FLOW ANALYSIS OF OVERTAKING VEHICLES: CFD STUDY

by

Ahmed Mohamed Saad

Dissertation submitted in partial fulfillment of
the requirements for the
Bachelor of Engineering (Hons)
(Mechanical Engineering)

JUNE 2010

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

FLOW ANALYSIS OF OVERTAKING VEHICLES: CFD STUDY

by

Ahmed Mohamed Saad

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,

(Dr Ahmed Maher Said Ali)
Project Supervisor
(for Dr. Ahmed Maher)

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

JUNE 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.

____

(Ahmed Mohamed Saad)

ABSTRACT

The aerodynamic forces occurring when one road vehicle overtakes another are to be investigated using two-dimensional (2D) computational fluid dynamics. Although a lot of researches have been carried on the aerodynamics of the flow of overtaking vehicles, very little information is available on flow around full scale vehicles. Increasing the size of models 10 times would results in the increase of the mesh size, making it more expensive and requires higher performance hardware. This research studied the aerodynamic forces by the quasi static approach using CFD analysis; it also laid the grounds for further transient and 3D investigation on the full scale model and discussed their feasibility.

The purpose of this report is to discuss the outcomes of my final year project with some guidance on how to reach these results for further investigations.

ACKNOWLEDGEMENTS

Praise be to Allah, The Most powerful The Most Gracious and The Most Merciful for His endless blessings throughout my life and the success He granted me in accomplishing this project.

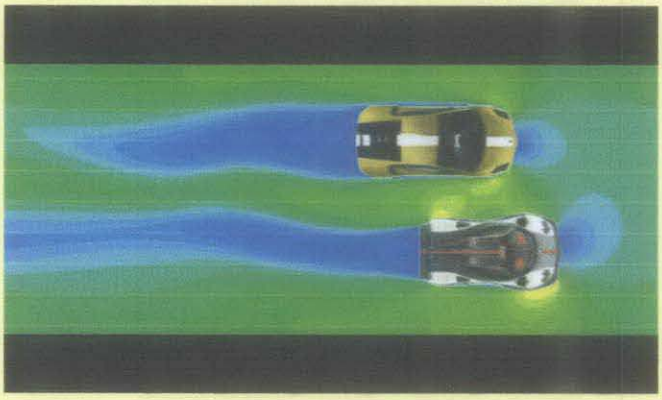
Throughout this whole duration of this project, I've learnt a lot and gained priceless experiences. I would like to acknowledge the following people for their help, advice, support and encouragement throughout this duration. Without them, this project could never have been done with so much content.

I could never have come so far without the constant supervision and support of my supervisor, Dr Ahmed. Thank you for your constant guidance, advice and encouragement throughout this whole year. Your help is greatly appreciated.

To my friends both from UTP and elsewhere, thanks for always being there for me. Thank you for sharing the knowledge with me.

Finally, to all who have helped me in one way or another, thank you.

FLOW ANALYSIS OF OVERTAKING VEHICLES: CFD STUDY



by

Ahmed Mohamed Saad

Universiti Teknologi PETRONAS
Bachelor of Engineering (Hons)
(Mechanical Engineering)

JUNE 2010

Table of Contents

Table of Contents

Abstract

The aerodynamic forces occurring when one road vehicle overtakes another are to be investigated using two-dimensional (2D) computational fluid dynamics. Although a lot of researches have been carried on the aerodynamics of the flow of overtaking vehicles, very little information is available on flow around full scale vehicles. Increasing the size of models 10 times would results in the increase of the mesh size, making it more expensive and requires higher performance hardware. This research studied the aerodynamic forces by the quasi static approach using CFD analysis; it also laid the grounds for further transient and 3D investigation on the full scale model and discussed their feasibility.

The purpose of this report is to discuss the outcomes of my final year project with some guidance on how to reach these results for further investigations.

Introduction

Background of Study

As one vehicle passes another vehicle, the air flow between the vehicle change instability of the vehicles due to the change in pressure, drag and lift forces.

