CERTIFICATION OF APPROVAL

Computational Fluid Dynamics study of Airflow through a Car's Radiator

by

Mohammad Amer Qais Abro

A project dissertation submitted to the
Mechanical Engineering Programme
Universiti Teknologi PETRONAS
in partial fulfillment of the requirement for the
BACHELOR OF ENGINEERING (Hons)
(MECHANICAL ENGINEERING)

Approved by,

Prof. Dr. Vijay R Raghavan

Vijay R. Raghavan
Professor
Mechanical Engineering Department
Universiti Teknologi PETRONAS

UNIVERSITI TEKNOLOGI PETRONAS
TRONOH, PERAK
January 2010

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

MOHAMMAD AMER QAIS ABRO

ABSTRACT

Computers are now an integral part of engineering helping us achieve solutions to evermore complex tasks. This study aims to better understand computer modeling and analysis using the necessary software for Fluid Dynamics and computer modeling. The study will also help better understand the current designs of automotives and fluid mechanics in general.

There are many ways that performance of a car can be measured, by wind-tunnel testing, longer empirical studies or CFD but most of these concentrate on optimization of external fluid-flow. This study sees the relationship between airflow under and over the hood.

At the end of this report, the author will conclude on the analysis and research that has been done to look at the approach of the using CFD in order to study and understand fluid flow through different circumstances and for computer modeling. Hence, better understand today's technological advances.

ACKNOWLEDGEMENT

It is a pleasure to thank those who made this thesis possible. Many people have assisted me through the whole process of developing this project. I would like to first thank God for his blessing in completing my final year project.

I am heartily thankful to my supervisor, Prof. Dr. Vijay R Raghavan for his full support, encouragement and supervision from the preliminary to the concluding level of my project. It would have been next to impossible to write this thesis without his help and assistance.

My gratitude also goes to my beloved family who gave me the moral support I required.

I would also like to express my utmost gratitude to Ms. WanSalma Useng, Ms.Meera Madhavan, Mr.Mujawar Malik, Mr. Ahmed Fawad Ansari, Mr. Vinod Kumar and Mr. Asim Ali for aiding me throughout this project. Lastly, I offer my regards to my friends and all of those who have directly or indirectly encouraged me in any aspect in concluding this project.

TABLE OF CONTENTS

LIST OF FIGURES

Chapter 1: Introduction

1.1 Prblem Definition

External optimization of automotives has been widely studied which has led to better and more efficient cars. However studies for relationship between external drag and internal airflow through a car's radiator are limited.

1.2 Background

The aerodynamic drag coefficient of most passenger vehicles is now around **0.3**[1]. The use of body shape and external detail optimization has led to this low drag coefficient. The remaining areas of exploration and optimization are the underbody and cooling system. The cooling system of a typical passenger vehicle contributes between 6 and 10 percent to the overall drag of the vehicle. Furthermore engine cooling systems are designed to meet two rare and extreme conditions. Firstly, driving at maximum speed and secondly driving up a specified gradient at full throttle or while towing a trailer of maximum permitted mass. At all times, in fact the majority of the time, the cooling system operates below maximum capacity while incurring a drag penalty. The project is to see by how much the performance degradation takes place due to the shape of the intake.

1.3 Project specifications

1. Research on radiator specifications.
2. Research on radiator positioning
3. Study the airflow through differently positioned radiators
4. Computer modeling and analysis to come up with the result

1.4 Objectives

The objectives of the project are:

- Literature Review about car radiator design
- Study the placement of the radiator with respect to the car.
- Simulation of Airflow through Car radiator under different conditions
- Improve airflow conditions

1.5 Scope of Study

To achieve the objectives of this project, the scope of study are to find previous studies and analysis done on the subject matter and conduct in-depth research on designing of automotive radiators.

The project is limited to fluid flow only, thermal properties and changes are ignored.

Chapter 2: LITERATURE REVIEW

Although gasoline engines have improved a lot, they are still not very efficient at turning chemical energy into mechanical power. Most of the energy in the gasoline (perhaps 70% or two-thirds) is converted into heat, and it is the job of the cooling system to take care of some of that heat. In fact, the cooling system on a car driving down the freeway dissipates enough heat to heat two average-sized houses. [2] The primary job of the cooling system is to keep the engine from overheating by transferring this heat to the air, but the cooling system also has several other important jobs.

The engine in a car runs best at a fairly high temperature. When the engine is cold, components wear out faster, and the engine is less efficient and emits more pollution. So another important job of the cooling system is to allow the engine to heat up as quickly as possible, and then to keep the engine at a constant temperature. [2]

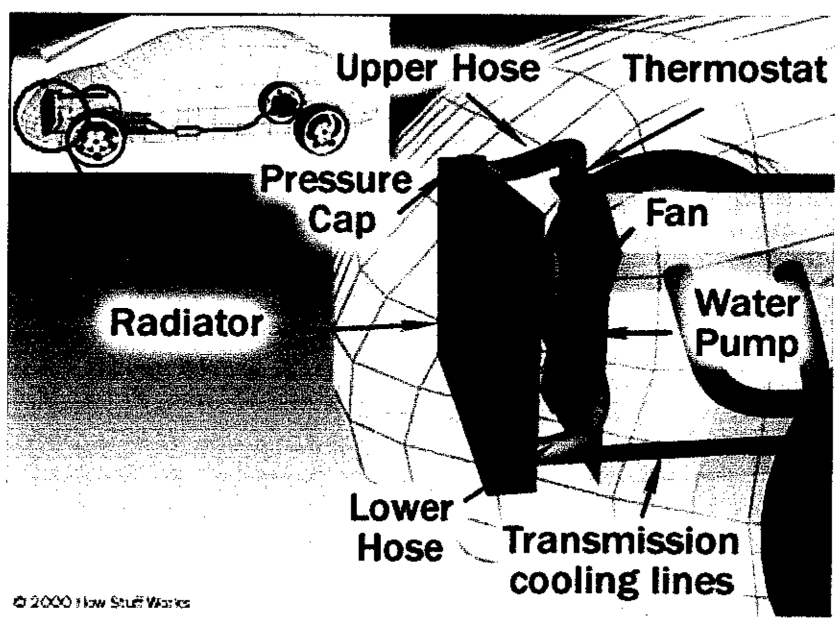


Figure1: Positioning of the Cooling system of an average sedan car.

Inside a car's engine, fuel is constantly burning. A lot of the heat from this combustion goes right out the exhaust system, but some of it soaks into the engine, heating it up. The engine runs best when its coolant is about 200 degrees Fahrenheit (93 degrees Celsius).

At this temperature:

The combustion chamber is hot enough to completely vaporize the fuel, providing better combustion and reducing emissions.

- The oil used to lubricate the engine has a lower viscosity (it is thinner), so the engine parts move more freely and the engine wastes less power moving its own components around.
- Metal parts wear less.

There are two types of cooling systems found on cars: liquid-cooled and air-cooled.

Liquid Cooling

The cooling system on liquid-cooled cars circulates a fluid through pipes and passageways in the engine. As this liquid passes through the hot engine it absorbs heat, cooling the engine. After the fluid leaves the engine, it passes through a heat exchanger, or radiator, which transfers the heat from the fluid to the air blowing through the exchanger. [3]

Air Cooling

Some older cars, and very few modern cars, are air-cooled. Instead of circulating fluid through the engine, the engine block is covered in aluminum fins that conduct the heat away from the cylinder. A powerful fan forces air over these fins, which cools the engine by transferring the heat to the air.

Since most cars are liquid-cooled, this study will focus on that system.

The pump sends the fluid into the engine block, where it makes its way through passages in the engine around the cylinders. Then it returns through the cylinder head of the engine. The thermostat is located where the fluid leaves the engine. The plumbing around the thermostat sends the fluid back to the pump directly if the thermostat is closed. If it is open, the fluid goes through the radiator first and then back to the pump.

There is also a separate circuit for the heating system. This circuit takes fluid from the cylinder head and passes it through a heater core and then back to the pump. [3]

Radiator is a type of heat exchanger. It is designed to transfer heat from the hot coolant that flows through it to the air blown through it by the fan.

Most modern cars use aluminum radiators. These radiators are made by brazing thin aluminum fins to flattened aluminum tubes. The coolant flows from the inlet to the outlet through many tubes mounted in a parallel arrangement. The fins conduct the heat from the tubes and transfer it to the air flowing through the radiator. [3]

The tubes sometimes have a type of fin inserted into them called a turbulator, which increases the turbulence of the fluid flowing through the tubes. If the fluid flowed very smoothly through the tubes, only the fluid actually touching the tubes would be cooled directly. The amount of heat transferred to the tubes from the fluid running through them depends on the difference in temperature between the tube and the fluid touching it. So if the fluid that is in contact with the tube cools down quickly, less heat will be transferred. By creating turbulence inside the tube, all of the fluid mixes together, keeping the temperature of the fluid touching the tubes up so that more heat can be extracted, and all of the fluid inside the tube is used effectively. [3]

Front-wheel drive cars have electric fans because the engine is usually mounted transversely, meaning the output of the engine points toward the side of the car. The fans are controlled either with a thermostatic switch or by the engine computer, and they turn on when the temperature of the coolant goes above a set point. They turn back off when the temperature drops below that point.

Rear-wheel drive cars with longitudinal engines usually have engine-driven cooling fans. These fans have a thermostatically controlled viscous clutch. This clutch is positioned at the hub of the fan, in the airflow coming through the radiator. This special viscous clutch is much like the viscous coupling sometimes found in all-wheel drive cars.

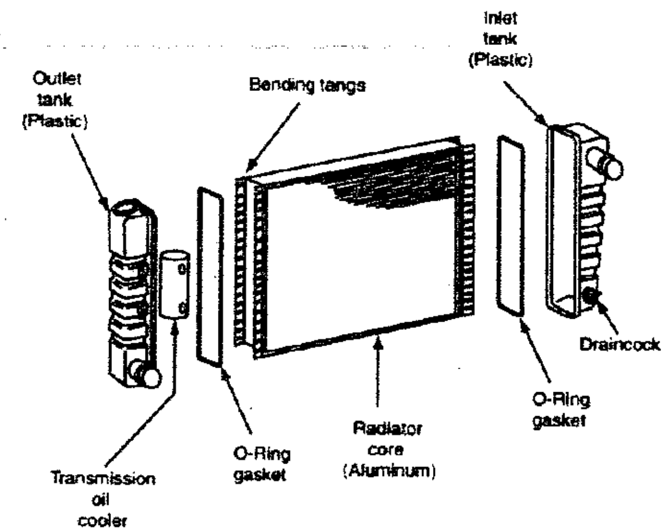


Figure2: components of radiator.

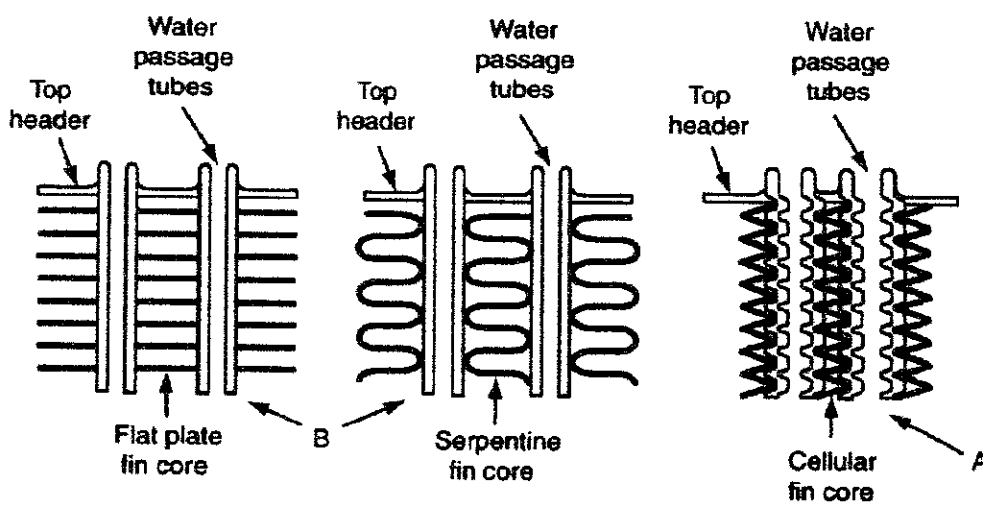


Figure3: Different fin designs.

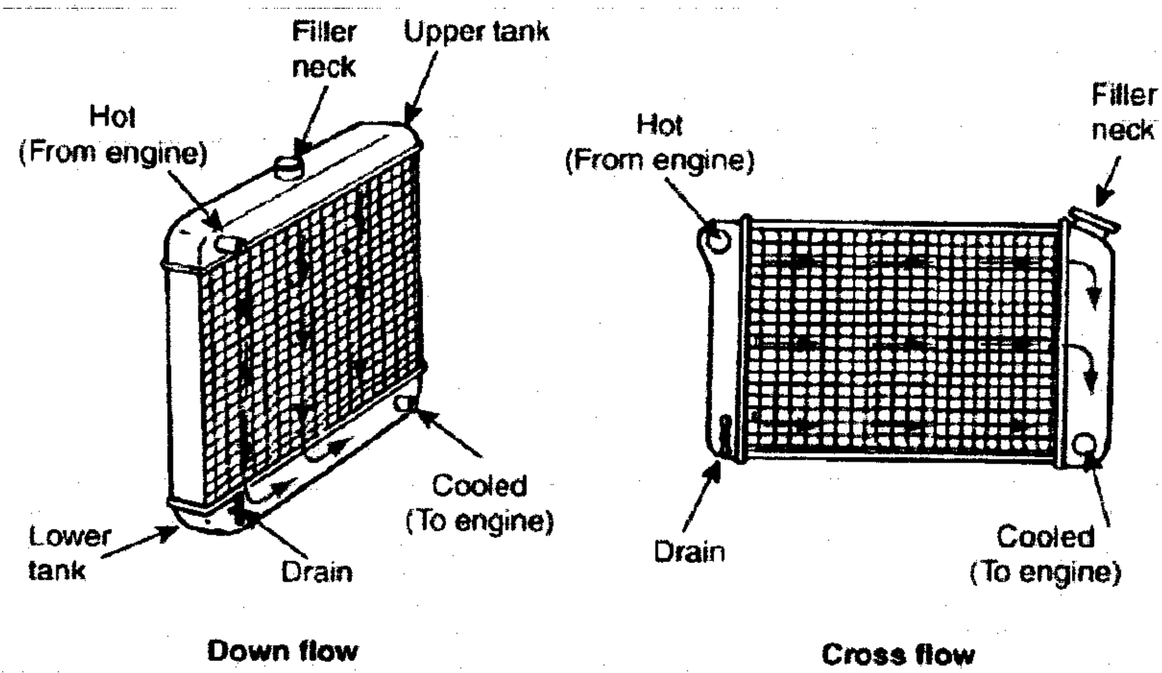


Figure4: The flow of engine-coolant through the radiator.

