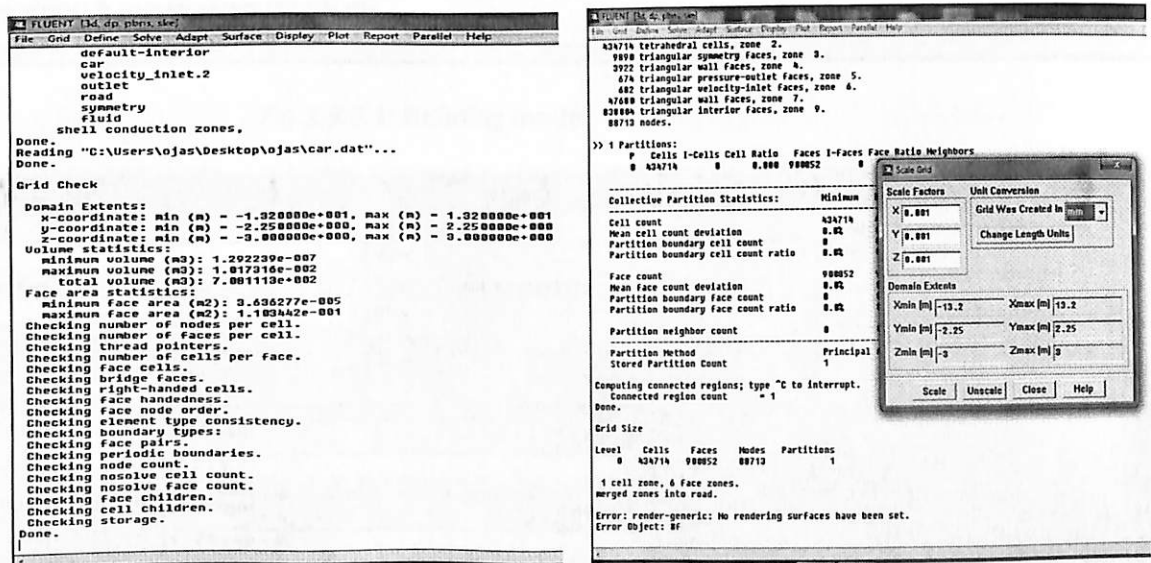


Fig 3.3.2.1: checking grid and scaling

3.3.3 Define Models & Material

For this turbulent exhaust pipe model K-epsilon model equation is selected. The standard k-epsilon model is a semi-empirical model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate. The model transport equation for k is derived from the exact equation. In the derivation of the k-epsilon model, the assumption is that the flow is fully turbulent, & effects of

molecular viscosity are negligible. The standard k-epsilon model is therefore valid for fully turbulent flows.

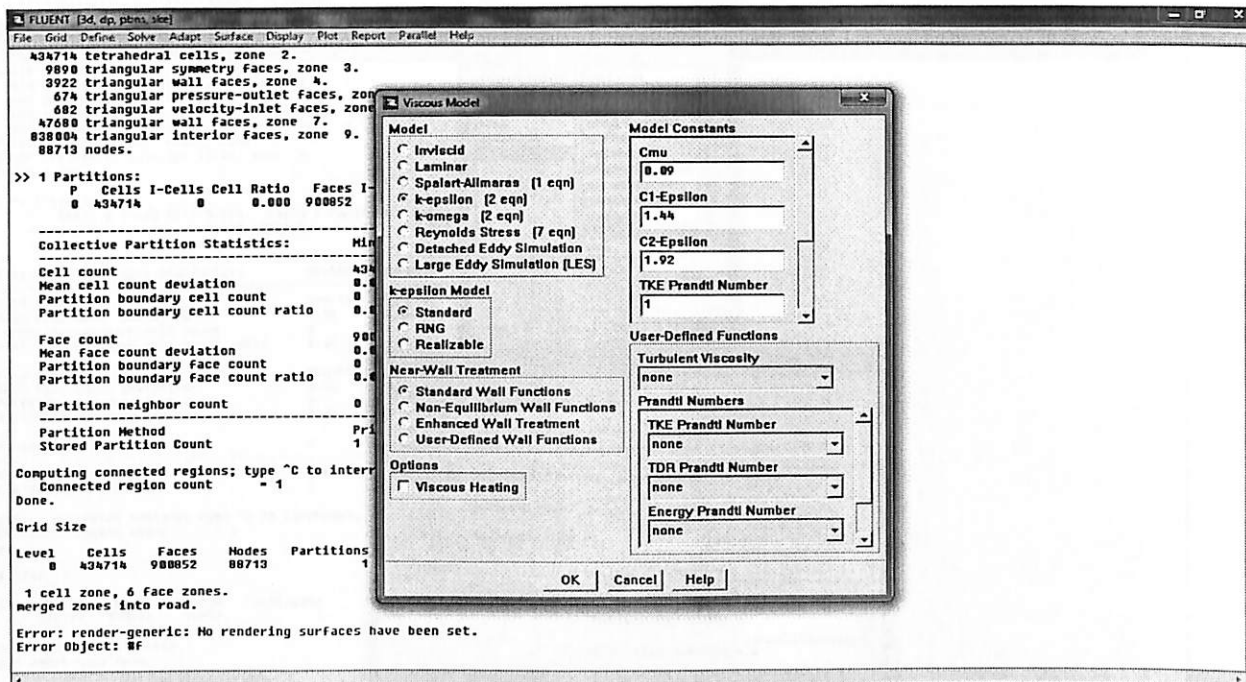


Fig 3.3.3.1: Defining model

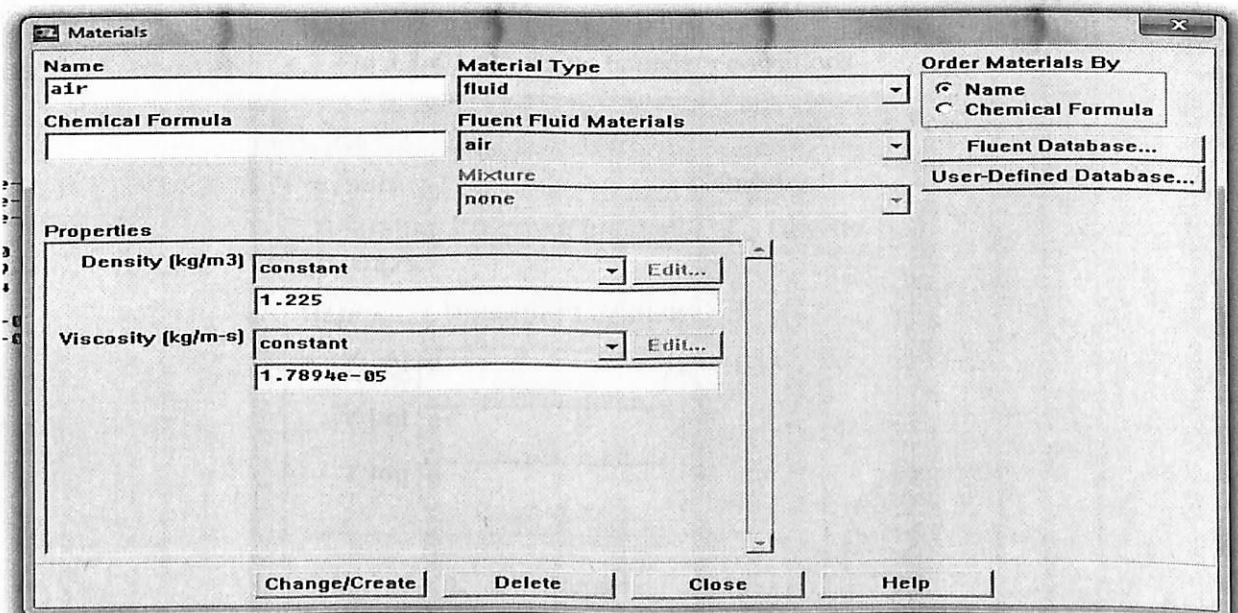
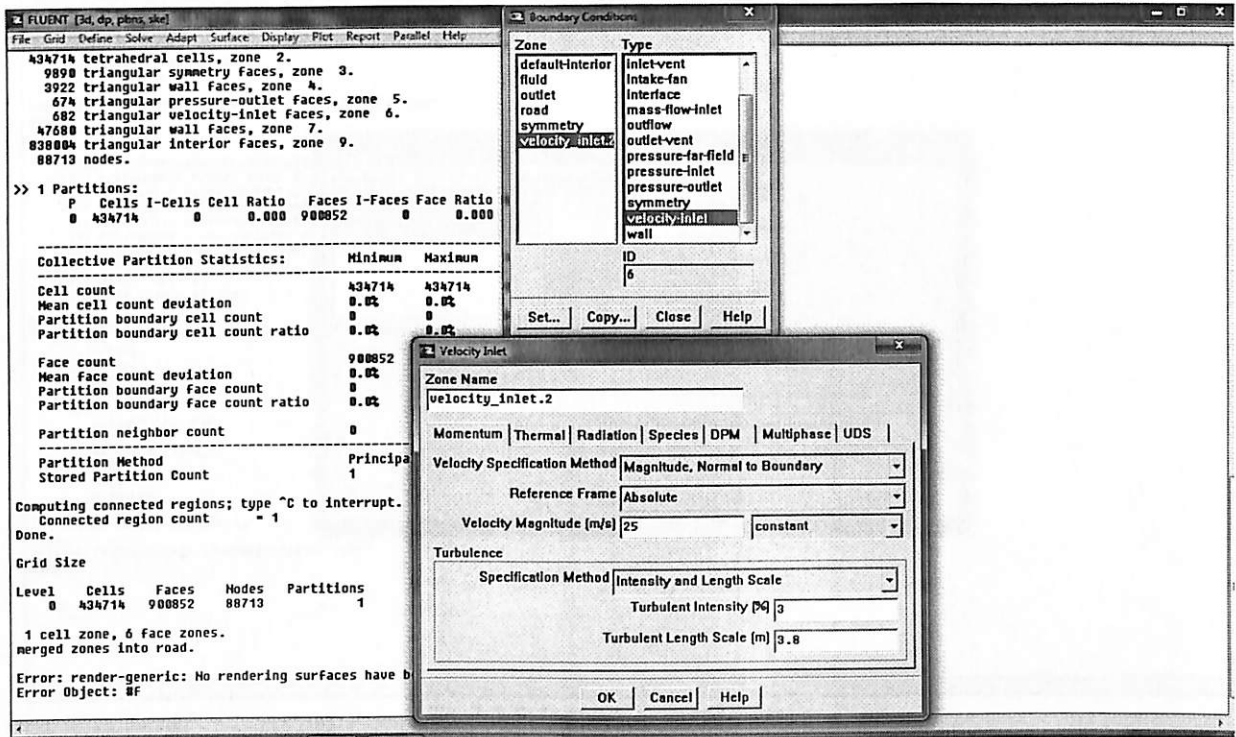