Background Theory

Bernoulli's equation: Bernoulli's principle states that for an inviscid flow, an increase in the speed of the fluid occurs simultaneously with a decrease in pressure.

In most flows of liquids, and of gases at low Mach number, the mass density of a fluid parcel can be considered to be constant, regardless of pressure variations in the flow. For this reason the fluid in such flows can be considered to be incompressible and these flows can be described as incompressible flow. Bernoulli performed his experiments on liquids and his equation in its original form is valid only for incompressible flow. A common form of Bernoulli's equation, valid at any arbitrary point along a streamline where gravity is constant, is:

$$\frac{v^2}{2} + gz + \frac{p}{\rho} = \text{constant}$$

Where:

v is the fluid flow speed at a point on a streamline,

g is the acceleration due to gravity,

z is the elevation of the point above a reference plane, with the positive z-direction pointing upward — so in the direction opposite to the gravitational acceleration,

p is the pressure at the point, and

ρ is the density of the fluid at all points in the fluid.

Drag force: refers to forces that oppose the relative motion of an object through a fluid

CFD: stands for computational fluid dynamics. The analysis of the flow using computer software.

Problem Statement

Although a lot of researches has been carried on the aerodynamics of the flow of overtaking vehicles, very little information is available on flow around full scale vehicles. Increasing the size of models 10 times would results in the increase of the mesh size, making it more expensive and requires higher performance hardware.

Objectives

The objective of this research is to understand and analyze the flow around two vehicles and its effect on overtaking vehicles.

Scope of Study

The scopes of study include the basic principles of external flow. These principles include the stream line of the flow and the forces acting on the body due to the flow.

The fields of the scope are summarized as following:

- Research on external flow on road vehicles.
- Research on air disturbance forces
- Study the airflow during vehicle overtaking.
- Using computer modeling and analysis.

Literature Review

Yamamoto and Nakagawa (1997) conducted a dynamic study on 1/10th scale models and concluded that, for a velocity ratio of $k<0.25$, dynamic effects could be neglected and the problem could be modelled statically. Telionis et al. (1979) conducted tow tank analysis on a greater range of velocity ratios and suggested that dynamic effects were insignificant up to $k<0.4$.

The unsteady computational fluid dynamics (CFD) analysis of Okumura and Kuriyama (1997) was conducted at higher velocity ratios and concluded that dynamic effects were important. For a velocity ratio of $k = 0.5$ the peak yawing moment of the vehicle being passed was almost 100% higher than that predicted using the quasi-steady approach.

The experimental study of Gillieron and Noger (2004) combined analytical methods and CFD with use of the new experimental overtaking test bench at the Institute Aerotechnique de Saint Cyr l'Ecole. The relative velocity range tested was 0–10 m/s (0–22 mph) on 1/5th scale Ahmed models with a free streamvelocity of 30 m/s (68 mph). Results indicated that the side force at a relative velocity of 10 m/ s (22 mph, $k = 0.32$) was 60–120% higher than that predicted by the quasi-steady analysis. The latest report of Noger et al. (2005) suggests that the aerodynamic forces on overtaking road vehicles become velocity dependent when the ratio of the relative velocity to the overtaken vehicle velocity (k) is greater than 0.2. Data presented for a velocity ratio of $k\approx0.5$ showed aerodynamic behavior differing significantly from that predicted using quasi-steady analysis.