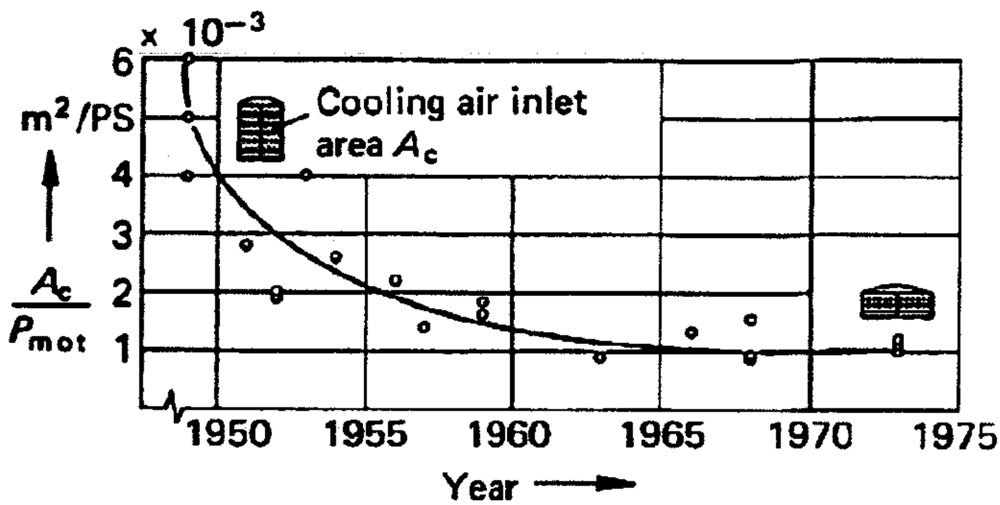


Figure5: *Cooling air intake area in relation to installed engine power versus year*

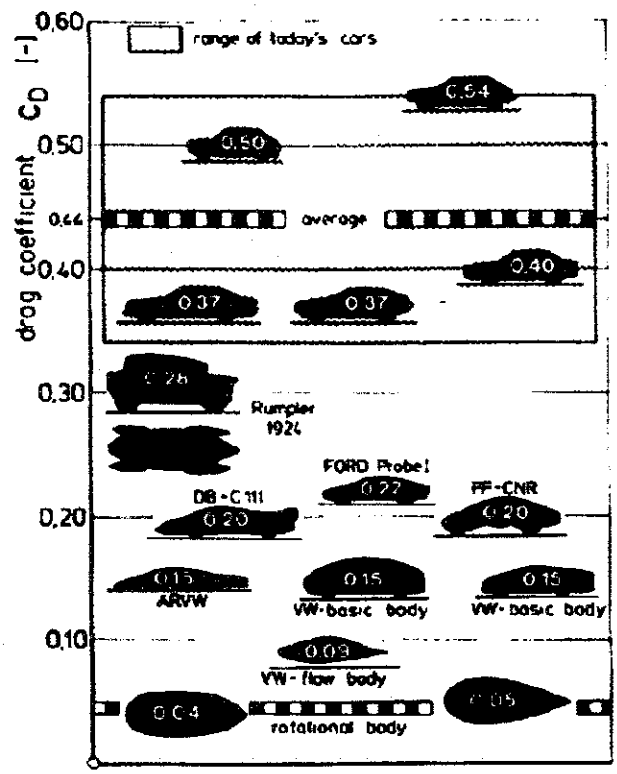


Figure6: *Drag coefficients of cars and ideal bodies*

Bahnsen demonstrated achievement of low aerodynamic drag of the Ford Probe III which had a drag coefficient of 0.22, which was equal to only 50% of the drag coefficient of a normal mid-sized family car at that time. [4] He further explained that this implied the engine power required would be significantly reduced by 36% or the fuel consumption would be lowered considerably by 27% for the same performance. Stapleford proved that reduction of aerodynamic drag could be done by minor modifications on a vehicle with add-on devices into the base vehicle, achieving as much as 30% drag reduction. Flegl and Bez indicated that a low stagnation- point vehicle offers good possibilities for favourable drag coefficient. [5]

Subsequently, the low aerodynamic drag concepts became a recognized development for modern vehicle design, achieved by low sloping hoods, soft and streamlined vehicle shapes, steeply raked windshields and high rear ends.

The drag coefficient is a result of external and internal flows. The largest contribution to drag from internal flows is the internal flow associated with engine cooling. Internal cooling drag is due to the momentum loss of the airflow entering through openings in the front-end to cool the radiator. It has been found that cooling drag contributes to around 5% – 10% of the total drag on most vehicles. [6]

In all mechanical systems, conversion of energy from the primary source to useful work cannot be achieved with 100% effectiveness. There is no exception for internal combustion engines. Only a fraction of the energy generated from the combustion of fuel in the cylinders produces useful work. For a typical passenger vehicle, considering the energy produced by fuel is dissipated approximately in three ways [7];

- Heat energy doing useful work: 35% - 45%
- Heat expelled with the exhaust gases: 30% - 40%
- Heat carried away by heat transfer: 22% - 28%

According to the above figures, there is an amount of 22% - 28% (almost one third of the total energy) of heat produced by combustion required to be dissipated. It is noted that part of this heat is usable in areas such as warming the cabin in cold weather for passenger comfort; and maintaining the engine at an optimum temperature (to achieve maximum combustion and lubrication efficiencies). The remainder is unwanted and must be removed. [9]

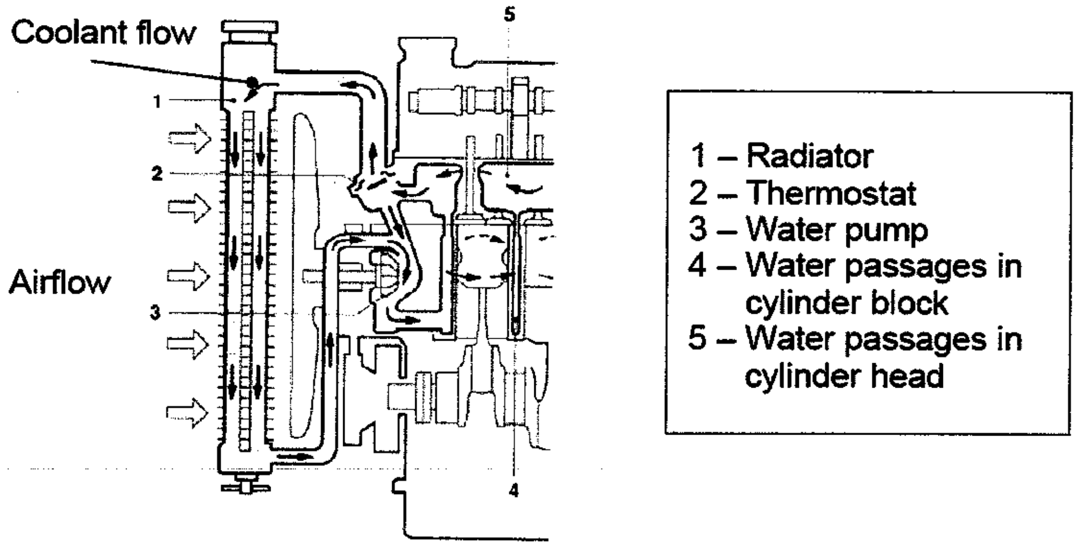


Figure 7: Design of radiator and airflow

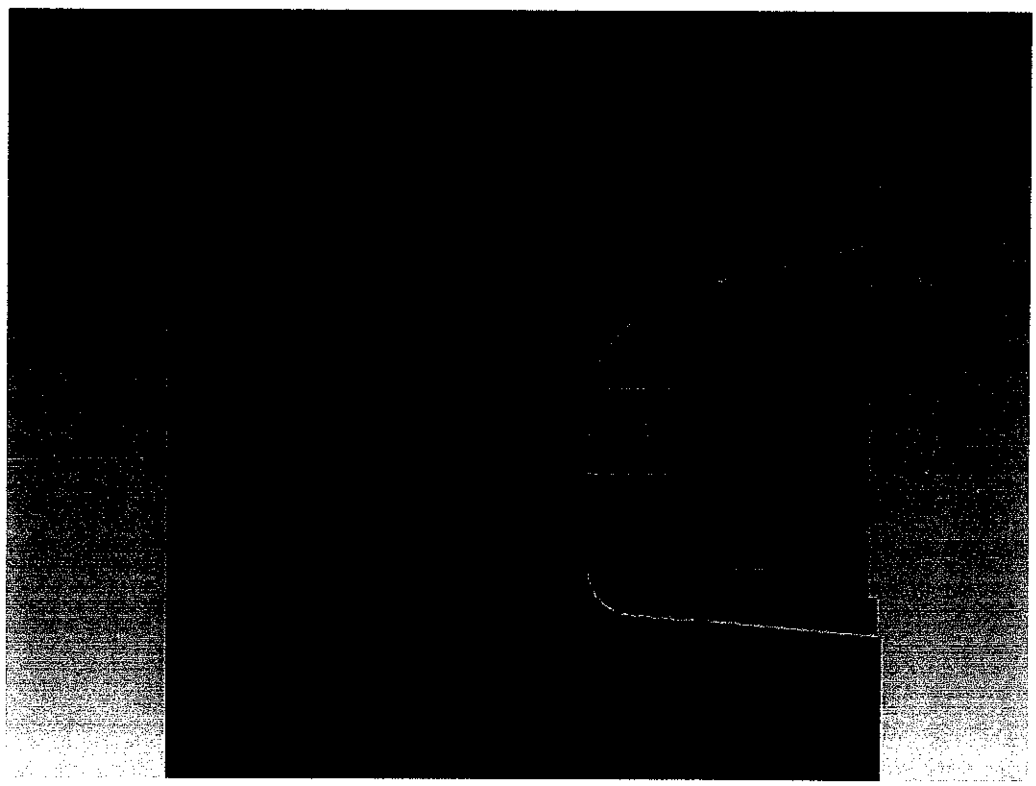


Figure 8: My basic design of radiator and airflow for CFD analysis.

With the concern of safety, locating bumpers with cross members in the vehicle front end is compulsory. As a consequence of this, the cooling air intake is usually split between top and bottom openings in the vehicle front end. This results in a reduction in the areas for air intakes and a distortion of the airflow in front of the radiator. The effect is that some of the air entering the front end becomes not productively used for cooling but possibly induces cooling drag.

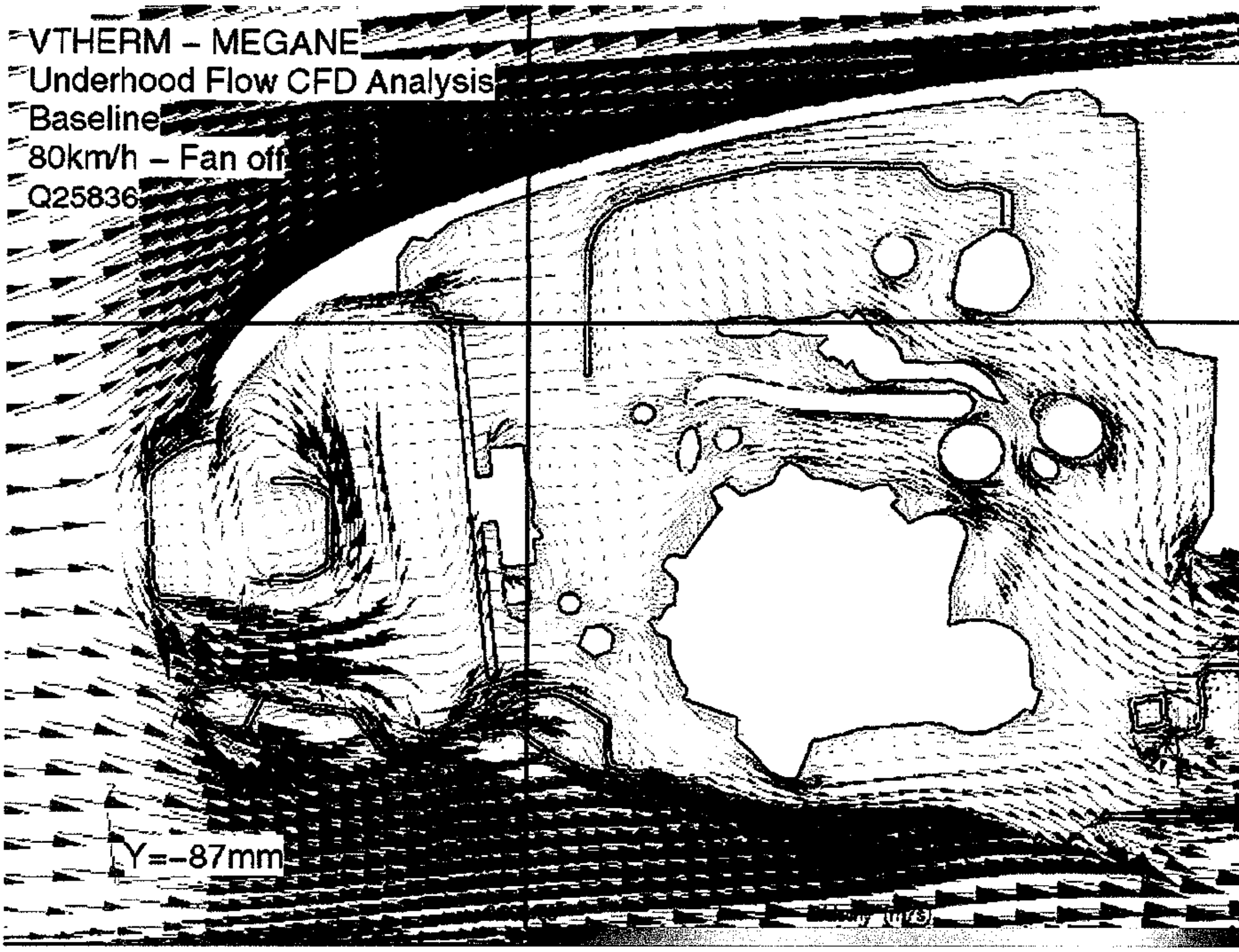


Figure 9: A previous study done on Airflow through a Car's Bonnet

Computational Fluid Dynamics[4]

Computer Aided Design (CAD)

The technology concerned with the use of computer systems to assist in the: *creation, modification, analysis and optimization* of a design

Examples of CAD are: AutoCad, Rhino, Catia

Computer Aided Manufacturing (CAM)

The technology concerned with the use of computer systems to *Plan, Manage and Control of manufacturing operation* through either direct or indirect use of computer interfacing

Example of CAE are automated assembly lines

Computer Aided Engineering (CAE)

The technology concerned with the use of computer systems to: *Analyze CAD geometry;* allowing the designer to simulate and study how the product (or the fluid flow or heat transfer) will behave so that the design can be refined and optimized.

Examples are: Fluent and, Ansys.

CFD or Computational Fluid Dynamics is a type of CAE that analyses fluid flow.

Computational fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the millions of calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. Even with high-speed supercomputers only approximate solutions can be achieved in many cases. Ongoing research, however, may yield software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial validation of such software is often performed using a wind tunnel with the final validation coming in flight test.

The fundamental basis of almost all CFD problems are the Navier–Stokes equations, which define any single-phase fluid flow. These equations can be simplified by removing terms describing viscosity to yield the Euler equations. Further simplification, by removing terms describing vorticity yields the full potential equations. Finally, these equations can be linearized to yield the linearized potential equations.