Fig 3.3.3.2: Defining material

3.3.4 Boundary Conditions

Inlet Velocity – 25 m/s

Turbulence Intensity = 3 %



- Fig 3.3.4.1 : Defining boundary conditions

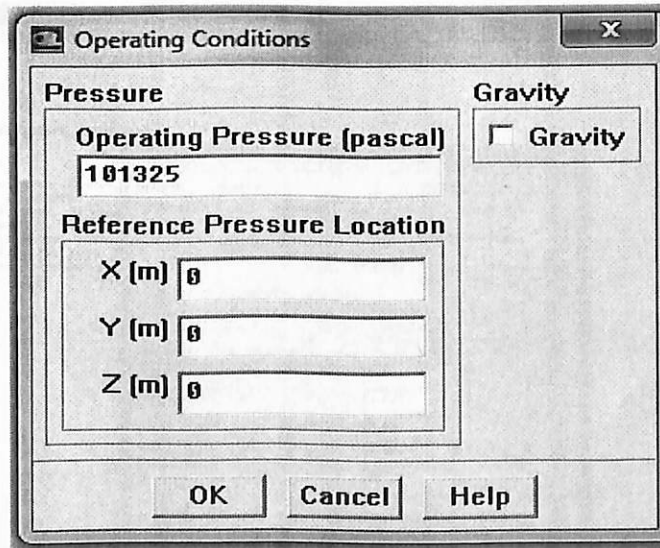


Fig 3.3.4.2: setting operating conditions

3.3.5 Define Solution Parameter

Solve → Control → Solution

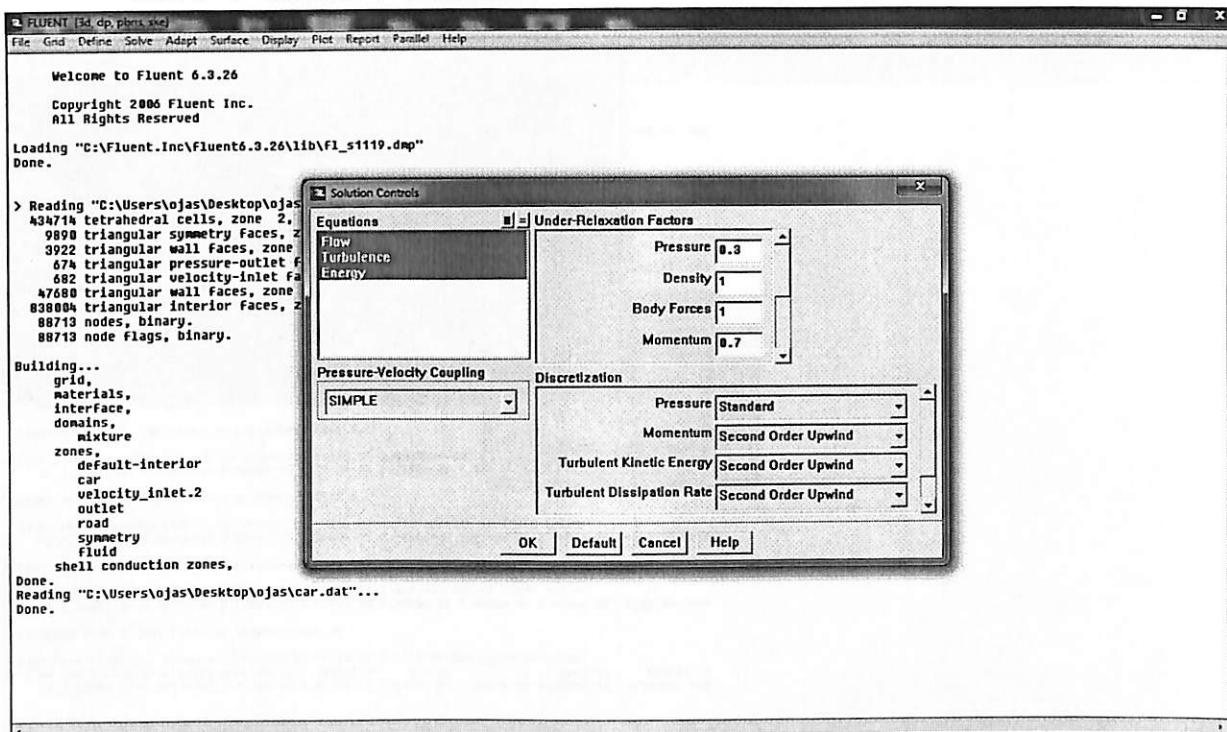


Fig 3.3.5.1: solution parameters

3.3.6 Enable The Plotting of Residual

Solve → Monitor → Residuals

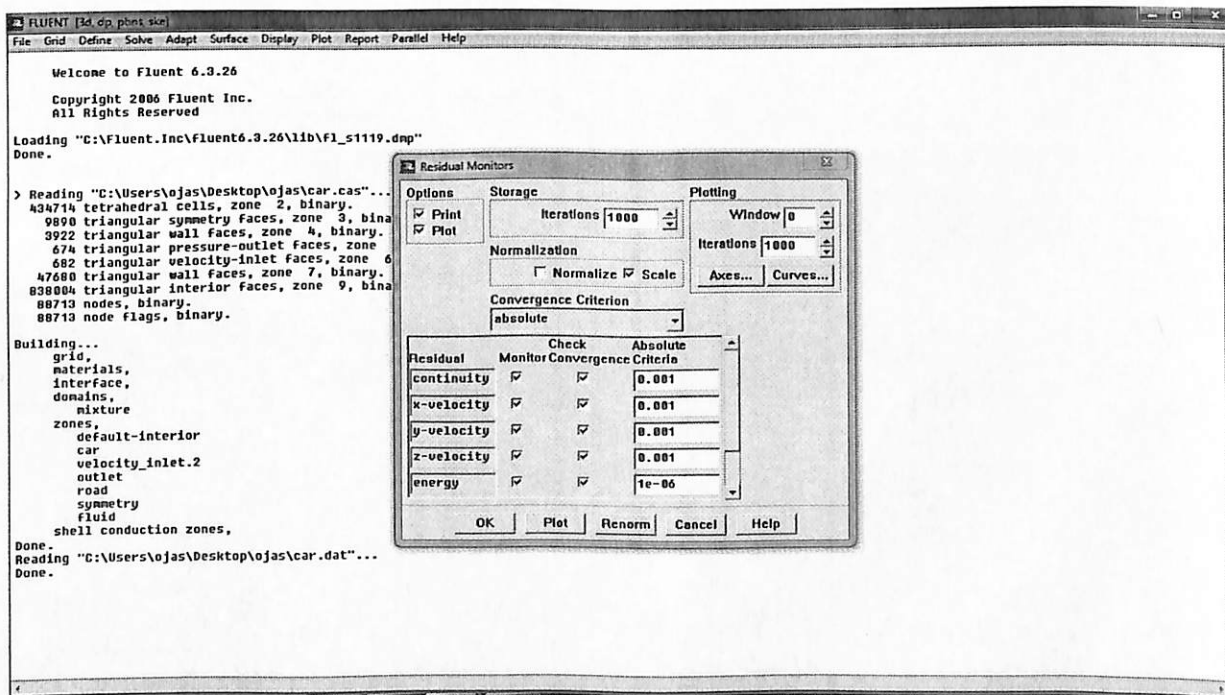


Fig 3.3.6.1: Plotting residue

3.3.7 Start the iteration.

Solve → Iterate

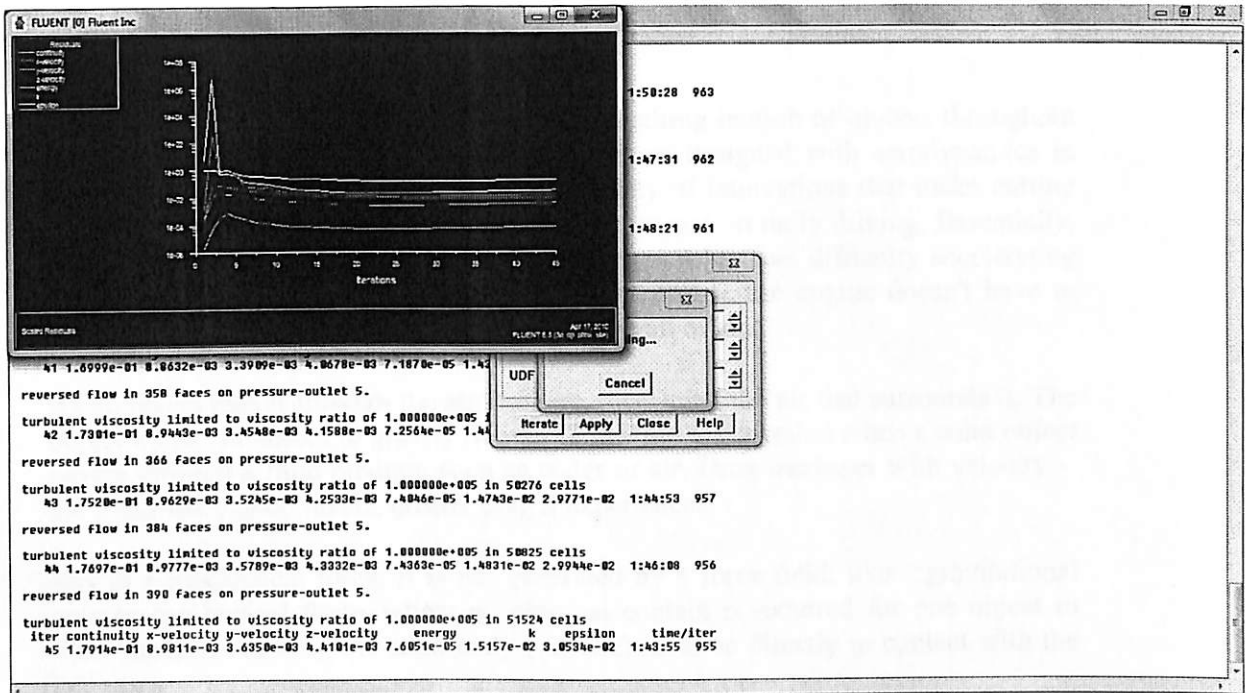


Fig 3.3.7.1: Starting iteration

4. LITERATURE REVIEW

4.1. Automobile Aerodynamics

Aerodynamics is the study of forces and the resulting motion of objects throughout the air. For a number of decades, cars have been designed with aerodynamics in mind, & carmakers are coming up with a variety of innovations that make cutting through that "wall" of air easier and less of an impact on daily driving. Essentially, having a car designed with airflow in mind means it has less difficulty accelerating and can achieve better fuel economy numbers because the engine doesn't have to work nearly as hard to push the car through the wall of air.

As an object moves through the atmosphere, displacing the air that surrounds it. The object is also subjected to gravity & drag. The Drag is generated when a solid object moves through a fluid medium such as water or air. Drag increases with velocity -- the faster the object travels, greater drag it experiences.

Drag is a mechanical force. It is not generated by a force field, like gravitational force or mechanical force, where no physical contact is required for one object to affect another. For drag the solid body (vehicle) must be directly in contact with the fluid (air).

For a vehicle moving in a straight line on horizontal ground; the air flow is dependent on the vehicle speed and the ambient wind. The wind has a non-uniform velocity profile because of the local topography and the earth boundary layer, and in general the velocity will fluctuate in both magnitude and direction. The aerodynamic forces and moments will act at the centre of pressure, but unlike the centre of gravity, the centre of pressure is not fixed and it depends on the air flow, the centre of pressure will tend to move forward at high velocities.

Since the vehicle and air velocity are not co-linear, there will be a yaw angle and a resultant side force. Also there is a lift force, which is a result of the asymmetrical flow above and below the vehicle, an effect that will evidently be influenced very strongly by the presence of the ground and the angle of incidence.