It is widely recognised that crosswinds can adversely influence vehicle handling and stability and create potentially hazardous driving conditions. However, little work has been completed on the influence of a crosswind on overtaking manoeuvres. Most existing studies concentrate on vehicles in isolation such as Docton's (1996) study on the impact of transient crosswinds on an individual passenger vehicle. Docton found that flow separation from the leeward side of the vehicle took a finite amount of time to develop as the vehicle moved into the crosswind. This delay in separation was accompanied by exaggeration of low-pressure regions, resulting in aerodynamic forces exceeding those predicted using steady analysis by 40–60%. Telionis et al. (1979) gave some consideration to the impact of a crosswind on the aerodynamic forces encountered by a small car passing a large truck. Their study on crosswinds at angles up to b ¼ 241 indicated increases in the drag and side force of 100–150% as the car emerged into the crosswind from the leeward wake of the truck. In the recent experimental study by Noger et al. (2005) it was observed that for yaw angles of b = 7101 there was considerable interaction between the overtaking vehicle and the wake of the overtaken vehicle. The combination of crosswind flow structures and the transient flow conditions occurring during overtaking maneuvers may therefore significantly amplify the aerodynamic forces on the vehicles and further reduce vehicle stability. However, at present the limited existing work means these flow conditions are not sufficiently understood.

Recently two studies were conducted by R.J. Corin, University of Durham and Nur Liyana, Universiti Teknologi Petronas to analyze the transverse flow of overtaking vehicles 1:10 scaled down. These CFD studies were to analyze the air forces on the overtaken vehicles due to the transverse flow and to compare the results from the quasi steady approach with the unsteady flow approach.

Another study was conducted on inter vehicle spacing. Inter-vehicle spacing on highways is considered and an analysis of spacing is presented, deduced from data from an instrumented highway. Vehicle drag reductions arising from close spacing are discussed and drag and lift data from wind-tunnel tests on two co-linear Ahmed bodies (representative vehicle shapes able to replicate typical car airflow, configured with 301 slant back angles) are given. Inter-body, non-dimensional spacing was varied from 0.1

to 4.0, based on vehicle length. Surprisingly, significant drag increases were found for the rear Ahmed body for spacing of 0.1–1.0, when compared to the drag of the body in isolation. For greater spacing, the drag of the rear body fell below the value of the isolated case, up to the maximum spacing considered. The lift coefficient of the rear body was also found to be very sensitive to spacing. It was concluded that the effect of the strong vortex system arising from the slant back was the cause of the drag and lift changes of the rear vehicle.

Another paper provides an overview of some of the CFD methods developed by the Team for Advanced Flow Simulation and Modeling. The paper also provides many examples of three-dimensional flow simulations carried out with these CFD methods and advanced parallel supercomputers. The methods and tools described in that paper include: stabilized finite element formulations; formulations for flows with moving boundaries and interfaces; mesh update methods; iterative solution techniques for large nonlinear equation systems; and parallel implementation of these methods. The paper's target is to be able to address effectively certain classes of flow simulation problems. These include: unsteady flows with interfaces; fluid-object interactions; fluid-structure interactions; airdrop systems; aerodynamics of complex shapes; and contaminant dispersion.

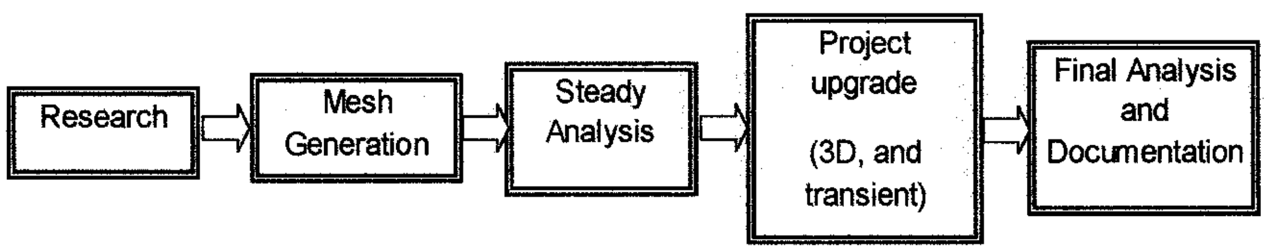
Two more studies were concerned with the effect of wings and spoilers on the car performance with the effect of the drag and lifting forces generated by those add-ons. The first study was done on the formula Mazda front wing to test its affect on the drag and lift forces according to different angles of attack. The aim of that study was to provide information about the front wings for that sports car. The second study dealt with 5 different designs of rear spoilers tested on 3D CFD model. The car model used was a Honda S2000; and the aim of the study was to test the different spoilers to observe the differences in drag and to select the best one in preference of the better fuel consumption.