Historically, methods were first developed to solve the Linearized Potential equations. Two-dimensional methods, using conformal transformations of the flow about a cylinder to the flow about an airfoil were developed in the 1930s. The computer power available paced development of three-dimensional methods. The first paper on a practical three-dimensional method to solve the linearized potential equations was published by John Hess and A.M.O. Smith of Douglas Aircraft in 1966. This method discretized the surface of the geometry with panels, giving rise to this class of programs being called Panel Methods. Their method itself was simplified, in that it did not include lifting flows and hence was mainly applied to ship hulls and aircraft fuselages. The first lifting Panel Code (A230) was described in a paper written by Paul Rubbert and Gary Saaris of Boeing Aircraft in 1968. In time, more advanced three-dimensional

Panel Codes were developed at Boeing (PANAIR, A502), Lockheed (Quadpan), Douglas (HESS), McDonnell Aircraft (MACAERO), NASA (PMARC) and Analytical Methods (WBAERO, USAERO and VSAERO). Some (PANAIR, HESS and MACAERO) were higher order codes, using higher order distributions of surface singularities, while others (Quadpan, PMARC, USAERO and VSAERO) used single singularities on each surface panel. The advantage of the lower order codes was that they ran much faster on the computers of the time. Today, VSAERO has grown to be a multi-order code and is the most widely used program of this class. This program has been used in the development of many submarines, surface ships, automobiles, helicopters , aircraft, and more recently wind turbines. Its sister code, USAERO is an unsteady panel method that has also been used for modeling such things as high speed trains and racing yachts. The NASA PMARC code from an early version of VSAERO and a derivative of PMARC, named CMARC, is also commercially available.

In the two-dimensional realm, quite a number of Panel Codes have been developed for airfoil analysis and design. These codes typically have a boundary layer analysis included, so that viscous effects can be modeled. Professor Richard Eppler of the University of Stuttgart developed the PROFIL code, partly with NASA funding, which became available in the early 1980s. This was soon followed by MIT Professor Mark Drela's Xfoil code. Both PROFIL and Xfoil incorporate two-dimensional panel codes, with coupled boundary layer codes for airfoil analysis work. PROFIL uses a conformal transformation method for inverse airfoil design, while Xfoil has both a conformal transformation and an inverse panel method for airfoil design. Both codes are widely used.

An intermediate step between Panel Codes and Full Potential codes were codes that used the Transonic Small Disturbance equations. In particular, the three-dimensional WIBCO code, developed by Charlie Boppe of Grumman Aircraft in the early 1980s has seen heavy use.

Developers next turned to Full Potential codes, as panel methods could not calculate the non-linear flow present at transonic speeds. The first description of a means of using the Full Potential equations was published by Earll Murman and Julian Cole of Boeing in 1970. Frances Bauer, Paul Garabedian and David Korn of the Courant Institute at New York University (NYU) wrote a series of two-dimensional Full Potential airfoil codes that were widely used, the most important being named Program H. A further growth of Progam H was developed by Bob Melnik and his group at Grumman Aerospace as Grumfoil. Antony Jameson, originally at Grumman Aircraft and the Courant Institute of NYU, worked with David Caughey to develop the important three-dimensional Full Potential code FLO22 in 1975. Many Full Potential codes emerged after this, culminating in Boeing's Tranair (A633) code, which still sees heavy use.

The next step was the Euler equations, which promised to provide more accurate solutions of transonic flows. The methodology used by Jameson in his three-dimensional FLO57 code (1981) was used by others to produce such programs as Lockheed's TEAM program and IAI/Analytical Methods' MGAERO program. MGAERO is unique in being a structured cartesian mesh code, while most other such codes use structured body-fitted grids (with the exception of NASA's highly successful CART3D code, Lockheed's SPLITFLOW code and Georgia Tech's NASCART-GT).[1] Antony Jameson also developed the three-dimensional AIRPLANE code (1985) which made use of unstructured tetrahedral grids.

In the two-dimensional realm, Mark Drela and Michael Giles, then graduate students at MIT, developed the ISES Euler program (actually a suite of programs) for airfoil design and analysis. This code first became available in 1986 and has been further developed to design, analyze and optimize single or multi-element airfoils, as the MSES program. MSES sees wide use throughout the world. A derivative of MSES, for the design and analysis of airfoils in a cascade, is MISES, developed by Harold "Guppy" Youngren while he was a graduate student at MIT.

The Navier–Stokes equations were the ultimate target of developers. Two-dimensional codes, such as NASA Ames' ARC2D code first emerged. A number of three-dimensional codes were developed (OVERFLOW, CFL3D are two successful NASA contributions), leading to numerous commercial packages.

Finite Volume Analysis

The **finite volume method** is a method for representing and evaluating partial differential equations in the form of algebraic equations [LeVeque, 2002; Toro, 1999]. Similar to the finite difference method, values are calculated at discrete places on a meshed geometry. "Finite volume" refers to the small volume surrounding each node point on a mesh. In the finite volume method, volume integrals in a partial differential equation that contain a divergence term are converted to surface integrals, using the divergence theorem. These terms are then evaluated as fluxes at the surfaces of each finite volume. Because the flux entering a given volume is identical to that leaving the adjacent volume, these methods are conservative. Another advantage of the finite volume method is that it is easily formulated to allow for unstructured meshes. The method is used in many computational fluid dynamics packages.

Typical Steps in Finite Volume Analysis

Five steps involved in the *procedure*

1. Computer modeling, mesh generation 2. Definition of materials properties	**Pre-Processor**
3. Assemble of elements 4. Boundary conditions and loads defined	**Solver**
5. Solution using the required solver and display results/data	**Post- Processor**

Step1: Divide / discretize the structure or continuum into finite elements. This is typically done using mesh generation program, called pre-processor (in our case GAMBIT)

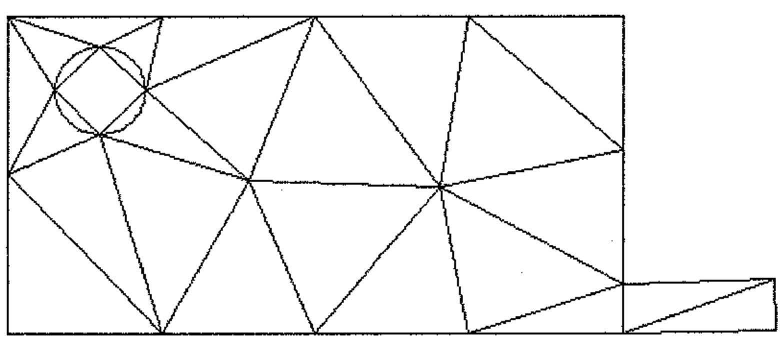


Figure10: Mesh Generation

Step2: Formulate the properties of each element.

Example: Nodal loads associated with all elements, deformation states that are allowed.

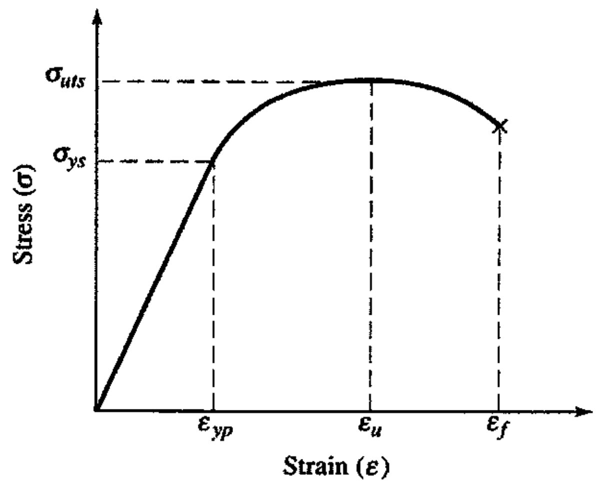


Figure11 : Example of Properties of Elements

Step3: Assemble elements to obtain FEA model

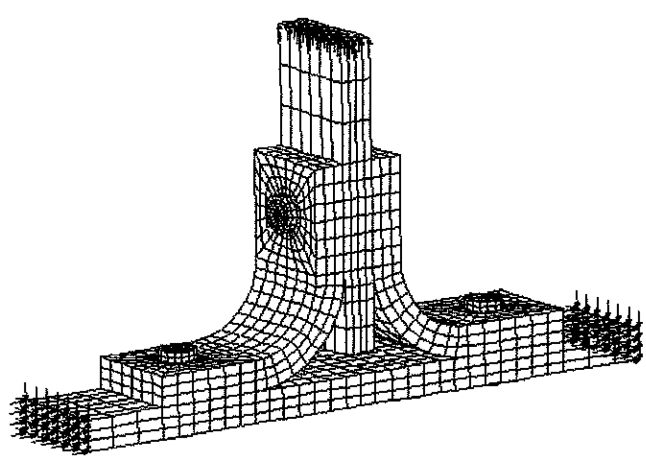


Figure12: Element Assembly

Step4: Specify the load and boundary conditions. Constraints, force, known temperatures, etc.

Step5: Solve simultaneous linear algebraic equations to obtain the solutions.

The modeling requirements include, simplified Model Geometry(example law of symmetry), Material Properties, Meshing (consider aspect ratio, element shape, symmetry and mesh refinement), Load Cases (surface, volume, or point loads), Boundary Condition (flow parameters)

The basic idea of Discretization is to replace the infinite dimensional linear problem, with a finite dimensional version: The elements are interconnected at points common to two or more elements (nodes or nodal points) and/or boundary lines and/or surfaces.

The transfer of load (force, displacement, heat flux, etc) between elements occurred at the common nodes between elements.

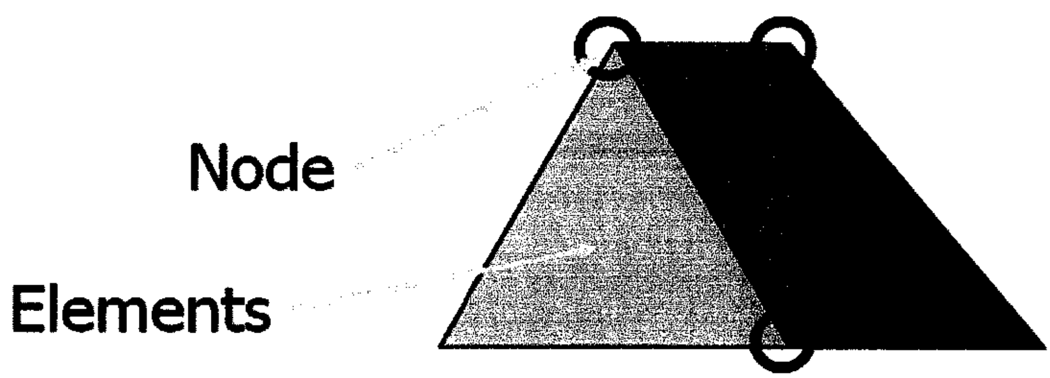
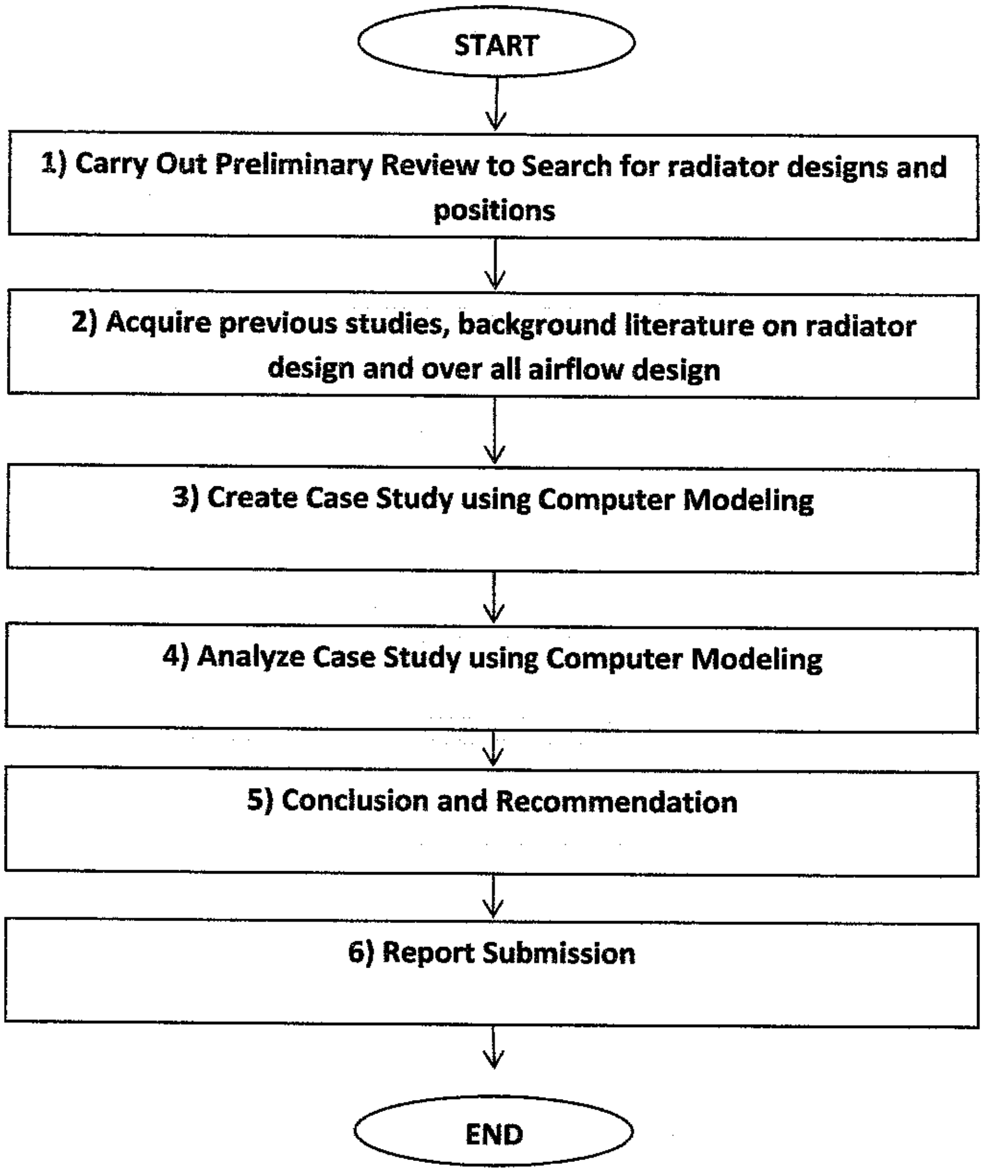


Figure13 : Discretization

Chapter 3: Planned Activities

Flow Chart



Methodology

This section describes the manner in which the project was carried out. The research project has been conducted and more valuable aspects of the project are discovered through this research. Since this is a research project with a final simulation being carried out, it was deemed important to acquire knowledge and references for every aspect of the project.

Important software for this project is Ansys. An initiative has been taken to gain better understanding of the software and how it could be used to effectively produce the required results. Other software used were AutoCAD and Rhino.

All other parameters necessary were obtained through the research for the project, this include pressure and temperature for air, properties of the fluid (air) used for the simulation of airflow through the radiator, the material through which airflow takes place, whether to use the actual model or simplify to a porous medium, better designing of radiator intakes and fans and many other related issues that might appear important at the later stage of the project. This research work should cover all these aspects so that the simulation time can be fully dedicated to simulation. For correct simulation, one need all parameters gathered and can be time consuming if all necessary information is not at the simulators disposal. The software depends on the inputs; hence it is important to have all of the inputs ready before the simulation can take place.