The drag and lift characteristics of a body are described by the dimensionless drag and lift coefficients, C_d and C_l . These are defined by the following equations:

$$\text{Drag, } D = \frac{1}{2} \rho v^2 A C_d$$

$$\text{Lift, } L = \frac{1}{2} \rho v^2 A C_l$$

Where,

ρ = air density

v = relative velocity or air velocity

A = vehicle frontal area

In order to predict the drag on an object, one needs to correctly predict the pressure field and the surface shear stress. This, in turn, requires the treatment and prediction of boundary layers and flow separation.

The aerodynamic abilities of an automobile are measured using the vehicle's coefficient of drag. Essentially, lower the C_d , more aerodynamic a car profile is, and the easier it can move through the wall of air pushing against it.

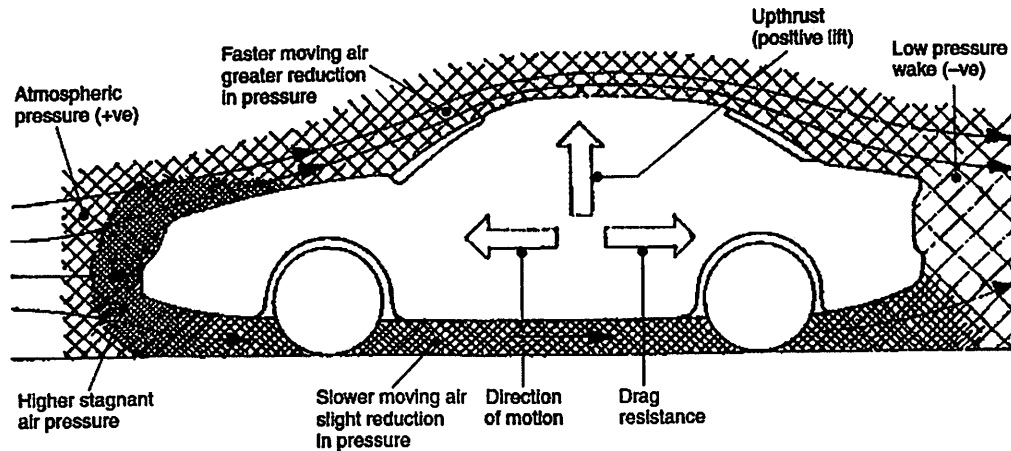


Fig 4.1.1: Pressure distribution on a car

4.1.1. Viscous Boundary layer

A laminar flow will be affected by a presence of restriction such as a wall. Most of the flow will be unaffected by the presence of the plate, except in the region closest to the wall, the velocity will decrease to zero. The flow away from the walls can be treated as inviscid, and sometimes may be approximated as potential flow. The region near the wall where the viscous forces are of the same order as the inertial forces is termed the boundary layer. The distance over which the effect of viscous forces is present is known as the boundary layer thickness. The thickness is a function of Reynolds number, as Re increases the thickness shall decrease.

Now consider a flat plate of length ' L ', infinite width, and negligible thickness, which lies in the x - z plane, and whose two edges correspond to $x=0$ and $x=L$. Assume that the plate is immersed in a low viscosity fluid whose unperturbed velocity field is $\mathbf{v}=U_0\mathbf{e}_x$. In the inviscid limit, the appropriate boundary condition at the surface of the plate, $v_y=0$ corresponding to the requirement of zero normal velocity--is already satisfied by the unperturbed flow. Thus the original flow is not modified by the presence of the plate. But, when we take the finite viscosity of the fluid into account, an additional boundary condition, $v_x=0$ corresponding to the no slip condition--must be satisfied at the plate. This additional constraint causes thin boundary layers, of thickness $\delta(x) \ll L$, to form above and beneath the plate. The fluid flow outside the boundary layers remains effectively inviscid, while that inside the layers is modified by viscosity.