In summary, although the impact of relative velocity on the aerodynamic forces on overtaking road vehicles have been investigated on a number of occasions the results remain inconclusive. There has also been little work investigating the fundamental dynamic effects associated with overtaking in a crosswind. The present work therefore uses two-dimensional (2D) CFD to overcome the practical problems associated with wind tunnel testing and provide a visualization of the changes occurring in the flow-field during overtaking maneuvers. The CFD method was used to compare the quasi-steady and the unsteady simulations to clarify the significance of dynamic effects on the overtaking maneuver and identify the limitations of the quasi-steady modeling approach. The scope of this project extends beyond existing work to analyze the transient aerodynamic effects associated with a crosswind and provide further fundamental understanding of their impact on passing maneuvers.

Methodology

In this chapter the author will discuss the processes involved in this research, the list of software to be used will be stated and the time line would be discussed

The methodology used in this project includes research for better understanding (Literature Review), Preliminary Design and Numerical testing (CFD simulation).



ID	Task Name	Week 1	2	3	4	5	6	7	8	9	10	11	12	13	14
1															
2	Final year project 1														
3															
4	primary project work														
5	(literature review, data BG, work with digitizer)														
6															
7	secondary project														
8															
9	preliminary research														
10	research on relevant data														
11	project planning														
12															
13	car modeling in CAD														
14	exporting models to gambit and mesh generation									X					
15															
16	transfering the mesh to fluent												X		
17	Testing the meshes with iterations														X
18															

ID	Task Name	Week 16	17	18	19	20	21	22	23	24	25	26	27	28	29
19															
20	Final year project 2														
21															
22	Modifying the mesh design, and boundary conditions														
23	Running iterations while car are neck and neck			X											
24															
25	Changing the vehicle positions and running the simulation														
26	Recording and documenting the data							X							
27															
28	Putting the basics for future modifications														
29															
30	Creating 3-D vehicle graphics and mesh														
31	Testing the 3D flow capabilities and requirements										X				
32															
33	Creating UDF and testing for transient flow analysis														
34															

The following sections discuss on research, preliminary design and numerical testing.

RESEARCH (LITERATURE REVIEW)

Literature review includes research on CFD analysis, importance of aerodynamic characteristic of road vehicles and research done on testing the flow around cars by using computational or experimental method.

VEHICLES GRAPHICS DESIGN

The vehicle models designs are derived from the designs of race cars.

Model 1

- Based on Lamborghini Gallardo layout.
  - Width: 1.9m
  - Length: 4.4m

Model 2

- Based on Pagani Zonda F layout
  - Width: 2.1m
  - Length:4.3m

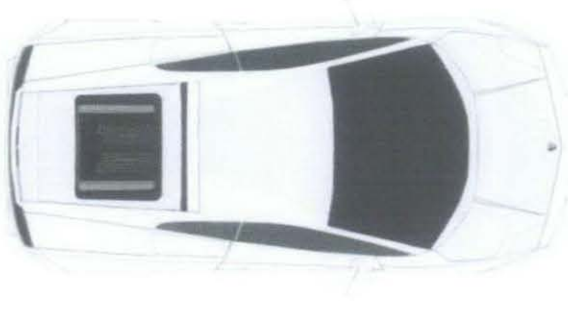


Figure 1: car model 1

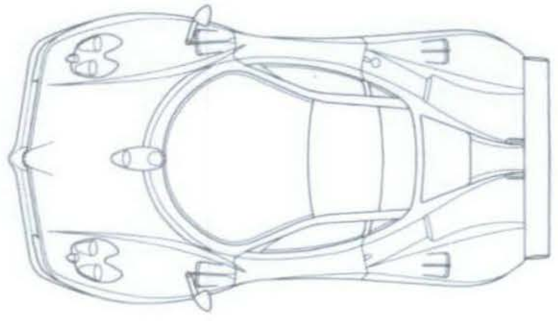


Figure 2: car model 2

Mesh Generation

The mesh is divided into 2 main sections, each is divided into 2 more sections (boundary, and boundary extension), where the step size of the mesh in the extension is the double of that in the main boundary.