Model Setup

Below is a sample of how the experiments were conducted

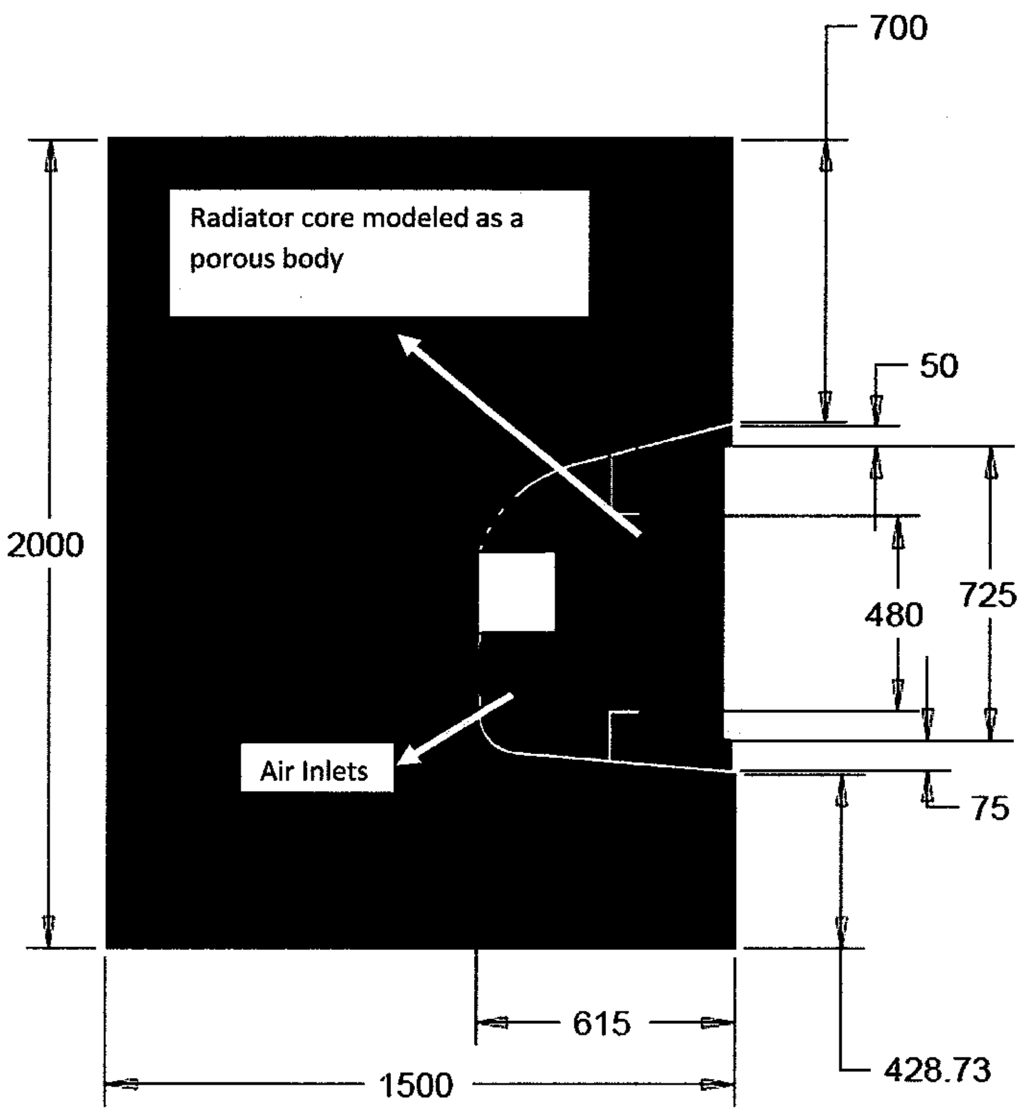


Figure14: Simplified representation of an actual car (dimensions are in mm)

The way to go about the study was to take the radiator (as shown above) and assume it to be a porous medium. There were four designs studied using CFD, The 2 models had no cowl (one had a single air intake one had double shown above). The remaining 2 models had air flow directed in to the radiator.

FLUID EQUATIONS AND MODELLING

Basically simplified version of the Navier-Stokes equations are used to for CFD

The derivation of the Navier–Stokes equations begins with an application of Newton's second law: conservation of momentum (often alongside mass and energy conservation) being written for an arbitrary portion of the fluid. In an inertial frame of reference, the general form of the equations of fluid motion is:[2]

$$\rho\left(\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v}\cdot\nabla\mathbf{v}\right) = -\nabla p + \nabla\cdot\mathbb{T} + \mathbf{f},$$

where $\mathbf{v}$ is the flow velocity, ρ is the fluid density, p is the pressure, $\mathbb{T}$ is the (deviatoric) stress tensor, and $\mathbf{f}$ represents body forces (per unit volume) acting on the fluid and ∇ is thedel operator. This is a statement of the conservation of momentum in a fluid and it is an application of Newton's second law to a Continuum; in fact this equation is applicable to any non-relativistic continuum and is known as the Cauchy momentum equation.

This equation is often written using the substantive derivative *D***v**/*Dt*, making it more apparent that this is a statement of Newton's second law:

$$\rho\frac{D\mathbf{v}}{Dt} = -\nabla p + \nabla\cdot\mathbb{T} + \mathbf{f}.$$

The left side of the equation describes acceleration, and may be composed of time dependent or convective effects (also the effects of non-inertial coordinates if present). The right side of the equation is in effect a summation of body forces (such as gravity) and divergence of stress (pressure and shear stress).

In CFX we use the K-epsilon model for simulation analysis

The K-epsilon model is one of the most common turbulence models. It is a two equation model, that means, it includes two extra transport equations to represent the turbulent properties of the flow. This allows a two equation model to account for history effects like convection and diffusion of turbulent energy.

The first transported variable is turbulent kinetic energy, k. The second transported variable in this case is the turbulent dissipation, ϵ. It is the variable that determines the scale of the turbulence, whereas the first variable, k, determines the energy in the turbulence.

There are two major formulations of K-epsilon models. That of Launder and Sharma is typically called the "Standard" K-epsilon Model. The original impetus for the K-epsilon model was to improve the mixing-length model, as well as to find an alternative to algebraically prescribing turbulent length scales in moderate to high complexity flows.

Tthe K-epsilon model has been shown to be useful for free-shear layer flows with relatively small pressure gradients. Similarly, for wall-bounded and internal flows, the model gives good results only in cases where mean pressure gradients are small; accuracy has been shown experimentally to be reduced for flows containing large adverse pressure gradients

Transport equations for standard k-epsilon model

For turbulent kinetic energy k

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k + P_b - \rho\epsilon - Y_M + S_k$$

For dissipation ϵ

$$\frac{\partial}{\partial t}(\rho\epsilon) + \frac{\partial}{\partial x_i}(\rho\epsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right] + C_{1\epsilon}\frac{\epsilon}{k}\left(P_k + C_{3\epsilon}P_b\right) - C_{2\epsilon}\rho\frac{\epsilon^2}{k}$$

Modeling turbulent viscosity

Turbulent viscosity is modelled as:

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

Production of k

$$P_k = -\rho \overline{u_i' u_j'} \frac{\partial u_j}{\partial x_i}$$

$$P_k = \mu_t S^2$$

Where S is the modulus of the mean rate-of-strain tensor, defined as :

$$S \equiv \sqrt{2 S_{ij} S_{ij}}$$

Effect of bouyance

$$P_b = \beta g_i \frac{\mu_t}{\Pr_t} \frac{\partial T}{\partial x_i}$$

where $\Pr_t$ is the turbulent Prandtl number for energy and g_i is the component of the gravitational vector in the ith direction. For the standard and realizable - models, the default value of $\Pr_t$ is 0.85.

The coefficient of thermal expansion, β, is defined as

$$\beta = -\frac{1}{\rho}\left(\frac{\partial \rho}{\partial T}\right)_p$$

Model constants

$$C_{1\epsilon} = 1.44, \quad C_{2\epsilon} = 1.92, \quad C_\mu = 0.09, \quad \sigma_k = 1.0, \quad \sigma_\epsilon = 1.3$$

NUMERICAL TESTING (COMPUTATIONAL FLUID DYNAMIC –CFD SIMULATION)

Then the preliminary design will be justified by using Computational Fluid Dynamic (CFD) software, Ansys CFX. It is used for simulation, visualization, and analysis of fluid flows and in this project, for modeling flow conditions in and around moving objects. Drawing shapes and configurations of design, a geometric modeling and grid generation tool, Ansys work bench is used, to allow import of geometry from most Computational Aided Design (CAD) packages. Meshing is done on CFX itself

Experimental Setup Conditions

Two domains are defined here, named Air_Domain and Porous_Domain.

One domain interface: named Air_Porous_Interface.

<u>In domain Air_Domain</u>:

Inlet Condition: 100km/h

Symmetry Boundary Condition is assumed.

Outlet is at 0Pa i.e. Atmospheric conditions.

Fluid Used is Air at 25C and 1atm.

Walls are considered to be smooth

Thermal Model: none

<u>In Porous Domain</u>:

Porosity Area factor is assumed to be 0.5. i.e. 50% is available flow area.

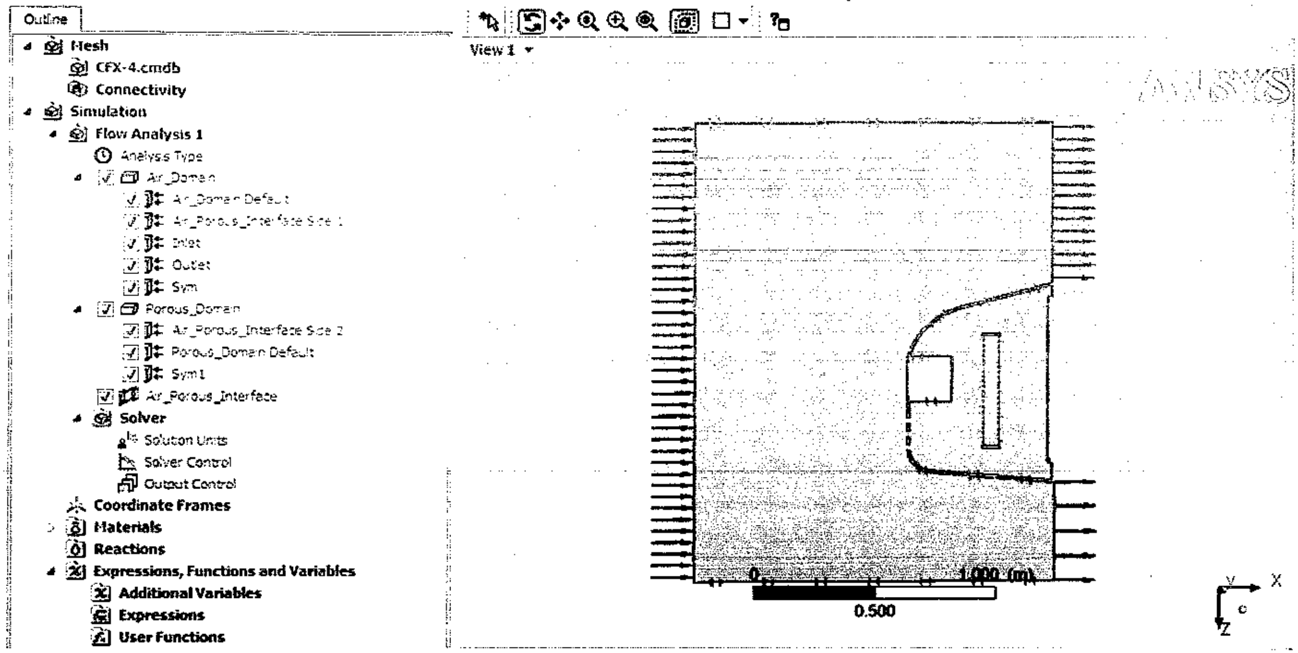


Figure15:Boundary condition example

Meshing

Method/Type: Tetrahedron

Face and Edge Mesh is used to improve overall mesh

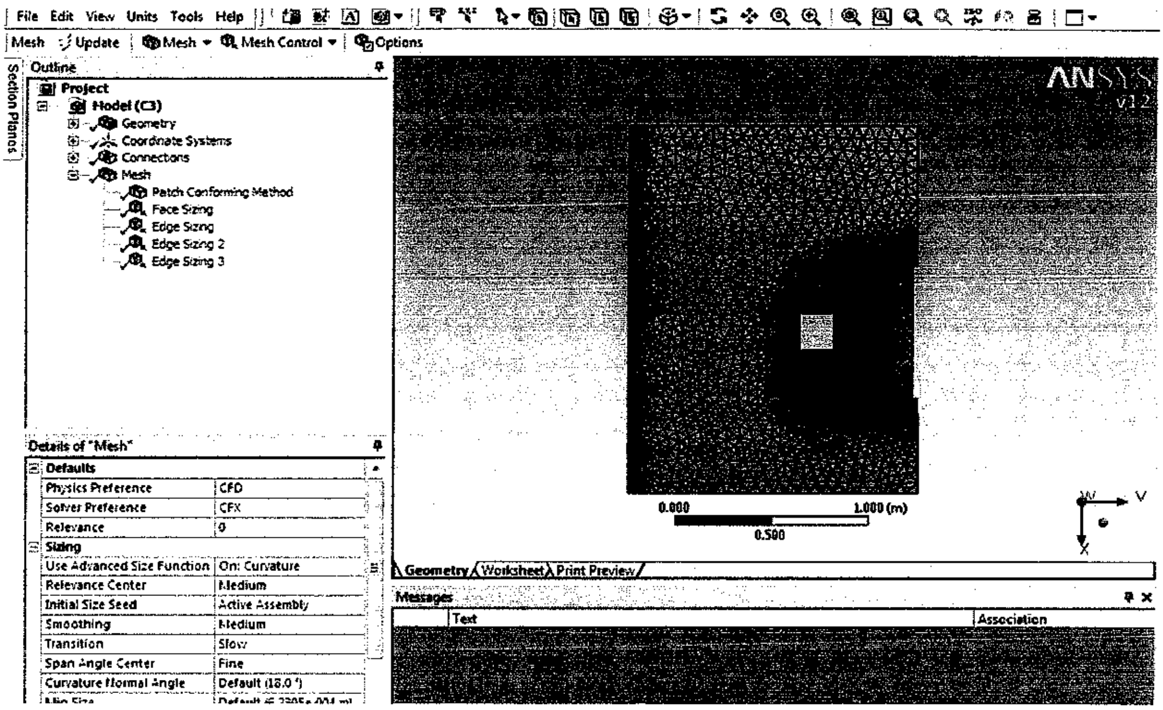


Figure16:Mesh Sample

Domain	Nodes	Elements
Air_Domain	14206	39529
Porous_Domain	593	1516
All Domains	14799	41045

Next a Residual graph was taken for a convergence path of the solutions for continuity and Momentum equations.