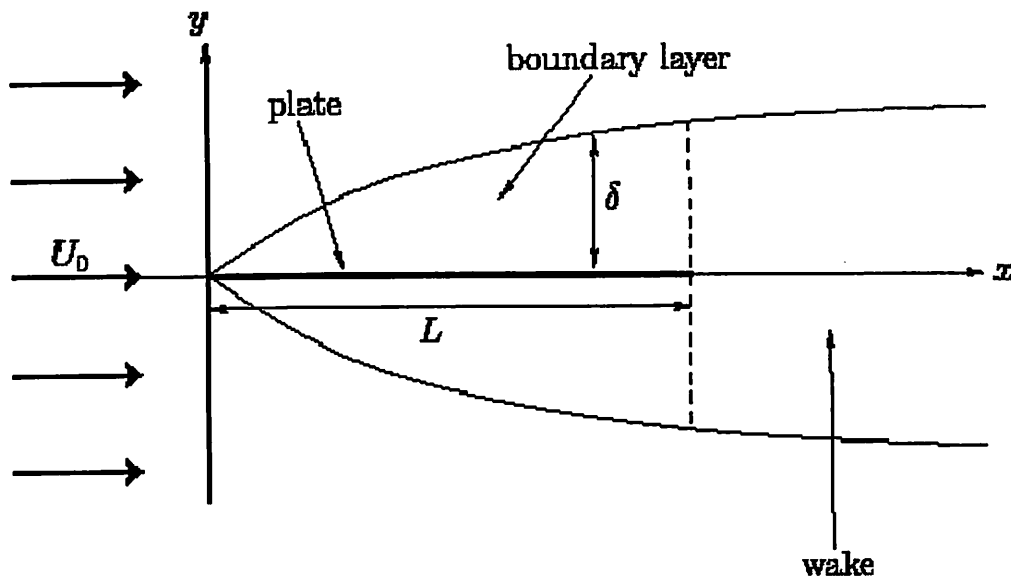


Fig 4.1.1.1: Boundary layer on flat plate

4.1.2. Boundary layer separation

When a high Reynolds number fluid shall pass around a streamlined obstacle, such as a slender plate which is aligned with the flow, a relatively thin boundary layer forms on the obstacle's surface.

Suppose, however, that the obstacle is not streamlined, that is the surface of the obstacle is not closely aligned with the streamlines of the unperturbed flow pattern. In this case, the typically observed behaviour, is shown in the figure, the flow pattern of a high Reynolds number irrotational fluid around a cylindrical obstacle (whose axis is normal to the direction of the unperturbed flow). It can be seen that a *stagnation pt.*, at which the flow velocity is locally zero, formed in front of the obstacle. Also, a thin boundary layer covers the front side of the obstacle. The thickness of this layer is found to be smallest at the stagnation pt., and increases towards the back side of the obstacle. Though, at some point on back side, boundary layer separates from the obstacle's surface to form a vortex-filled wake whose transverse dimensions are similar to those of the obstacle itself. This phenomenon is called *boundary layer separation*. Outside the boundary layer, and the wake, it is observed that the flow pattern is irrotational and fundamentally inviscid.

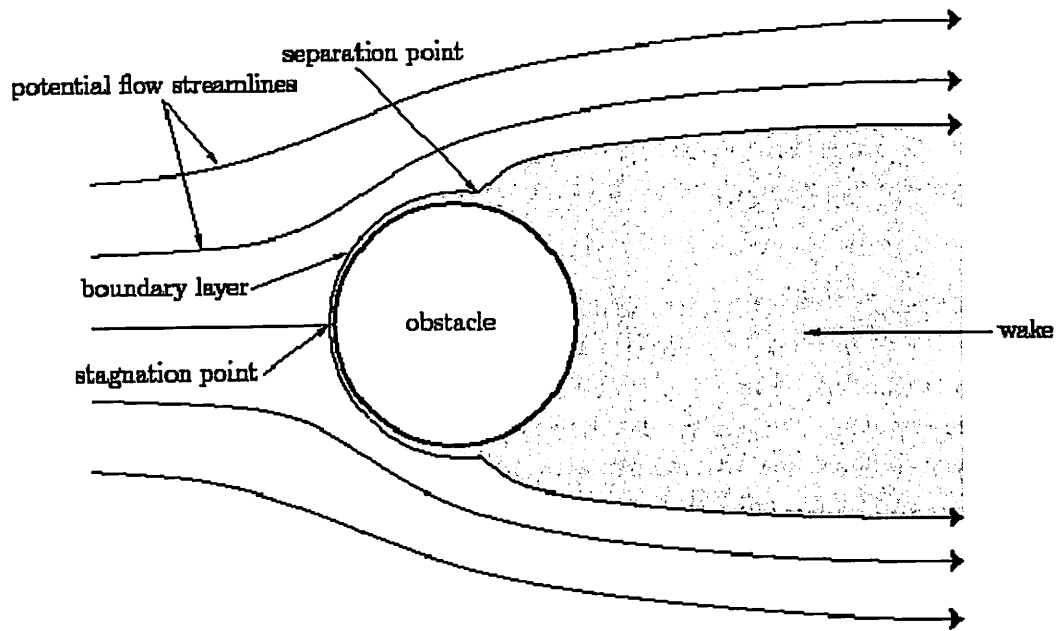


Fig 4.1.2.1: Boundary layer separation

To measure the aerodynamic effectiveness of a car in actual conditions, engineers make use of the wind tunnel a concept borrowed from the aircraft industry. A wind tunnel is essentially a massive tube with fans that produce airflow over an object inside. The fans create wind at different speeds to simulate real-world conditions, while the object inside never really move. . Computers are used to calculate the drag coefficient (C_d), as wind moves over the car in the tunnel. With advancement in modern computing technology in recent years computer simulations are starting to replace wind tunnels to measure the aerodynamics of a car or aircraft.

4.2.CFD (Computational Fluid Dynamic)

CFD is one of the branches of fluid mechanics that makes use of numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform millions of calculations required to simulate the intersection of fluids and gases with the complex surfaces used in engineering.

4.2.1 Discretization methods in CFD

There are 3 discretization methods in CFD:

1. Finite difference method (FDM)
2. Finite volume method (FVM)
3. Finite element method (FEM)

Finite difference method (FDM) : A finite difference method discretization is based upon the differential form of the PDE to be solved. Each derivative gets replaced with an approximate difference formula (that can generally be derived from a Taylor series expansion). The computational domain is generally divided into hexahedral

cells (the grid) and the solution will be obtained at each nodal point. The FDM is easiest to understand when the physical grid is Cartesian but through the use of curvilinear transforms the method can be extended to domains that are not easily brick-shaped elements. The discretization gives a system of equations of the variable at nodal points, & once the solution is found, then we have a discrete representation of the solution.

Finite Volume Method (FVM): A finite volume method (FVM) discretization is based upon an integral form of the PDE to be solved (e.g. conservation of energy, momentum, or mass). PDE is written in a form which can be solved for a given volume (or cell). The computational domain is discretized into finite volumes and then for every volume the 12 governing equations are solved. The resulting equations usually involve fluxes of the conserved variable and thus the calculation of fluxes is very important. The basic advantage of this method over FDM is it does not require the use of structured grids and the effort to convert the given mesh into a structured numerical grid internally is completely avoided.

Finite Element Method (FEM): A finite element method (FEM) discretization is based upon a piecewise representation of the solution in terms of specified basis functions. The computational domain is divided up into smaller domains (finite elements) and the solution in each element is constructed from the basis function. The actual equation that is solved in each element is constructed from the basis functions. The actual equations which are solved in each element are constructed from the basis functions. The actual equations which are solved are typically obtained by restating the conservation equation in weak form; the field variables are written in terms of the basis functions, the equation is multiplied by appropriate test function and then integrated over an element. Since the FEM solution is in terms of specific basis functions, a great deal more is known about the solution than for either FDM or FVM. This can be a double-edged sword as the choice of basis function is very important and boundary conditions may be more difficult to formulate. Again a set of equations is obtained that must be solved to obtain a solution.

Comparison of the three methods is difficult, primarily due to many variations of all three methods. FVM & FDM provide distinct solutions, while FEM provides a continuous (up to a point) solution. FVM & FDM are usually considered easier to provide better conservation properties but opinions vary on this point also.