Dimensions:

- Length = 30m
- Width = 16m

Main boundary: 5m, boundary extension: 3m (each)

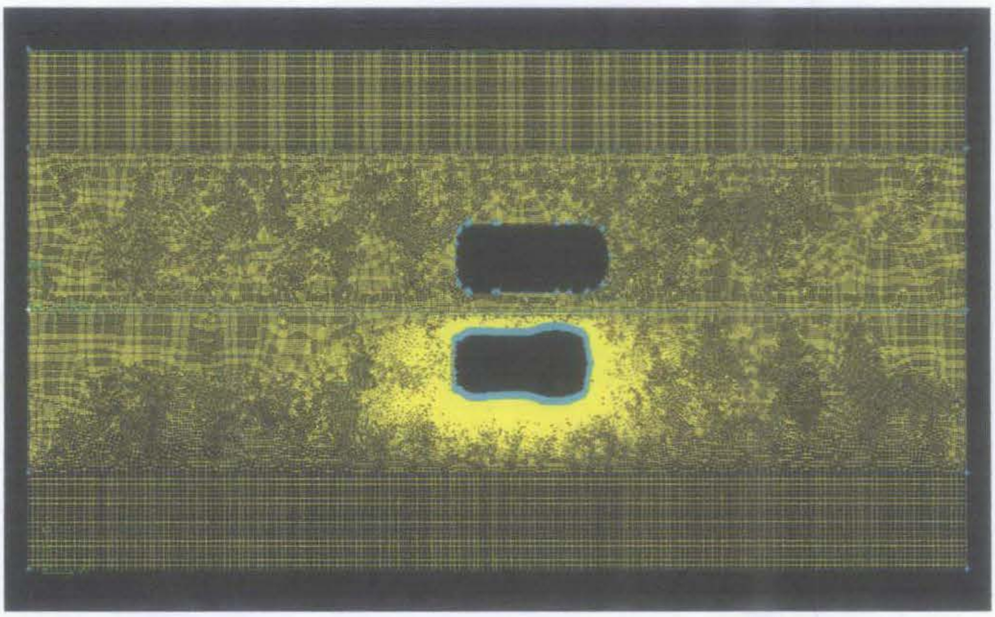


Figure 3: CFD mesh at neck and neck position

The above figure shows mesh with the step size 7 in the boundary mesh and 15 in the extension.

UDF programming

- Programming the UDF With C++ compiler
- Integrate the UDF to the CFD software

Software used in conducting this study:

AutoCAD: a computer aided design software used to design the models, the output files are saved in DWG format.

Solid Works: another computer aided design software used to enhance the designs and to convert the DWG format to another format (IGES) which is readable by the mesh generation software.

Gambit: geometry modeling and grid generation software, used to generate the mesh used in the analysis

Fluent: numerical simulation software and a computational fluid dynamics anlyser, used to analyze the case study.

Microsoft Visual Studio: to compile and integrate the UDF into Fluent.

CFD analysis

Steady analysis

Setup and Solution

Start the 2D version of FLUENT.

Step 1: Grid

1. Read the grid file valve.msh.

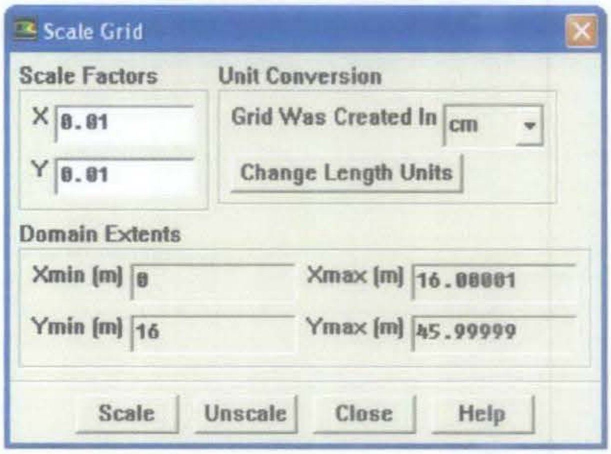
   File□–→ Read□–→Case...