Two limits were set:

1. if 100 iterations complete terminate solution
2. Residual value is less than 1E-4 solution ends

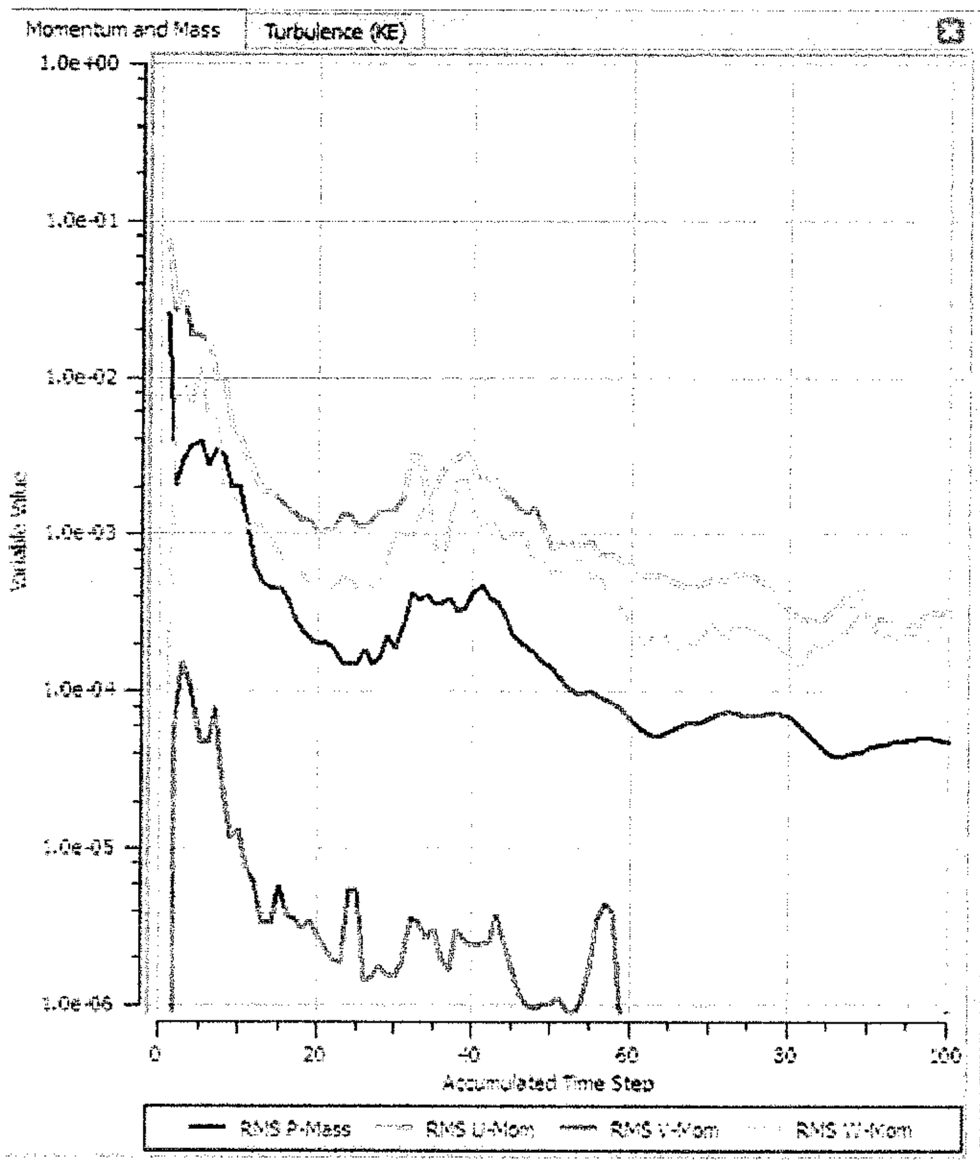


Figure17: Residual

After this for all four models Grid Independence Test was done

Due to approximating the solutions there are 2 errors involved

i. Discretization Error

ii. Truncation Error

Discretization Error is due to creating finite volumes to simplify and reach a solution

Truncation Error is due to the limited accuracy of the computer to tabulate result thus the rest of the digits are rounded off.

To minimize these two errors solutions for a model are compared under different meshes. Once the solution is less than 1% in comparison Grid Independence has been achieved

Thus the momentum of air solutions for 2 meshes of the above model were

Mesh1: -2.9739

Mesh2: -2.9594

% difference = 0.49%

Similarly total Pressure forces were

Mesh1: 2.9044

Mesh2: 2.8889

% Difference = 0.53%

% change in Viscous forces= 0.46%

Thus Grid Independence has been achieved. Below are the 2 models and their cell size information

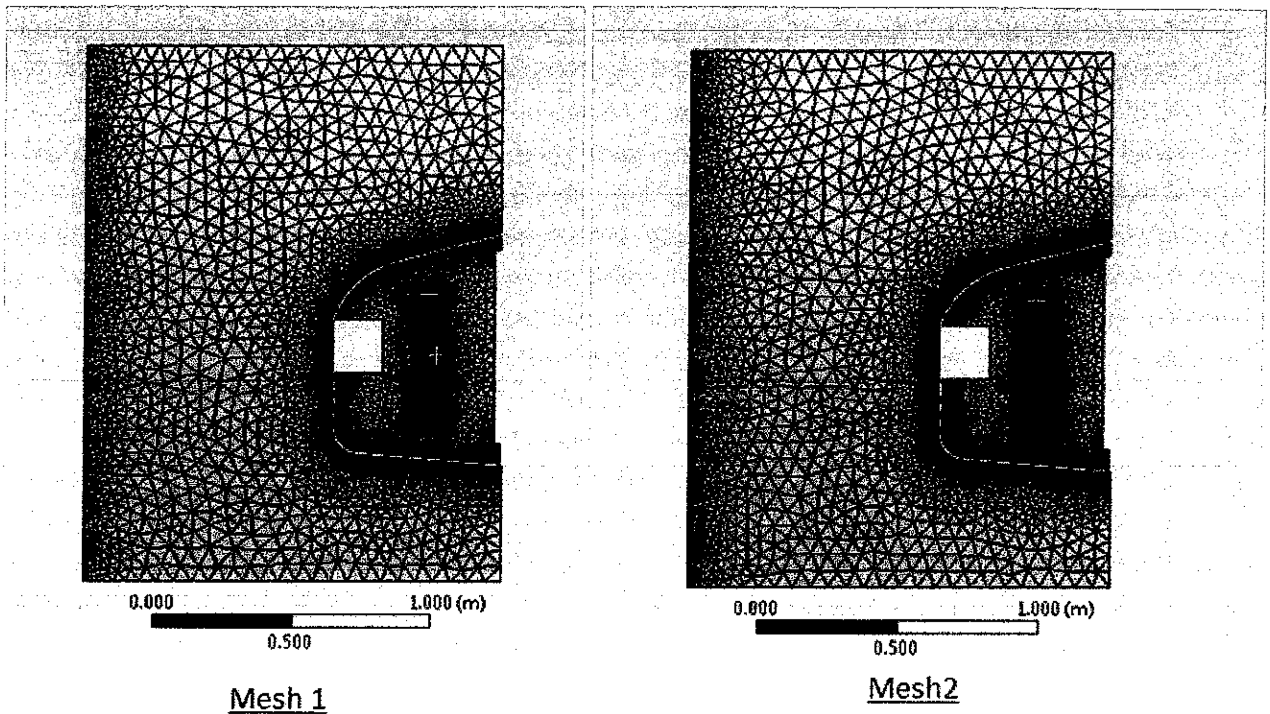


Figure18: Different Meshes for model 1

Domain	Nodes	Elements
Air_Domain	14206	39529
Porous_Domain	593	1516
All Domains	14799	41045

Domain	Nodes	Elements
Air_Domain	31964	106784
Porous_Domain	593	1516
All Domains	32557	108300

In the rest of the models mesh 2 was chosen as preferred mesh for the solutions

The four models simulated were as follows over all dimensions remained constant

The first model had only one air intake and no cowl to direct the flow in to the radiator

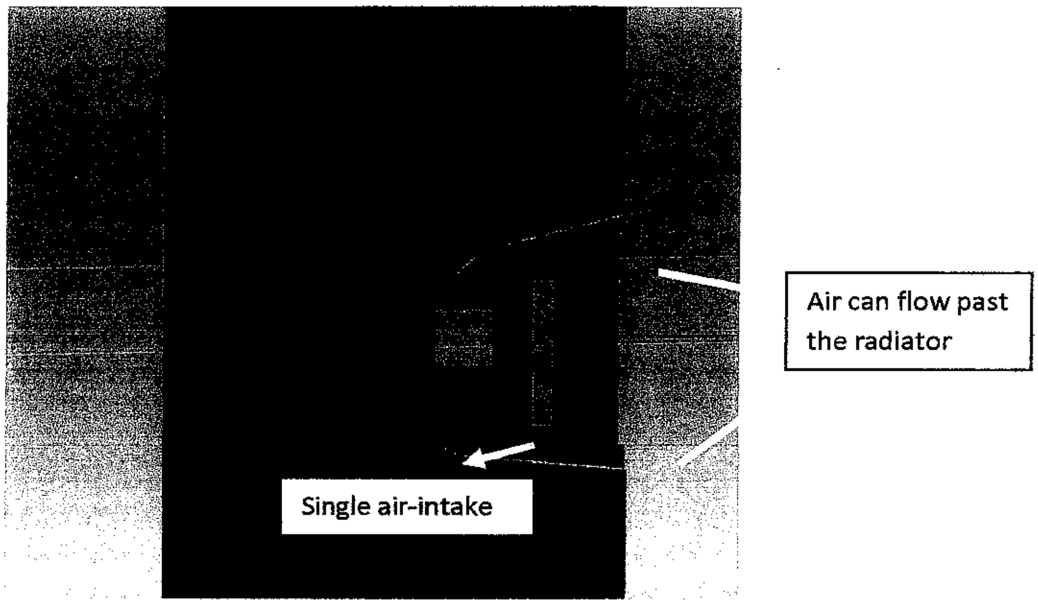


Figure19: Model 1A

Second model was the same as above except that air was forced to flow through the radiator

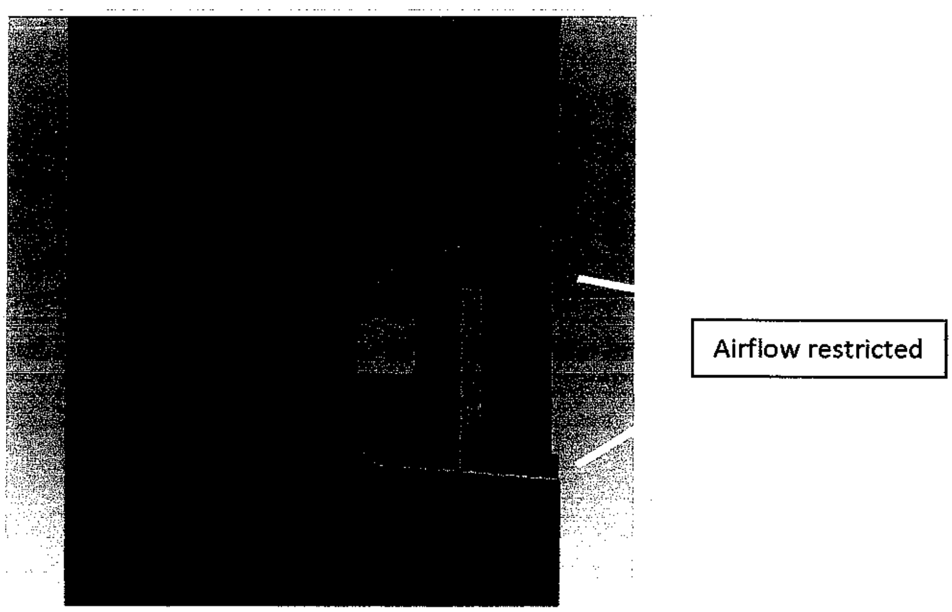


Figure20: Model 1B

The third model had two airflow intakes but air could flow past the radiator (same as 1A)

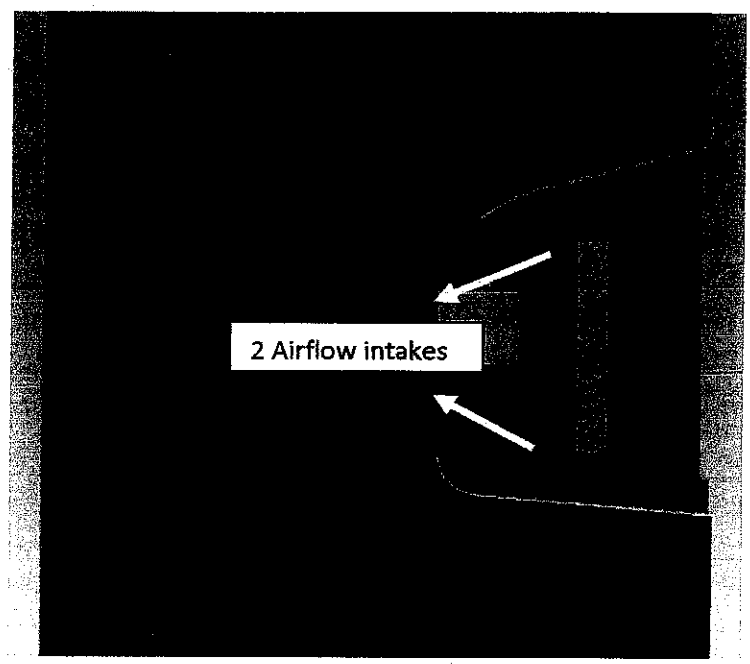


Figure21: Model 2A

The fourth model was same as Model 2A except that air was forced to flow through the radiator