4.2.2 How Does CFD code work?

CFD codes are structured around the numerical algorithm that can be used to tackle fluid problems. To provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces to input problem parameters and to examine the result. Hence all codes contain three main elements:

1. Pre-processing
2. Solver
3. Post-processing

Pre-Processing:

This is the first step in building and analyzing a flow model. Pre-processor consists of input of a flow problem by means of an operator friendly interface and subsequent transformation of this input into form of suitable for the use by the solver. The user actions at the pre-processing stage involve:

- Definition of the geometry of the region.
- Generating grids by division of domain into a no. of smaller non-overlapping sub domains (or control volumes or elements selection of physical or chemical phenomena that need to be modelled).
- Definition of fluid properties
- Specification of appropriate boundary conditions at cells which coincide with or touch the boundary. The solution of a flow problem (velocity, pressure, temperature, etc) is defined at nodes inside each cell. The precision of CFD solutions is governed by number of cells in the grid. In general greater numbers of cells better the solution accuracy. Both the accuracy of the solution and its cost in terms of necessary computer hardware and calculations time are dependent on the fitness of the grid.

GAMBIT (CFD Pre-processor): GAMBIT is a state of the art pre-processor for engineering analysis. With advanced geometry & meshing tools in a powerful, flexible and easy to use interface, GAMBIT can dramatically reduce preprocessing times for many applications. Complex models can be made directly within GAMBIT's solid geometry modeller or imported from major CAD/CAE systems. Using a virtual geometry overlay and advanced cleanup tools, imported geometries are quickly converted into suitable flow domains. A widespread set of highly-automated and size function driven meshing tools ensures that the best mesh can be generated whether structures, multiblock, unstructured or hybrid.

Solver:

The CFD solver does the flow calculations and produces the result. FLUENT, FloWizard, FIDAP, CFX and POLYLOW are some of the types of solvers. FLUENT is used in most industries. FloWizard is the most general-purpose rapid flow modelling tool for design and process engineers built by Fluent. POLYFLOW (and FIDAP) are also used in a broad range of fields, along with emphasis on the materials processing industries. Fluent and CFX 2 solvers were developed independently by ANSYS and have a number of things in common, but they also have important differences. Both are control volume based to get high accuracy and rely heavily on a pressure based solution for broad applicability. They differ mainly in the ways they integrate the fluid flow equation and in their equation solution strategies. The CFX solver uses finite elements (cell vertex numeric's), similar to those used in mechanical study to discretize the domain. In contrast, the FLUENT solver uses finite volume (cell centered numeric). CFX software focuses on 1 approach to solve the governing equation of motion (coupled algebraic poly grid), while the Fluent product offers several solution approaches (density, segregated and coupled pressure based methods).

The FLUENT CFD code has extensive interactivity, so we can changes to the analysis at any time during the process. This saves time and enables to filter designs more efficiently. Graphical user interface (GUI) is intuitive, which helps to cut down the learning curve and make the modelling process faster. In addition, FLUENT's adaptive and vibrant mesh capability is unique and works with a wide range of physical models. This potential makes it possible and simple to model complex moving objects in relation to flow. It provides the broadest range of rigorous physical models that have been validated against multiphase, reacting flows, rotary equipments, moving and deforming objects, turbulence, radiation, acoustics and dynamic meshing. The FLUENT solver has constantly proven to be fast and reliable for a wide range of CFD applications. The speed to solution is quicker because suite of software enables us to stay within one interface from geometry building through the solution process to post-processing and final output.

Post-Processing:

This is the final step in CFD analysis and it involves the organization and interpretation of the predicted flow data and the production of CFD images and animations. FLUENT's software includes full post processing capabilities. Fluent exports CFD's data to third-party post-processors and visualization tools such as Ensight, Field view and TechPlot. In addition, FLUENT CFD solutions are easily coupled with structural codes as well as to other engineering process simulation tools. Thus FLUENT is general purpose CFD software ideally used for incompressible and mildly compressible flows. Using the pressure based segregated finite-volume method solver, FLUENT contains physical models for a wide range of applications including turbulent flows, heat transfer, reacting flows, chemical mixing, combustion and multiphase flows.

4.2.3 Advantages of CFD:

Over the past few decades, CFD has been used to improve process design by allowing engineers to simulate the performance of alternative configuration, eliminating guesswork that would normally be use to establish equipment geometry and process conditions.

Advantages of CFD can be summarized as:

- It provides the flexibility to change design parameters without the expense of hardware changes therefore costs less than laboratory or field experiments, allowing engineers to try more options of designs than would be feasible otherwise.
- It has a faster turnaround time experiments.
- It guides engineer to the root of problem and is therefore well suited for trouble shooting.

- It provides inclusive information about the flow field, especially in regions where measurement are either difficult or impossible to obtain.
- CFD solutions are reliable can very much depict the real flow conditions. The numerical methods embedded in FLUENT solver are improving day by day. The results too obtained are consistent with experimental results.
- CFD allows an engineer to analyze and examine data at numerous locations in the model.

4.2.4 Limitations of CFD:

In spite of large advantages of CFD, it has some limitations which are as follows:

Physical Models- CFD solutions rely upon physical models of real world processes (e.g. turbulence, compressibility, chemistry, etc.) The CFD solutions will only be as accurate as the physical models are.

Numerical errors- Solving equations on a computer invariably introduces numerical errors. Round –off error is due to finite word available on the computer. Round-off errors can be always found in CFD solution but they may be very small in most cases. Second most common error found in CFD solution is truncation error, due to approximations in numerical models.

Boundary conditions- The CFD solution will only be as good as the initial/boundary conditions provided to the numerical model.

5. RESULTS AND DISCUSSIONS

5.1 Static Pressure Plot

The red area represents the high pressure at the front end.