2. Check the grid.

   Grid□–→Check

Note: *You should always make sure that the cell minimum volume is not negative, since* FLUENT *cannot begin a calculation if this is the case.*

3. Scale the grid.



Grid□-→Scale...

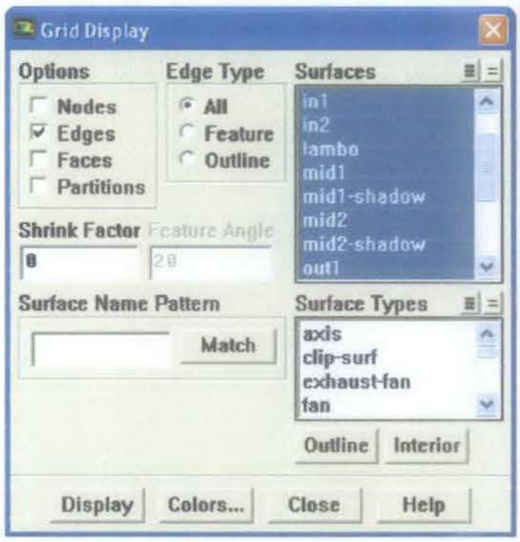
(a) Under Unit Conversion, select in from the drop-down list to complete the phrase Grid Was Created In cm (centimeters).

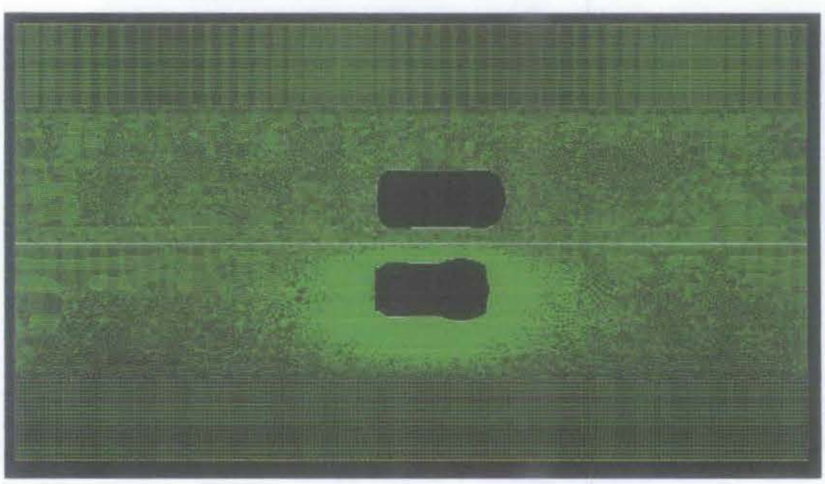
(b) Click Scale to scale the grid.

(c) Click Change Length Units to set centimeters as the working units for length, and then close the panel.

4. Display the grid

Display□–→Grid...