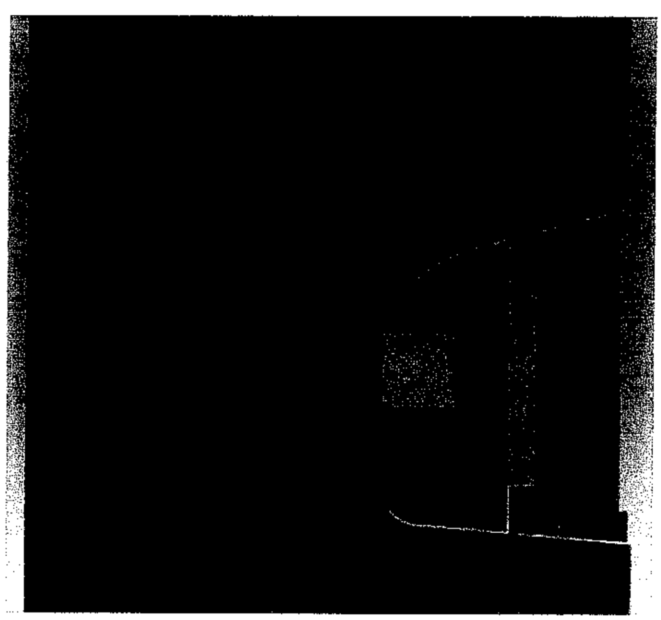


Figure22: Model 2B

Chapter 4: Result and Discussion

4.1 Result

The simulation analysis came up with the results as follows

For Model 1A

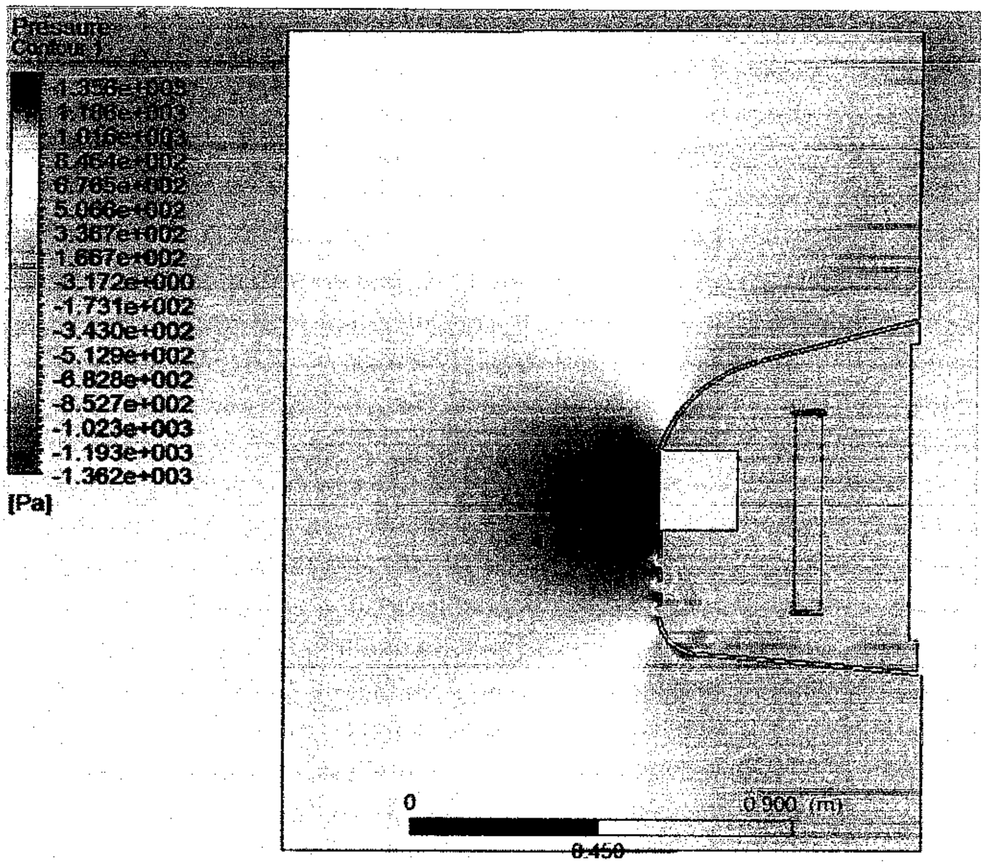


Figure 23: Model 1A Pressure Contour

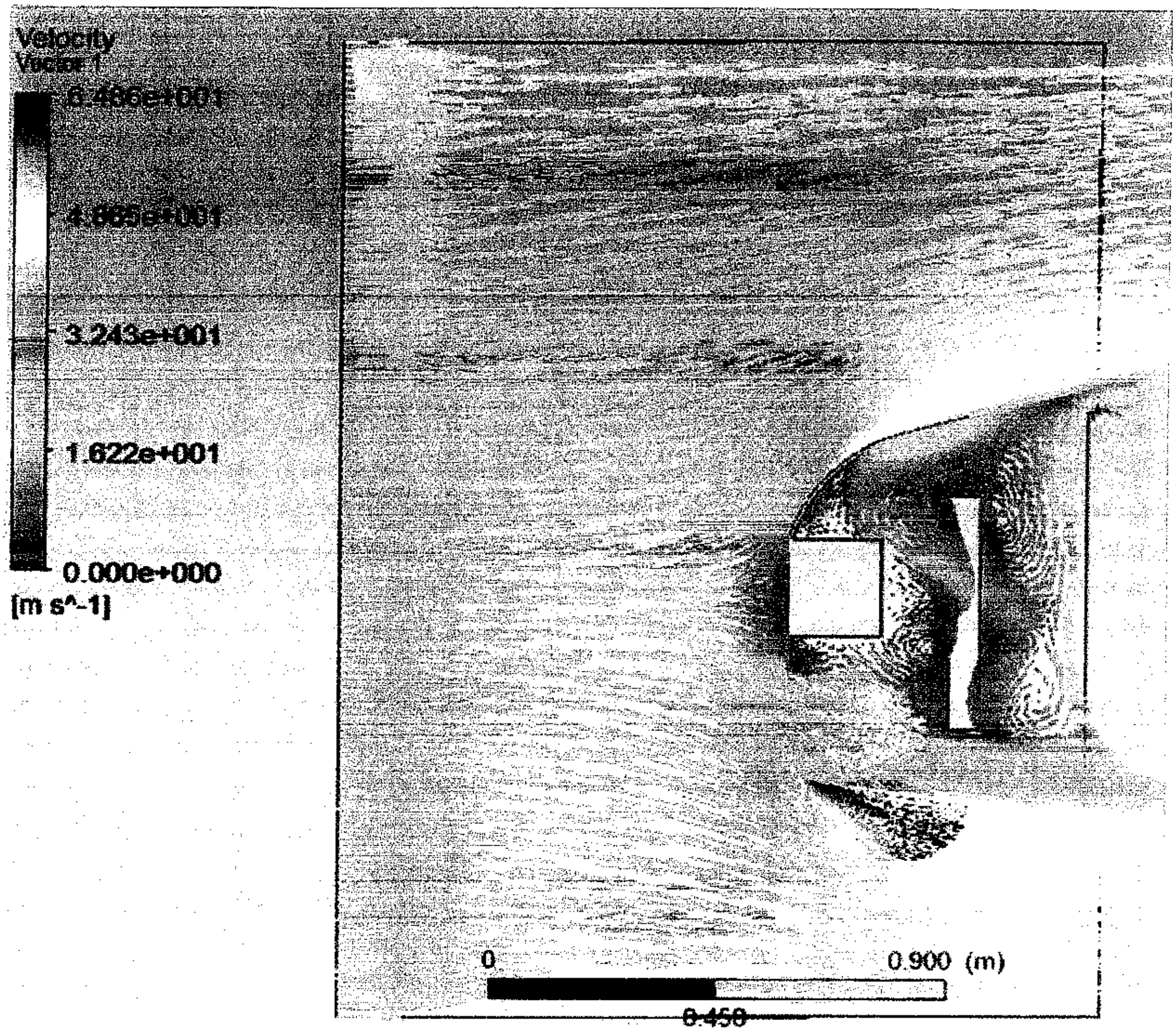


Figure 24: Model 1A Velocity Vectors

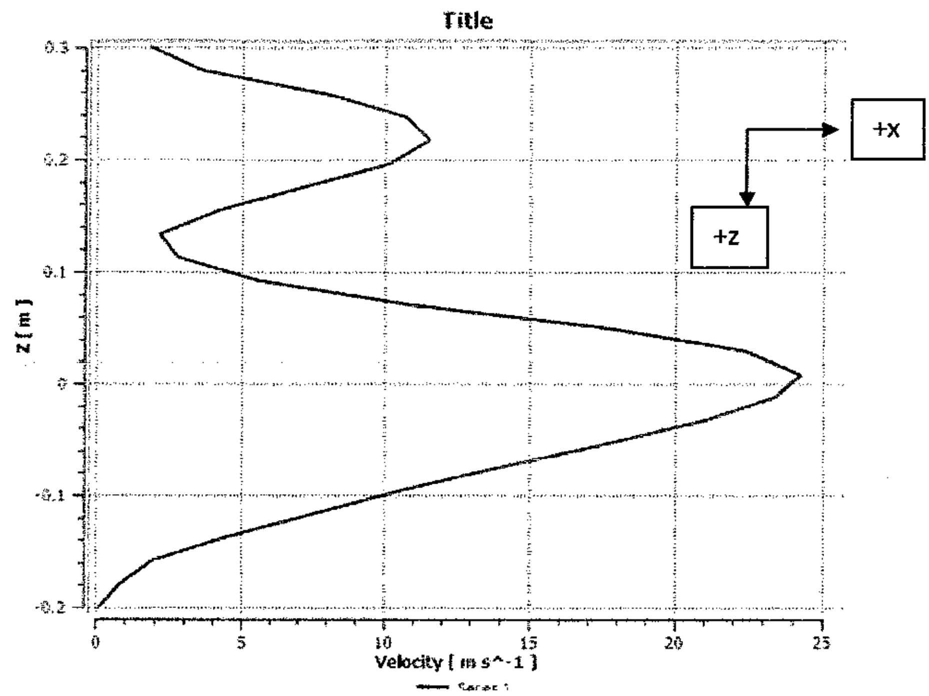


Figure 25: Model 1A Velocity Graph through the Radiator

Note that the positive Z direction is downwards for all models.