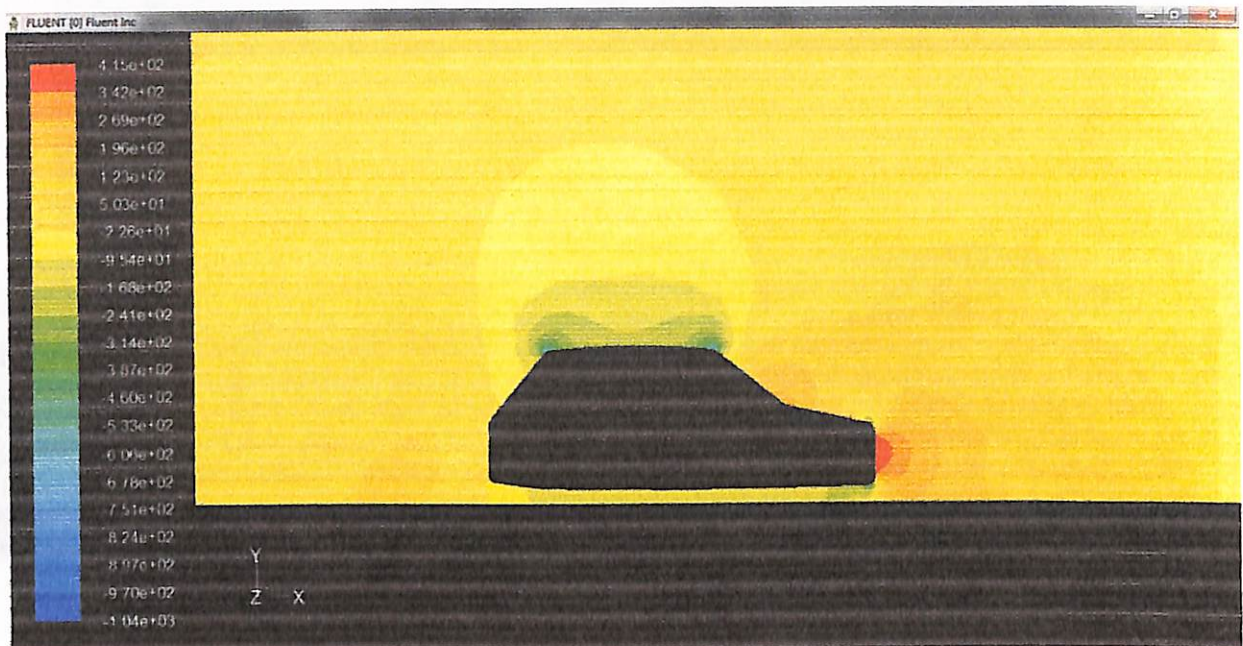


Fig5.1.1: Contours of Static Pressure

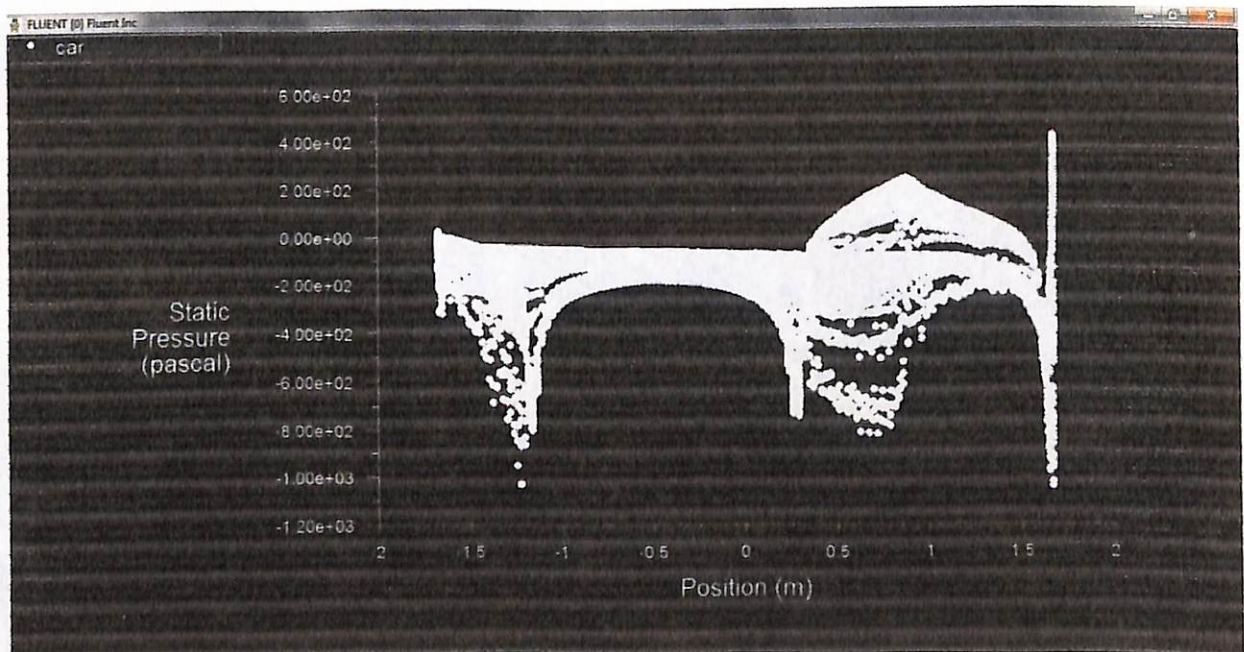


Fig 5.1.2: Static pressure vs. Car Length

5.2 Velocity Magnitude



Fig5.2.1: Contours of velocity magnitude

5.3 Turbulent Kinetic Energy

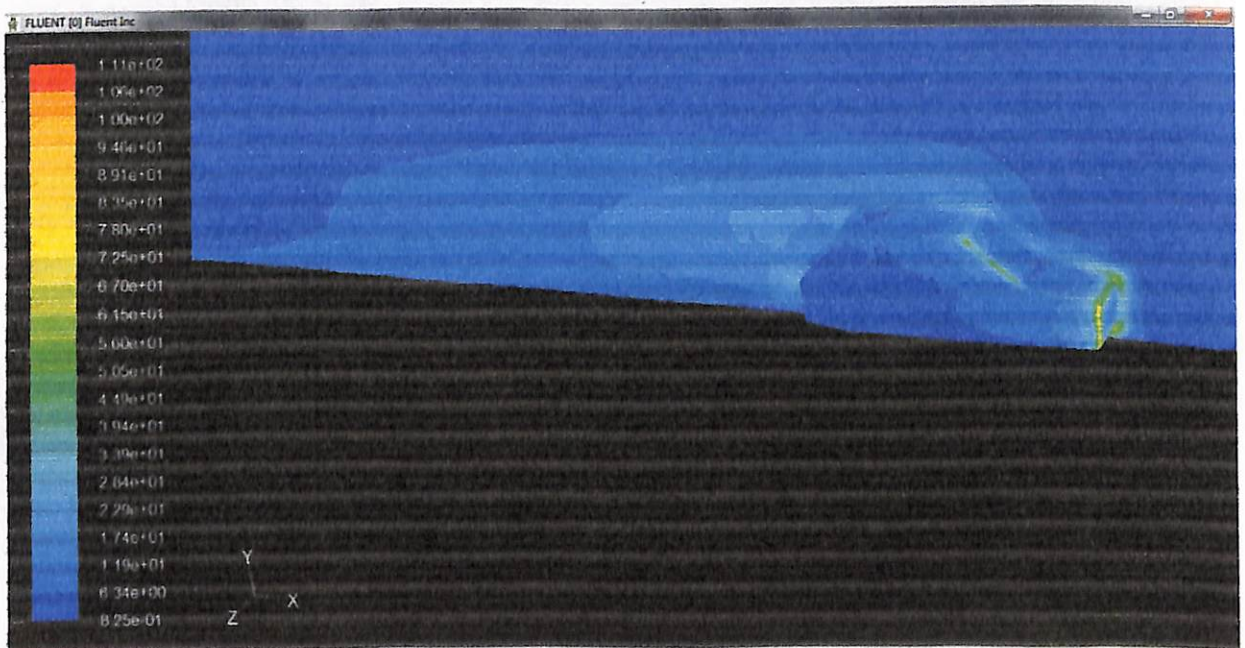


Fig5.3.1: Contours of Turbulent Kinetic Energy

5.4 Total Pressure Plot

The total pressure is a sum of static pressure & dynamic pressure.