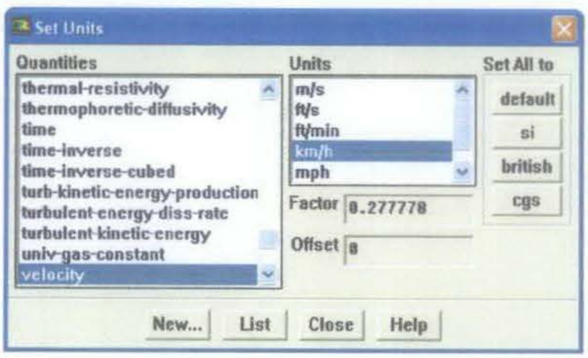




Step 2: Units

1. For convenience, define new units for velocity.

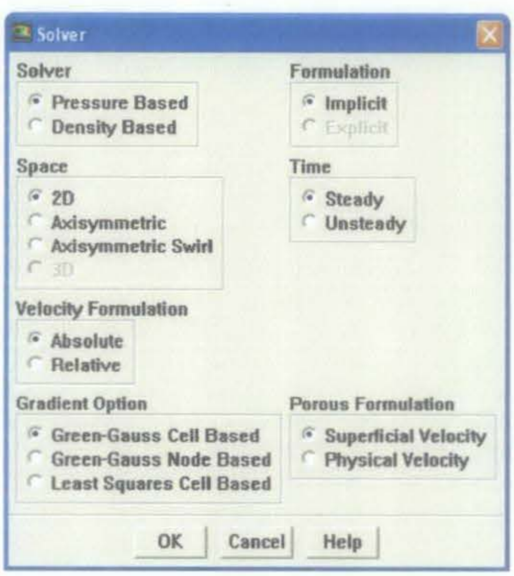
   Define□–→Units...



Step 3: Models

1. Set the solver settings.

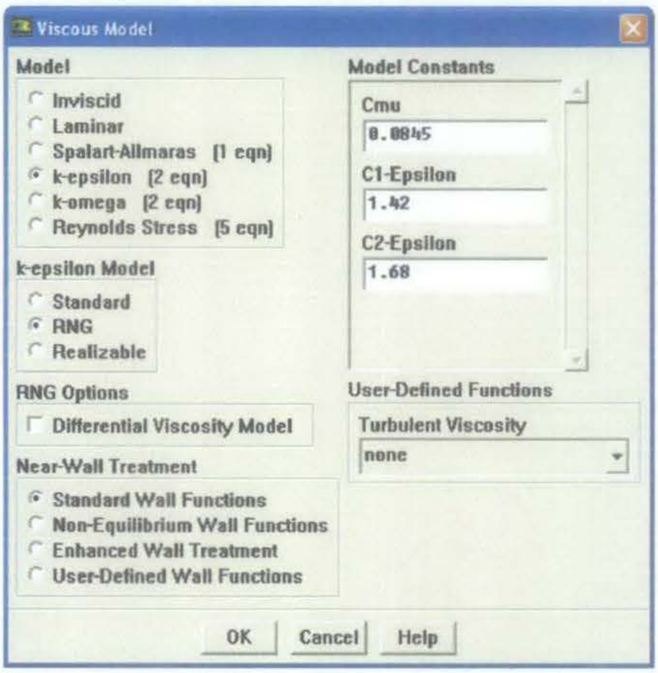
   Define□–→ Models□–→Solver...



   (a) Click OK to retain the default settings.

2. Enable the k-epsilon turbulence model.

Define□-→ Models□-→Viscous...



(a) Enable k-epsilon (2 eqn) in the Model list.

(b) Select RNG from k-epsilon mpdel

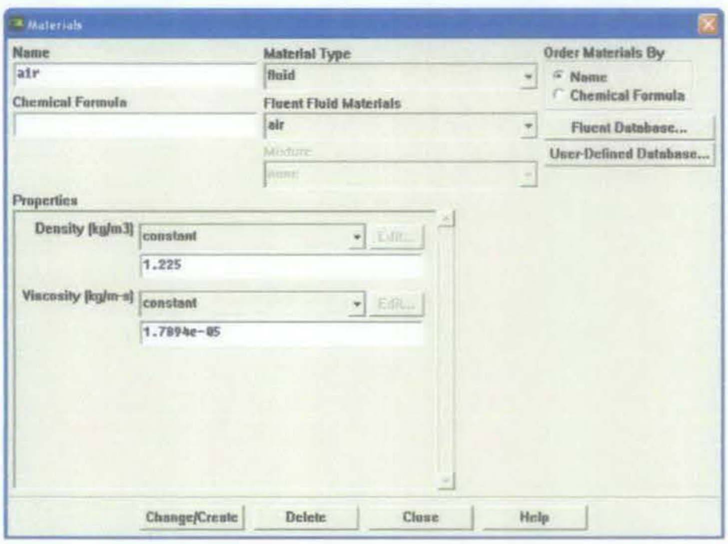
(c) Click OK to close the Viscous Model panel.