For Model 1**B**

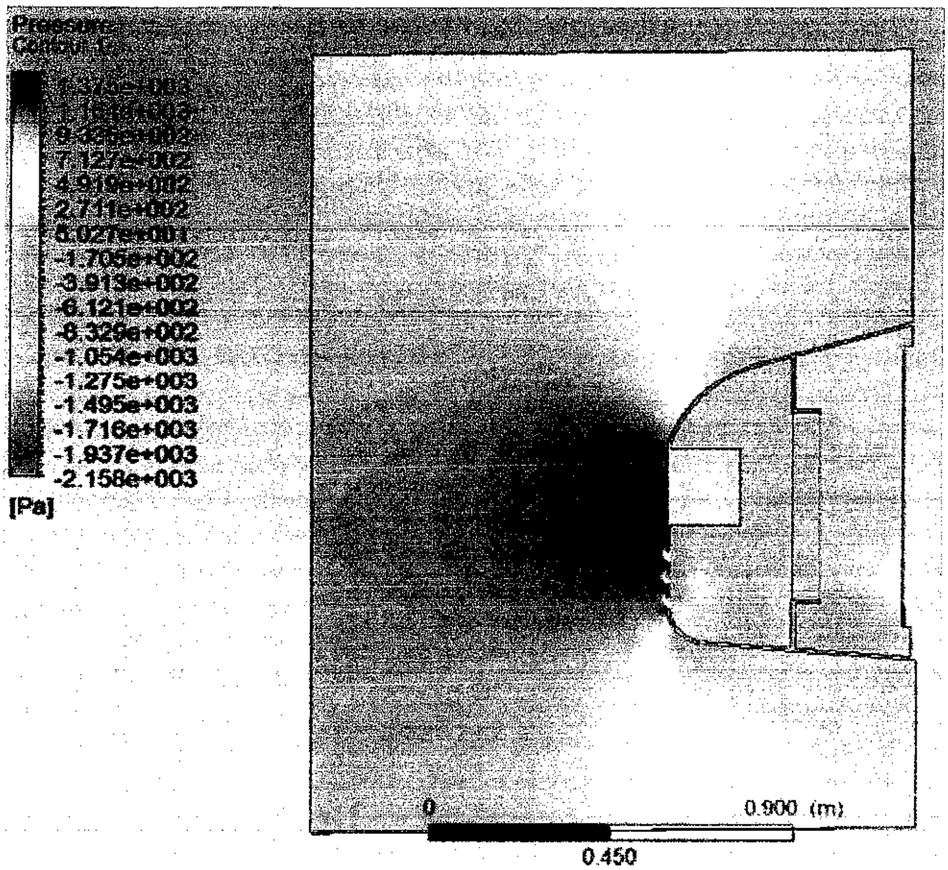


Figure 26: Model 1B Pressure Contours

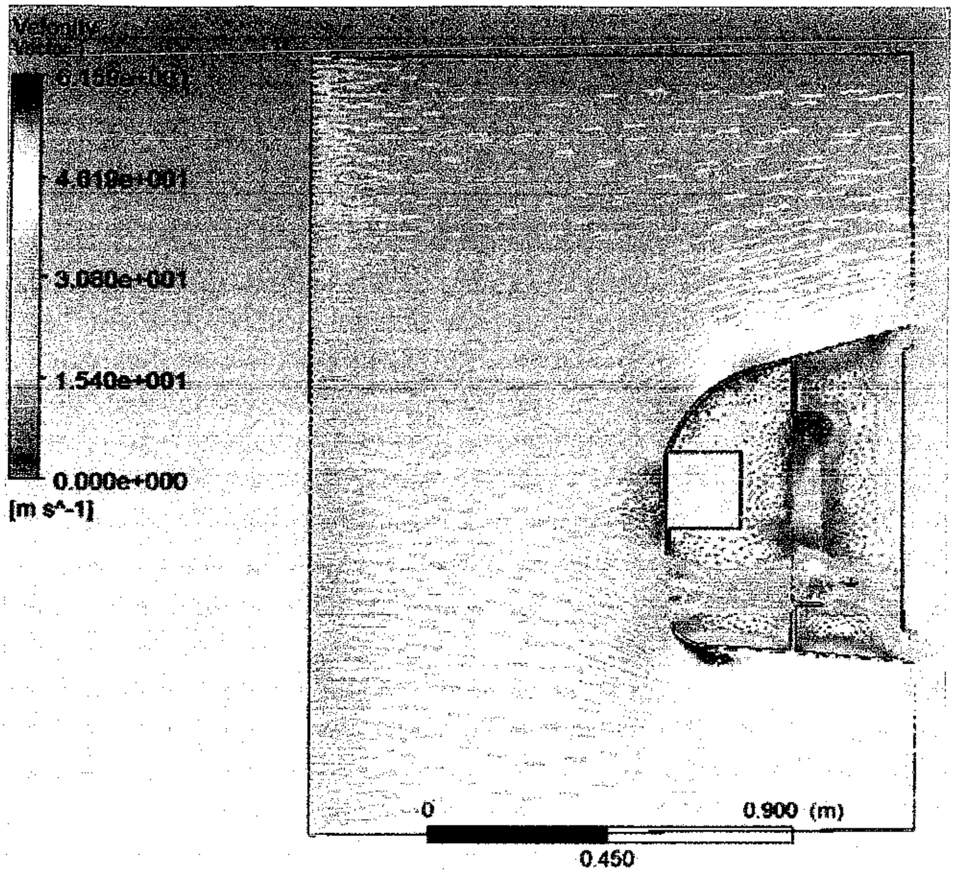


Figure 27: Model 1B Velocity Vectors

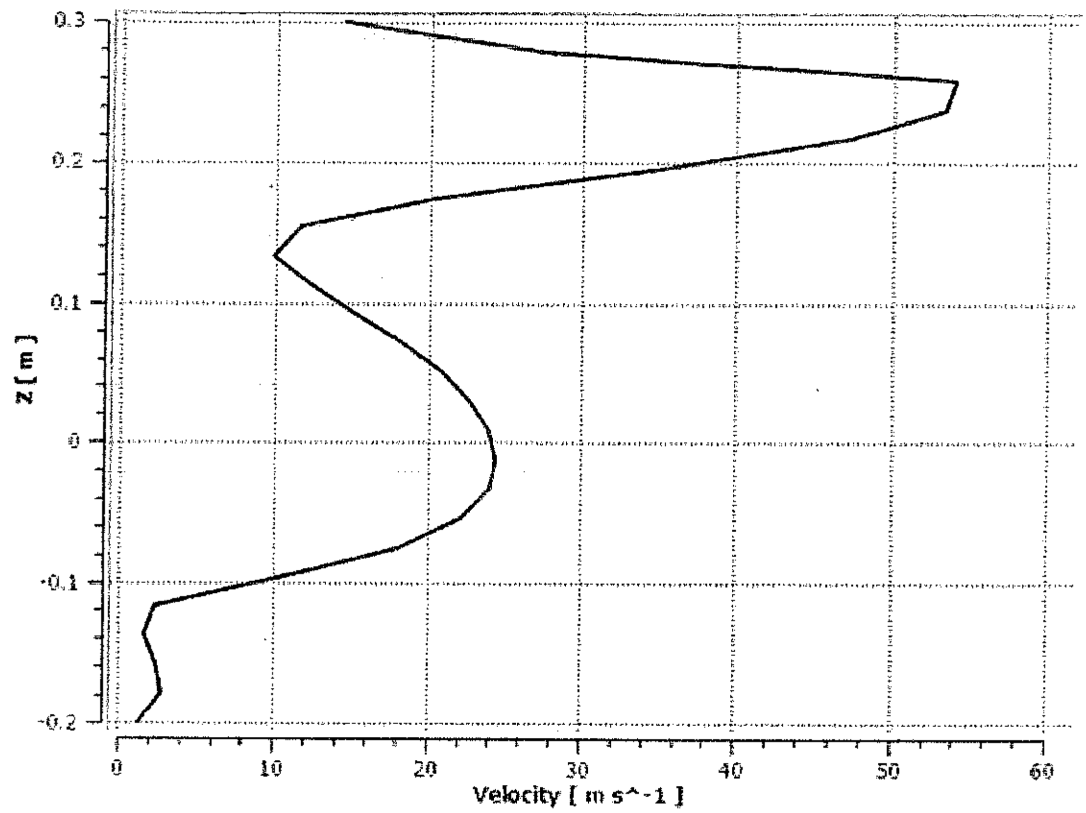


Figure 28: Model 1B Radiator outlet Velocity

For Model 2A

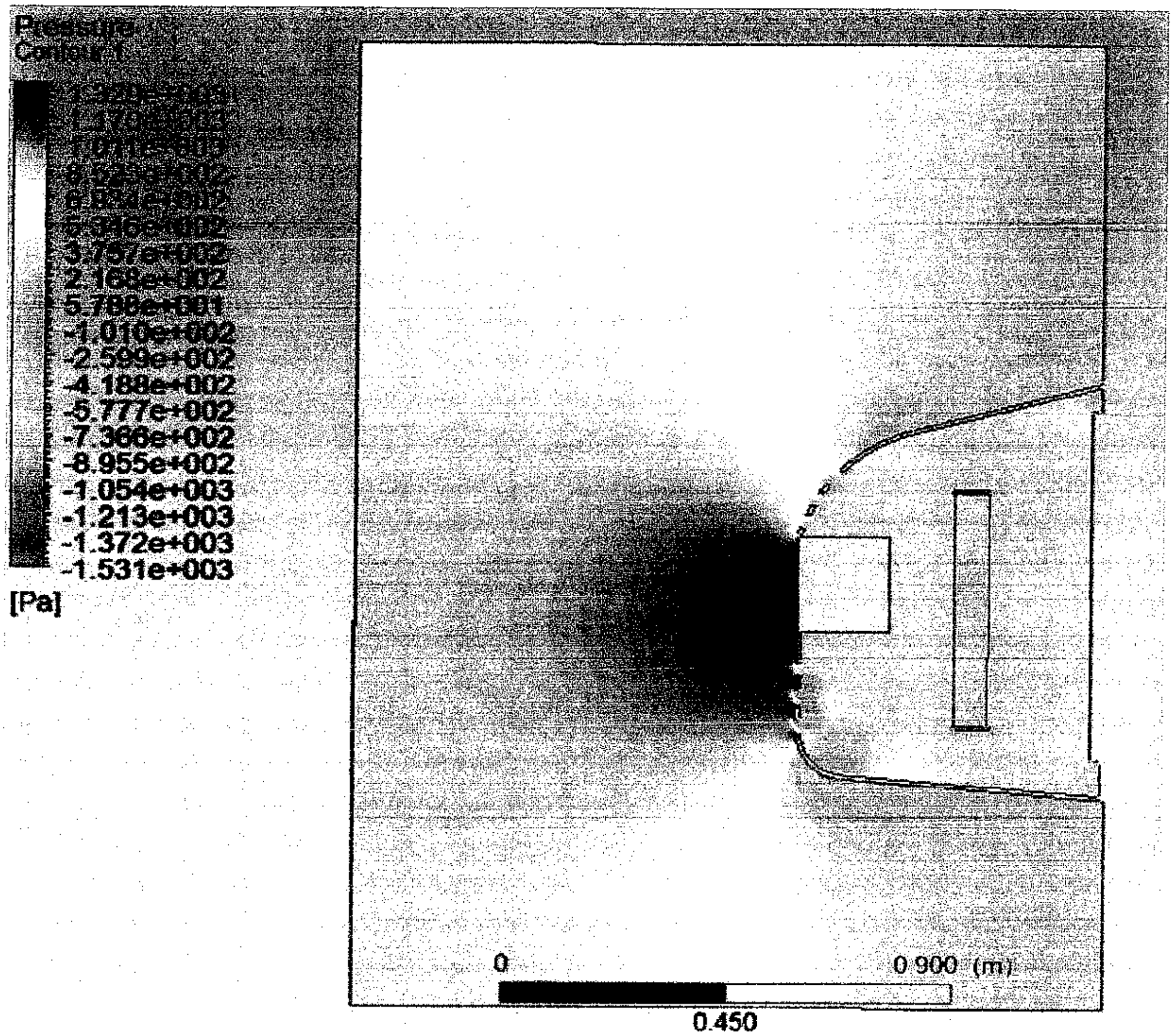


Figure 29: Model 2A Pressure Contours

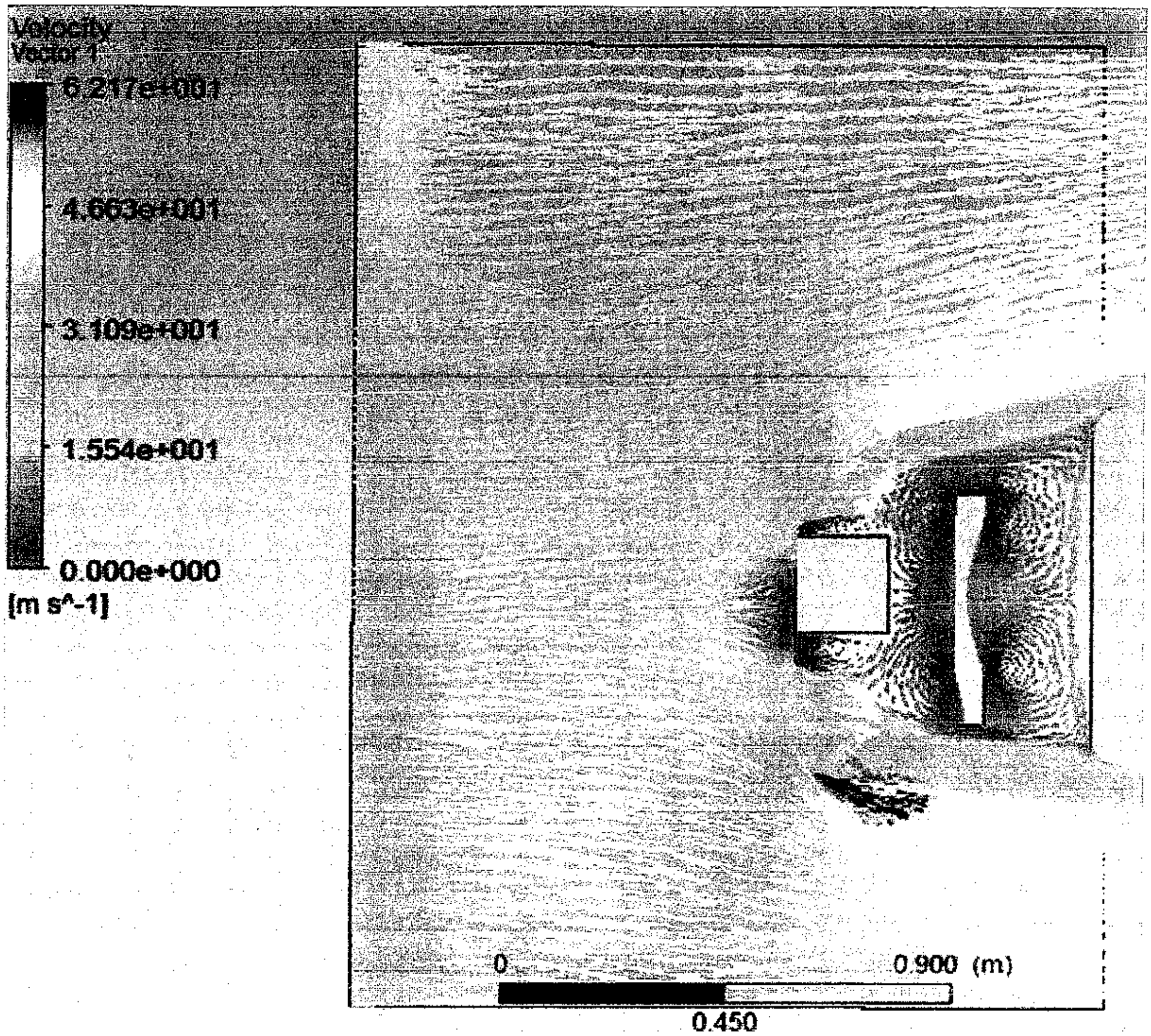


Figure 30: Model 2A Velocity Vectors

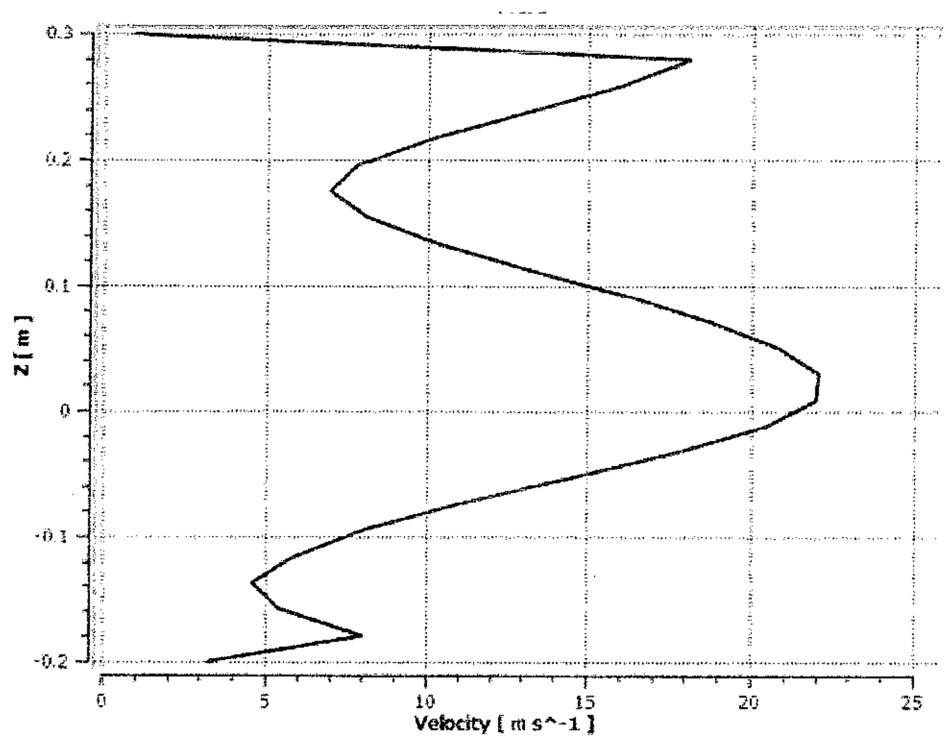


Figure 31: Model 2A Radiator outlet Velocity

For model 2B

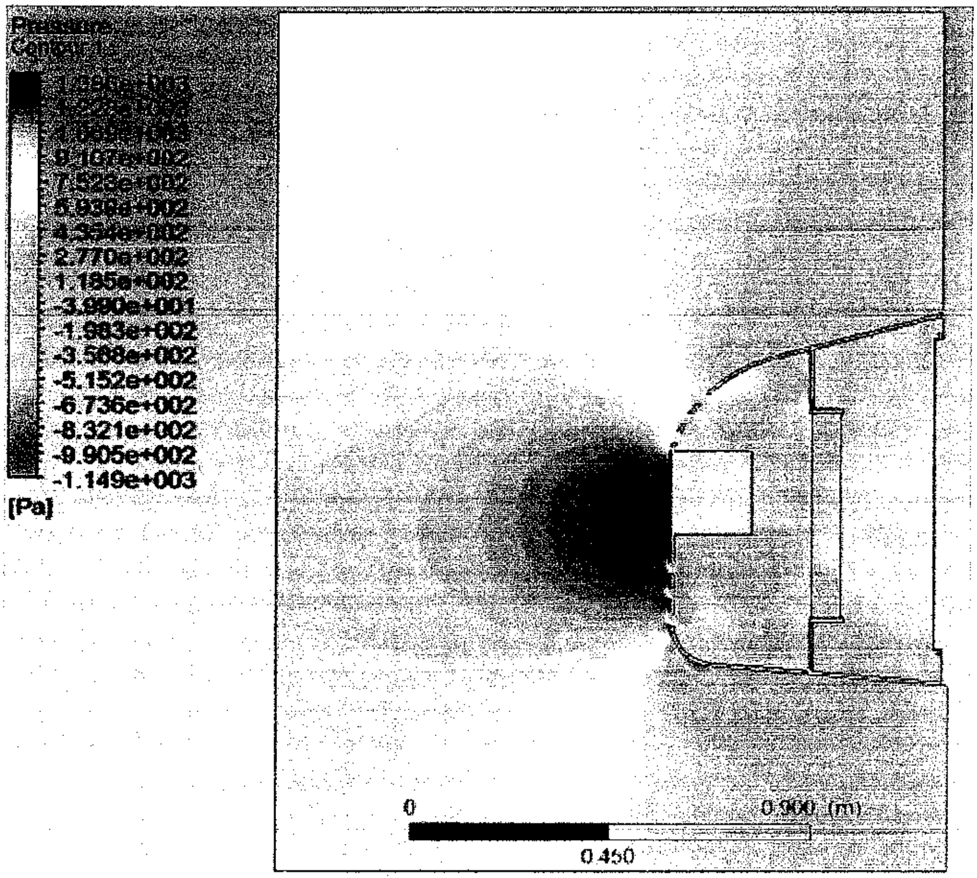


Figure 32: Model 2B Pressure Contours

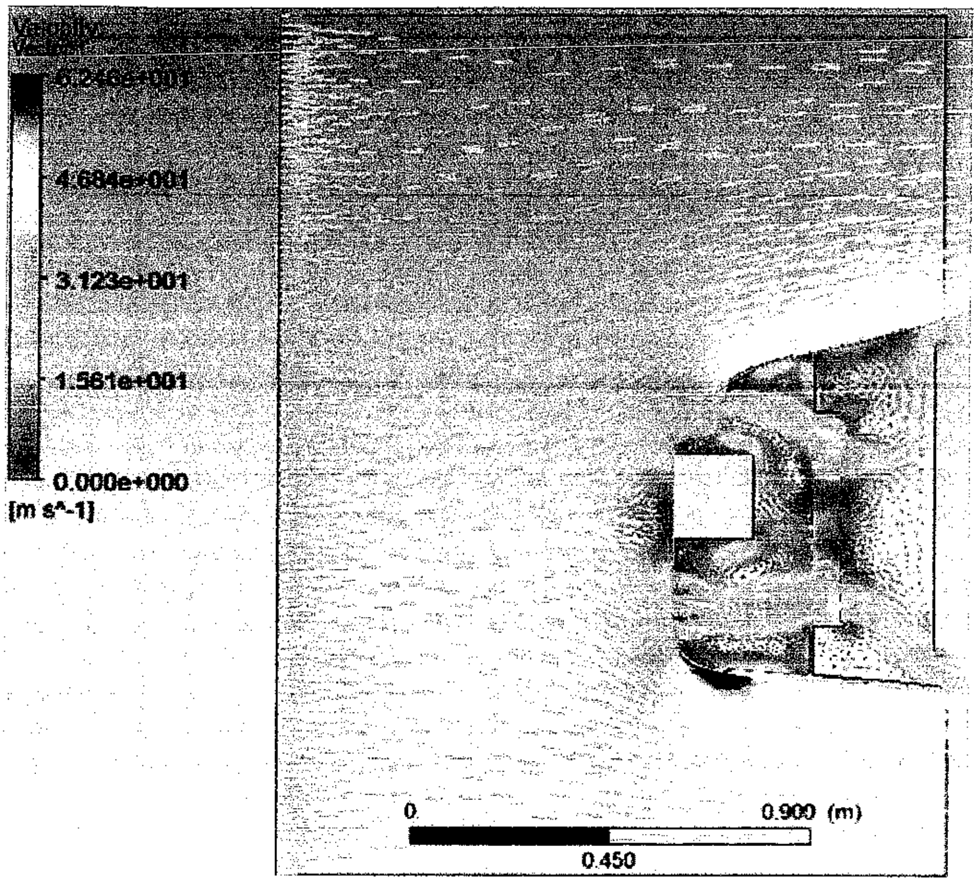


Figure 33: Model 2B Velocity Vectors

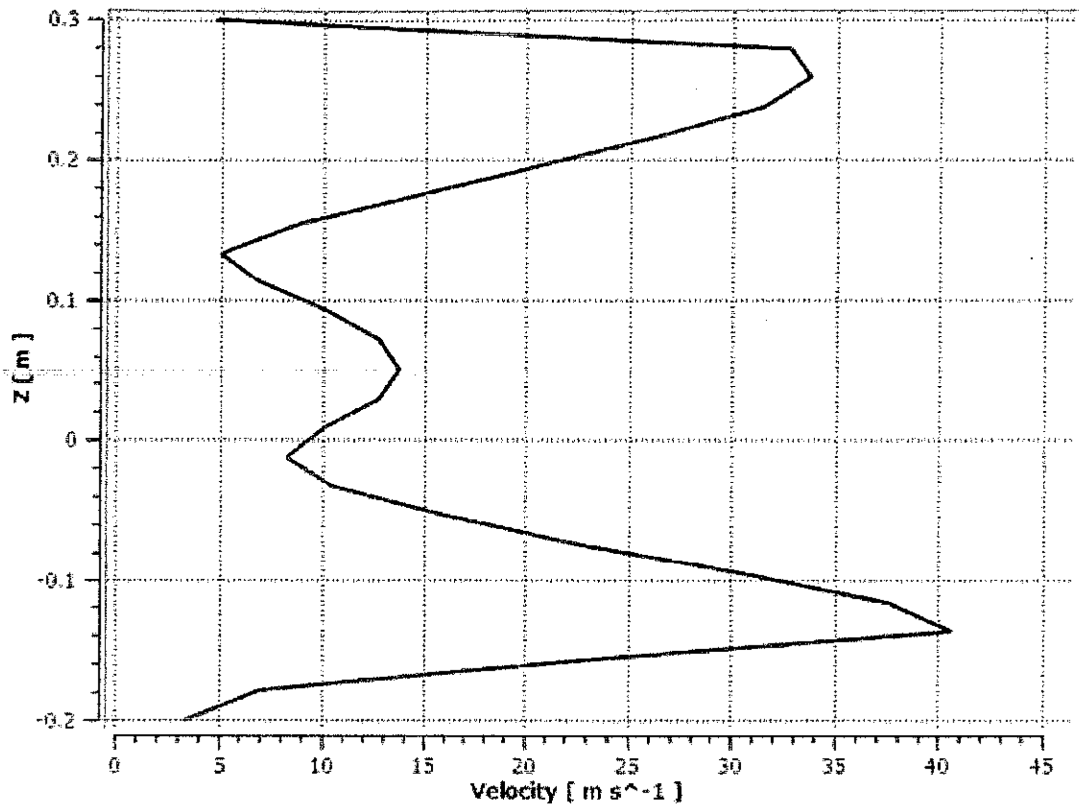


Figure 34: Model 2B Radiator outlet Velocity

The overall forces were computed by CFX and tabulated as below

Model	Overall Drag	Average Velocity Radiator outlet (m/s)	Standard Deviation (m/s)
1A	2.6410e+00	9.77926	14.93043
1B	3.0830e+00	12.1185	11.29366
2A	2.9739+00	17.76135	7.53701
2B	3.3032e+00	19.0794	6.25628

4.2 Discussion

Models 1A and 2A are similar and the air inside the hood is not dircted except for the fact that 2A has an extra intake

This extra intake from the above results improves the airflow by 81.6 % and airflow is also more uniform (less deviation).

But the drag increases by 12.6%

Models 1B and 2B are similar (the airflow is directed toward) except that 2B has an extra intake.

Average velocity increases by 57.4%

Here the drag increases further 7.1%

Further more in both the models where there are 2 intakes (1B and 2B)and the airflow is experiences more resistance being forced inside the hood thus increasing the Drag considerably

Here we see that the air flow through the radiator shows the maximum improvement but at the expense of increased Drag as well

Research also showed that reducing the front area of the radiator was detrimental to the performance of the car. [8] Increasing the number of fins also did not necessarily help in heat transfer since it stopped the airflow through the radiator, although street car racers do replace the factory brass radiators for better conducting aluminum radiators with multiple cores. [9]

Chapter 5: CONCLUSION AND RECOMMENDATION

Conclusion

According to CFD analysis done, the best way to improve upon a radiator design is by maximizing the airflow going in through the front of a car and making sure the airflow goes **in through** the radiator. A change in design of the front intake also may help improve airflow but drag must be taken into consideration [5]. Also fan covers and cowl improve in airflow efficiency.

Recommendation

The best way to improve the airflow and drag at the same time would be to improve exterior design. Currently the exterior is vertically flat to improve this we might make it rounder and smoother so that airflow is directed past the hood as well as inside. This may improve the drag while keeping airflow efficiency at its maximum.