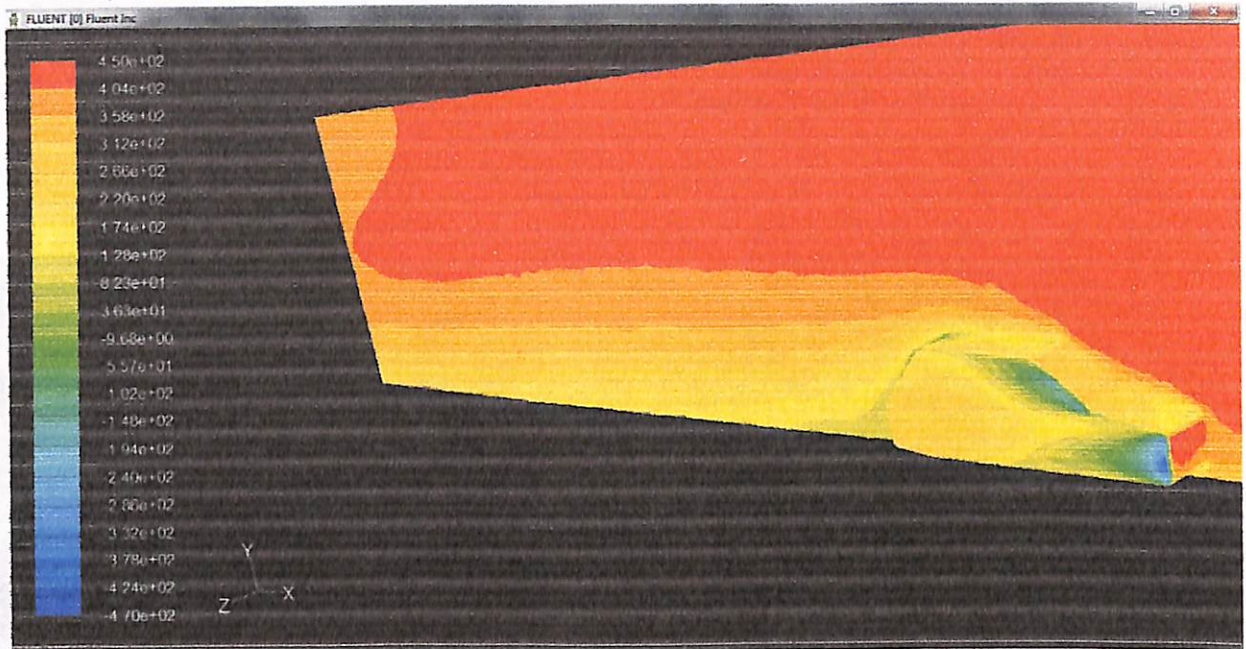


Fig5.4.1: Contours of total pressure

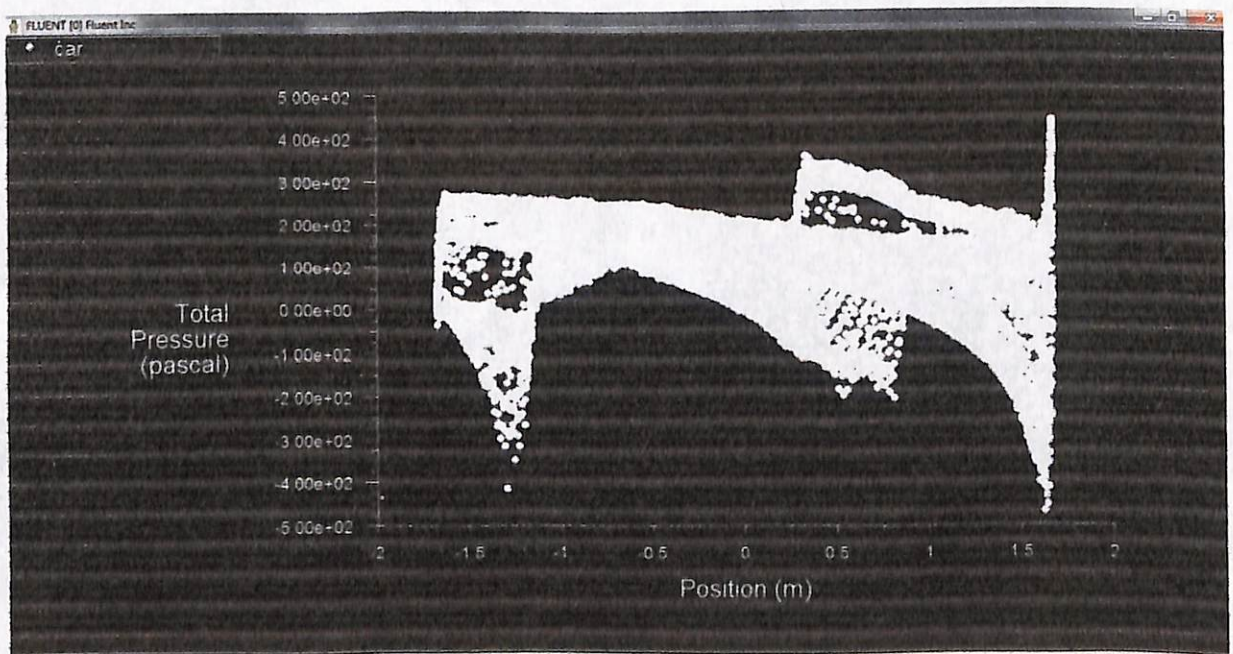


Fig5.4.2: Total Pressure vs. Car Length

5.5 Streamline

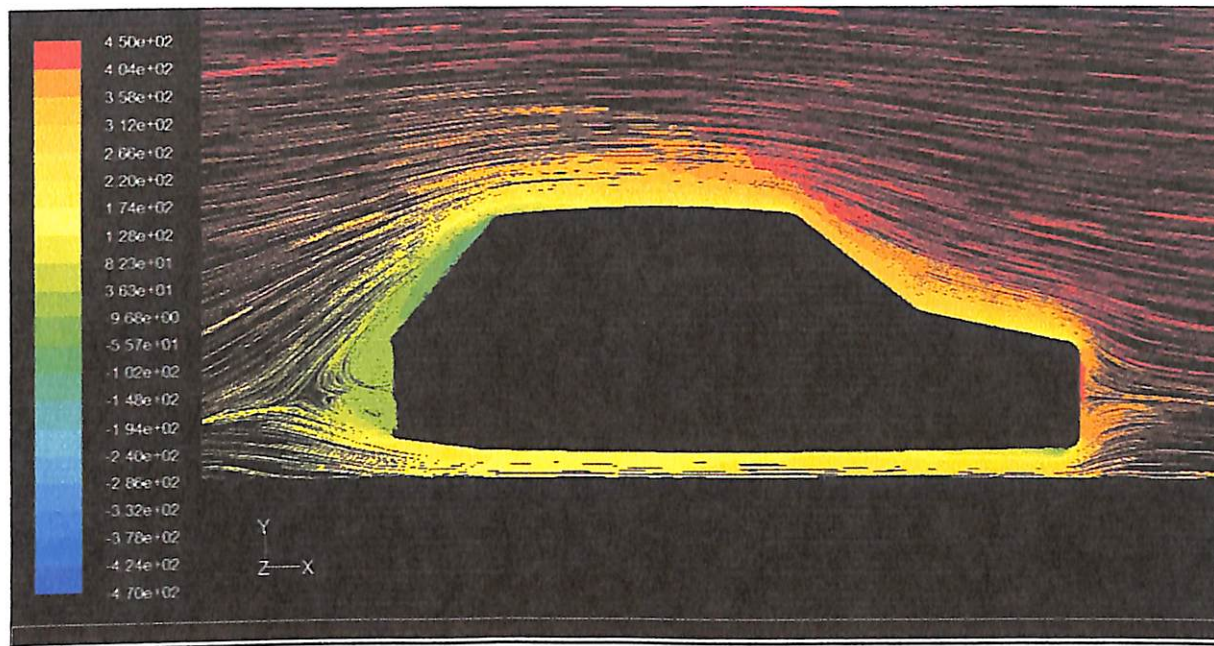


Fig: 5.5.1 Streamlines flow with respect to total pressure

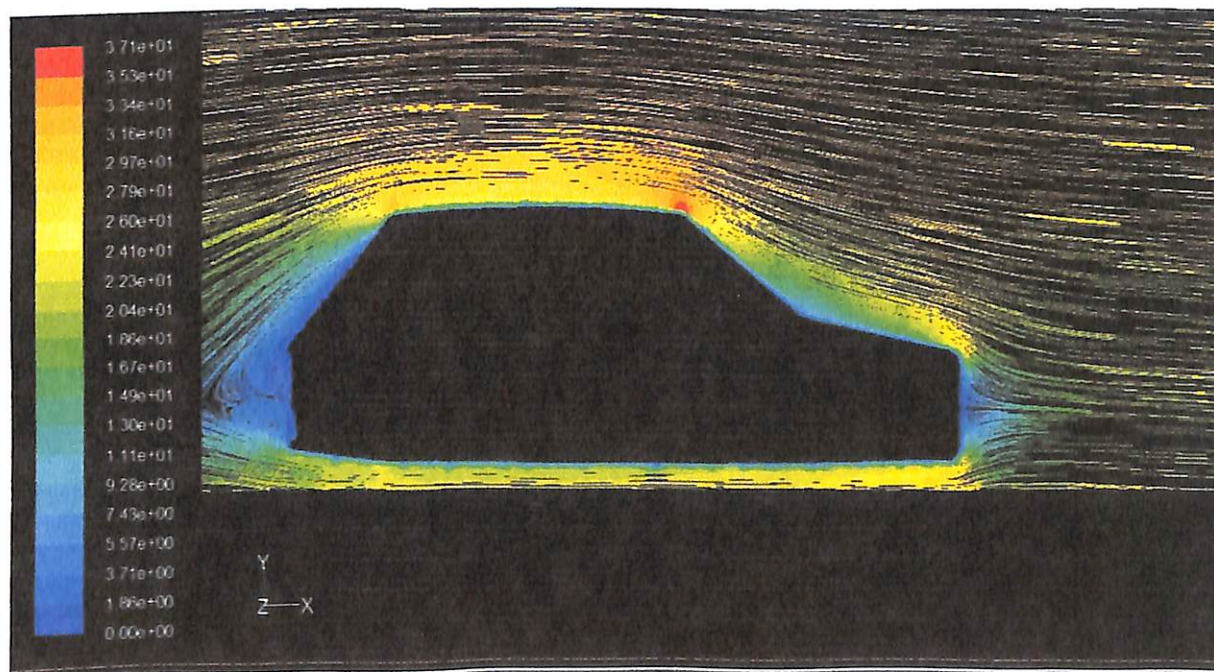


Fig: 5.5.2 Streamlines flow with respect to Velocity magnitude.

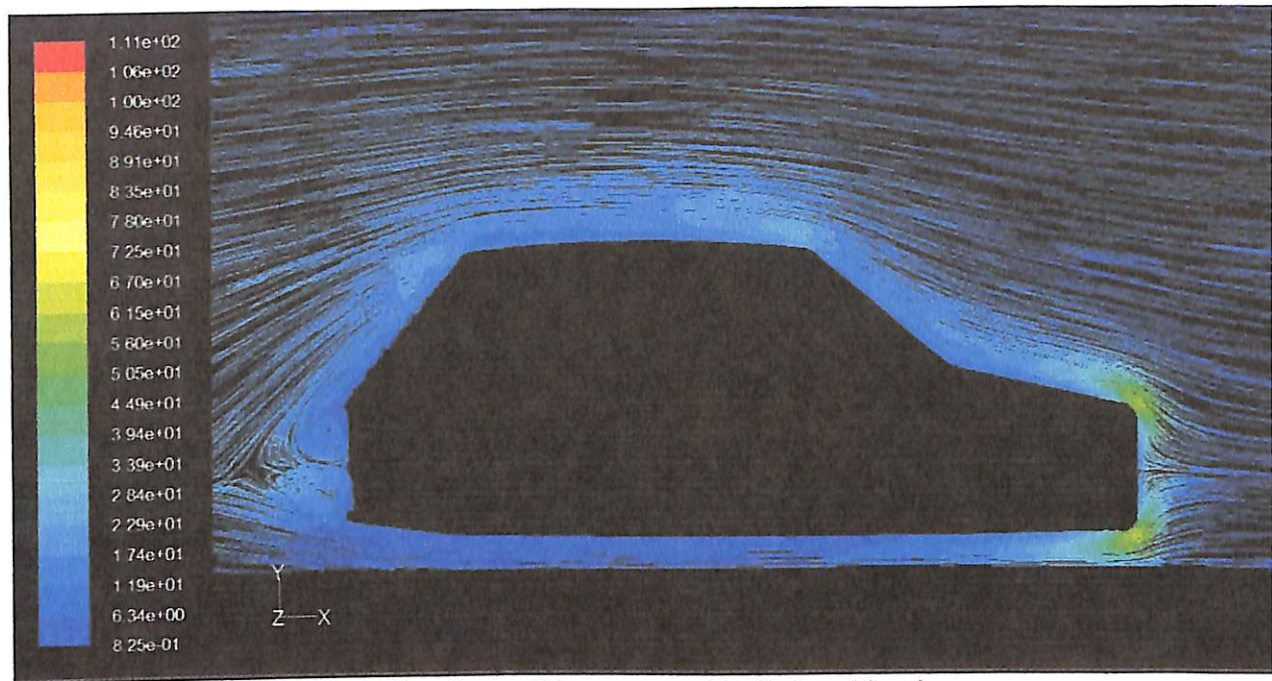


Fig: 5.5.3 Streamline flow with respect to Turbulent kinetic energy.

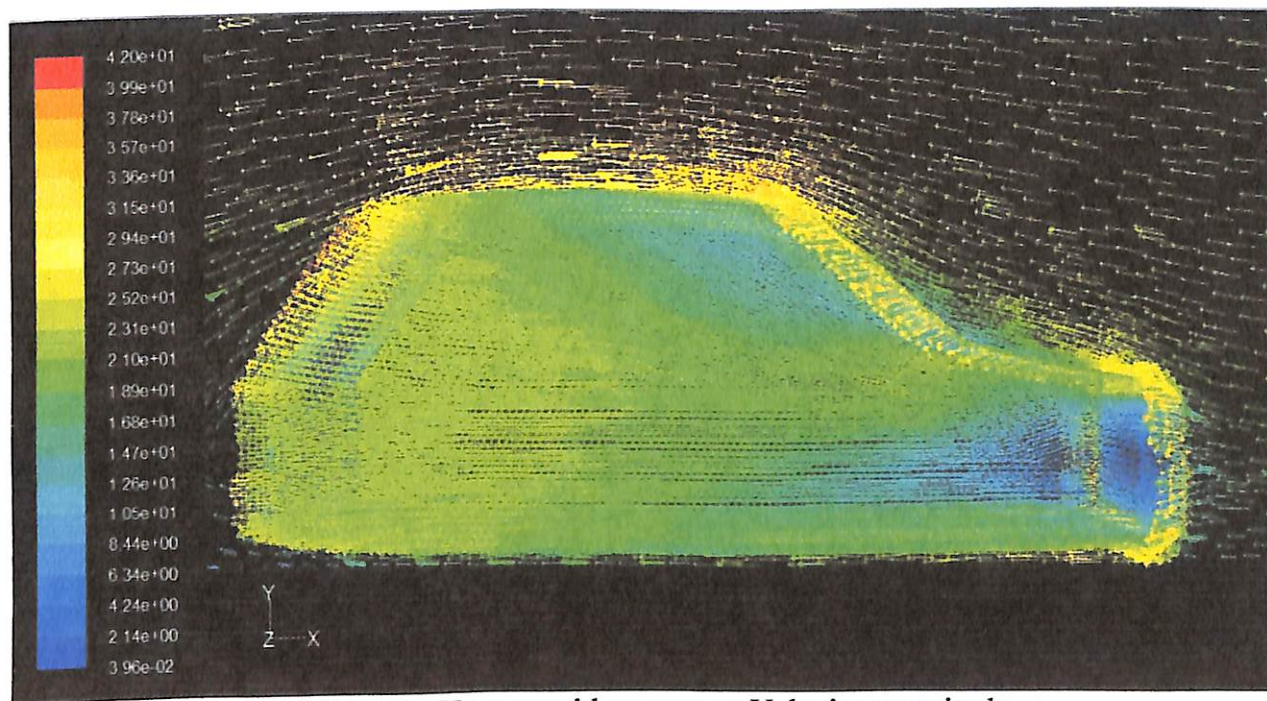


Fig: 5.5.4 Velocity Vectors with respect to Velocity magnitude.

5.5 Discussion

The analysis is done using the standard k-epsilon model.

- As demonstrated in the figure 5.1.1 the front end of the car experiences significantly higher pressure than the rest of the car. This can be attributed to its poor curvature and high projected area. The higher pressure concentration on the front end of the car contributes to an increase in the overall C_d value. The negative pressure at the roof is due to the continuous air flow over the roof with relatively faster moving air.
- The variation of the static pressure with respect to car length is shown in figure 5.1.2. The pressure is maximum at the front, which decreases suddenly and then increases gradually due to the localized flow separation. Particularly, air slows down when it approaches the front of the car and results that more air molecules are accumulated into a smaller space. Once the air stagnates in front of the car, it seeks a lower pressure area, such as the sides, top and bottom of car. As the air flows over the car hood, pressure is decreasing, but when reaches the front windshield it briefly increases. When the higher pressure air in front of the windshield travels over the windshield, it accelerates, causing the decreasing of pressure. This lower pressure literally produce a lift - force on the car roof as the air passes over it. The increase in the pressure at the rear is due to the flow separation. The rear end shape is the most critical factor in lowering the drag coefficient.
- As demonstrated in the figure 5.2.1 the kinetic energy at the front end becomes zero for a particular instance and increases gradually as the flow approaches the car body. The low velocity at the rear is due to the flow separation and formation of wakes. The magnitude of the velocity at the surface is zero and increases exponentially to the free stream velocity.
- The figure 5.3.1 shows the change of initial laminar flow into the turbulent flow near the bonnet. The formation of the turbulent boundary layer take place where the flow separation occurs and the randomness of the particles increases. In the turbulent flow the boundary layer is defined as the thin region on the surface of the car in which viscous effects are important.
- The total pressure plot in figure 5.4.1 is the sum of static and dynamic pressure. The static pressure is the measure of the pressure if the car is stopped suddenly for a particular instance of time.

Study of Flow over an Automobile

- Streamlines are used to study the air flow. Streamlines are studied with respect to pressure, velocity and turbulence intensity.
- The high pressure at the front is demonstrated in the figure 5.5.1. The negative pressure zone is represented by the green color contour at the rear as the flow separation takes place.
- The high turbulence zone at the front bonnet is demonstrated in the figure 5.5.3. The color contour clearly represent that there is high turbulence at the rear end.
- The figure 5.5.4 shows that the velocity of the air reduces as it approaches the front of the car. This velocity then increases due to a positive slope above the front end of the car. The airflow velocity over the roof reduces significantly and remains constant as it flows towards the rear of the car. As the flow is approaching from the inlet plane the density of the velocity vectors is initially low but as it approaches towards the car surface the density increases as it starts sticking to the car surface.

6. CONCLUSION

Using the Solidworks design software, the idea designing a 3D car was digitalized. The basic dimensions of Maruti 800 were used to visualize the air flow over the car profile using computational fluid dynamics software FLUENT. Computational fluid dynamics analysis was used to study the critical places in the geometry resulting in bad aerodynamics.

The results are obtained from the Fluent software after defining the boundary and operating conditions. These results are used to study the interaction of the car geometry with the air. The streamlines are used to study the air flow with respect to pressure, velocity and turbulence intensity plot. The pressure and velocity distribution over the car geometry is studied using the graphs obtained from Fluent. The area of the negative pressure developed at the front and rear end can be minimized using the fillet at those points. Leading to the obtained results an improved geometry can be made to reduce coefficient of drag and thus the fuel economy of the car can be increased. Redesign in terms of angle between the hood and windshield where the static pressure is high and also reducing the turbulence zone at the rear end of the car. These changes can be made to improve the air flow over the car.

Aerodynamic study is becoming most important factor in today's automotive industry for better fuel economy. The design of the car is made based upon the interaction with the air flow. It is extremely important for a design engineer to know the fundamentals of aerodynamics using CFD software's.

The project has been successful in studying the effect of aerodynamic forces on Maruti 800 and also in understanding the basic concepts of CFD software.

6.1 Future work

The reduction in the coefficient of drag C_d can be achieved by improving the airflow over the car geometry. Coefficient of drag can be reduced by making the continuous car profile and reducing the flow separation. Therefore by improving the shape of the car profile studied (i.e. maruti 800) in this project, better results in terms of fuel economy can be achieved. A comparative report can be prepared between the improved car geometry and the existing car geometry.

7. REFERENCES

1. Stone, Richard & K. Ball, Jeffery "Automotive Engineering Fundamentals". SAE Internationals
2. www.wikipedia.com/cfd
3. <http://www.afs.enea.it/funel/CFD/OpenFOAM3DCar>
4. Kumar Tirkey, Kunal & Kumar, Suresh "CFD Analysis over a Car body". S MCAD Center, MSRSAS Bangalore.
5. <http://farside.ph.utexas.edu/teaching/336L/Fluidhtml/node87.html>
6. <http://www.fluent.com/>
7. <http://auto.howstuffworks.com/aerodynamics>