The research plays an important role in any project to be carried out. This has proved to form a basis through which the final simulations were based. The simulation project has given a clear indication of how the fluid is flowing inside the radiator. This has in turn enabled us to visualize and predict airflow pressure the industrial users and designers to be able to design equipment that is efficient and result in the least amount of drag due to radiator. Certain parameter might be recommended to be changed whereas some will have to be left as they are for the best performance of the Radiator.

REFERENCES

[1]. Sridhar Maddipatla, Coupling of CFD and Shape Optimization for Radiator Design, 2008.

[2]. www.howstuffworks.com

[3]. Prediction of Aerodynamic Drag and Lift of the Nubira Hatchback Vehicle, N. Haidar, E, 2008.

[4]. Notes, Professor Hussain A. K. (2009)

[5]. Modeling of Vehicle Thermal Systems, D.J. Butler, S.P. Stevens, and VTMS4 Paper L07/C543-57.

[6]. computational Modeling of the Underbonnet Cooling Flow Characteristics of the New Daewoo Van, N. Haidar, E. Draper, 2nd MIRA International Conference on Vehicle Aerodynamics, October 1998.

[7]. Amtec Engineering Inc. Tecplot Users Manual Version 7,Amtec Engineering, 1998

[8]. Watkins, S. Wind-tunnel Modeling of Vehicle Aerodynamics; with Emphasis on Turbulent Wind effects on Commercial Vehicle Drag, PhD thesis, RMIT University, Melbourne, 1990.

[9]. Zhigang Yang, Jeffrey Bozeman and Fred Z. Shen, James A. Acre, "CFRM Concept at Vehicle Idle Conditions", SAE-2003-01-0613.

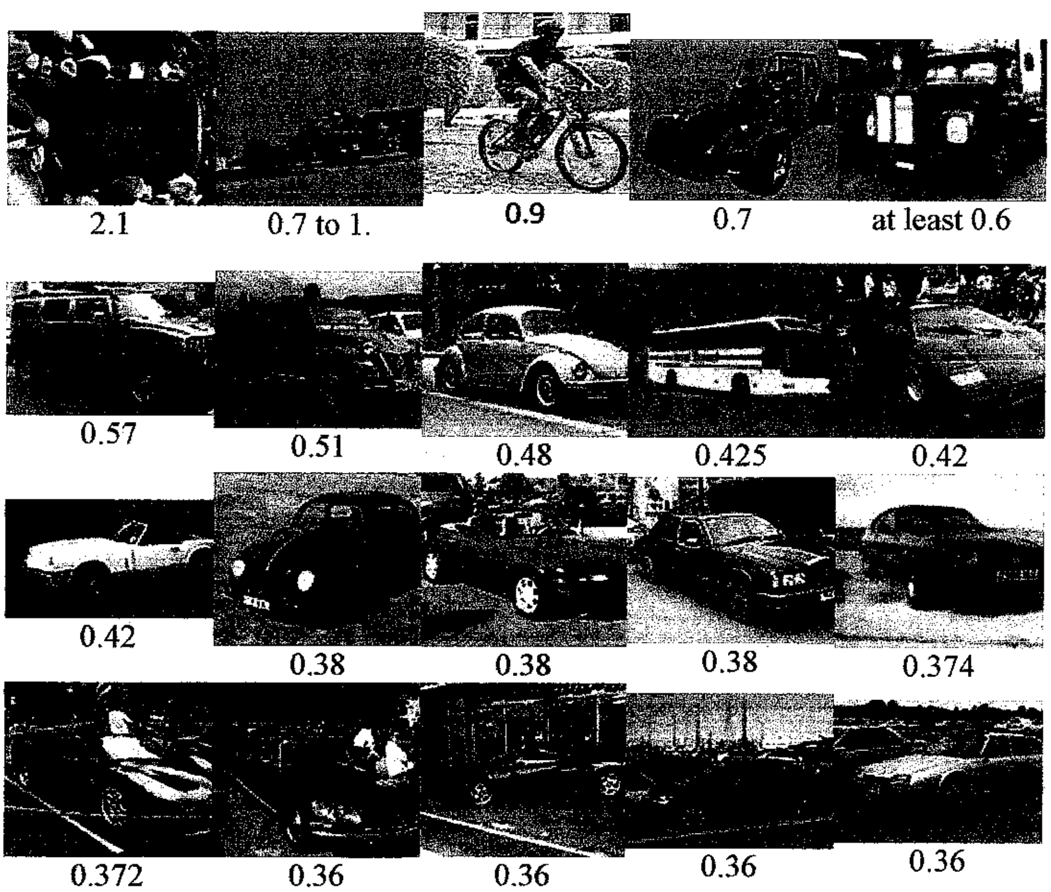
[10]. Ansys 6.1 User's Guide, Ansys Inc 2003-01-25.

Appendix A

Typical values and examples

The average modern automobile achieves a drag coefficient of between 0.30 and 0.35. SUVs, with their typically boxy shapes and larger frontal area, typically achieve a C_d of 0.35–0.45. A very gently inclined windshield gives a lower drag coefficient but has safety disadvantages, including reduced driver visibility. Certain cars can achieve figures of 0.25–0.30, although sometimes designers deliberately increase drag in order to reduce lift. Some examples of C_d follow. Figures given are generally for the basic model. Some "high performance" models may actually have higher drag, due to wider tires and extra spoilers.

Selected photographs with their C_d

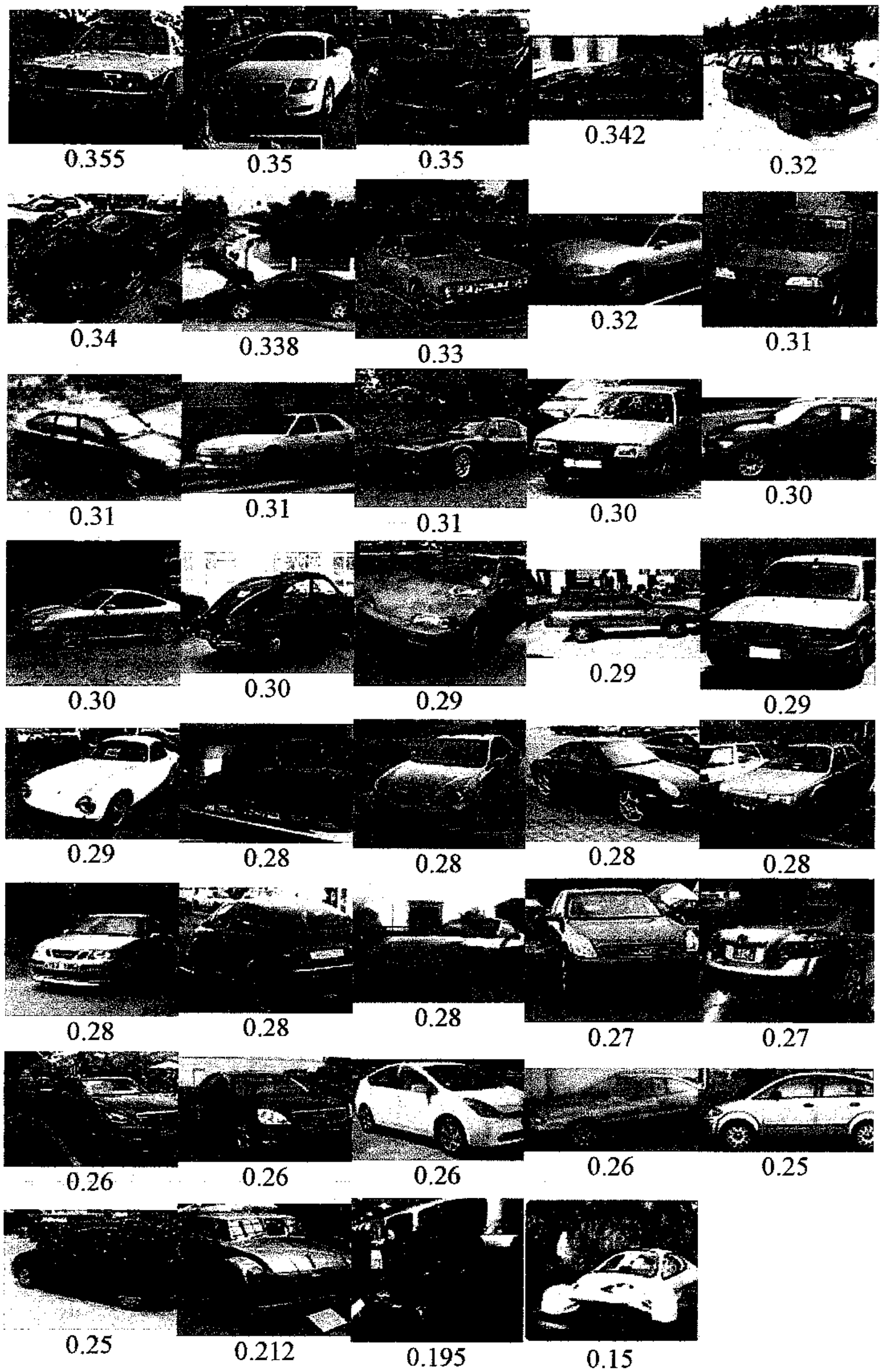


2.1 | 0.7 to 1. | 0.9 | 0.7 | at least 0.6

0.57 | 0.51 | 0.48 | 0.425 | 0.42

0.42 | 0.38 | 0.38 | 0.38 | 0.374

0.372 | 0.36 | 0.36 | 0.36 | 0.36


0.355 0.35 0.35 0.342 0.32
0.34 0.338 0.33 0.32 0.31
0.31 0.31 0.31 0.30 0.30
0.30 0.30 0.29 0.29 0.29
0.29 0.28 0.28 0.28 0.28
0.28 0.28 0.28 0.27 0.27
0.26 0.26 0.26 0.26 0.25
0.25 0.212 0.195 0.15

Appendix B

Model Properties and Formulae to be used for simulation

<u>Density</u>

$$\rho = \frac{P}{(287.0856)\,(T + 273.15)}$$

Where ρ is air density in Kg/m³, P is 101325 Pa (atmospheric pressure), and T is air temperature in °C.

<u>Viscosity</u>

$$\mu = 1.8402 \times 10^{-5} \left(\frac{T + 273.15}{298} \right)^{0.78}$$

Where μ is air viscosity in Kg/m-s and T is air temperature in C.

Apendix B

Gant-Chart