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.

Step 4: Materials

The default working fluid material in this problem is air.

Define□-→Materials...

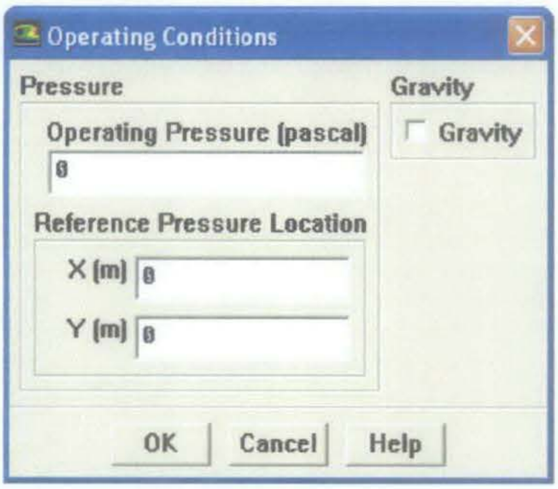


(a) Click OK to retain the default settings.

Step 5: Operating Conditions

Set the operating pressure to 0 pascal.

Define□-→Operating Conditions...

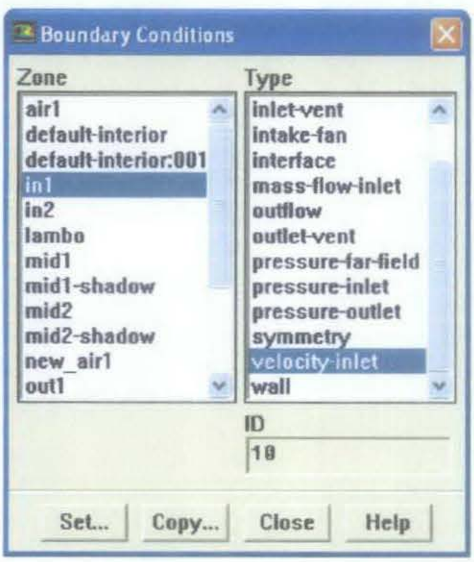


For this problem, you will work with absolute pressures.

Step 6: Boundary Conditions

Define□—→Boundary Conditions...

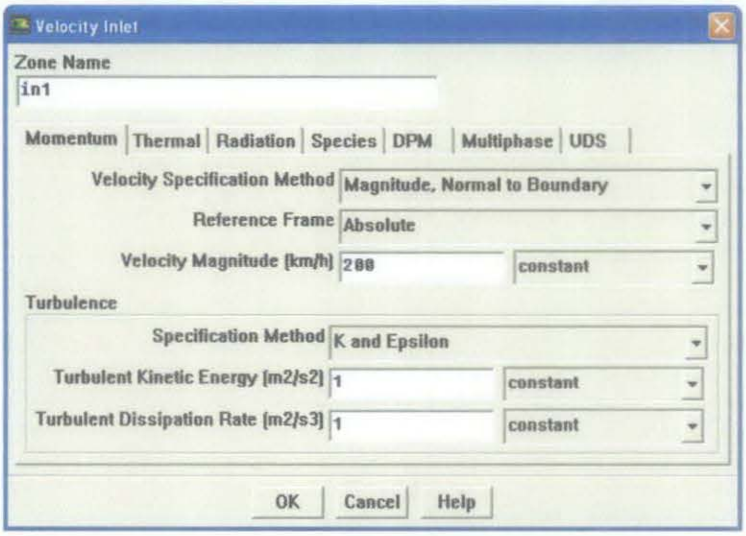
1. Set the boundary condition for velocity inlet



(a) Select in1 from the Zone list.

This Type *will be reported as* velocity-inlet.

(b) Click the Set... button.



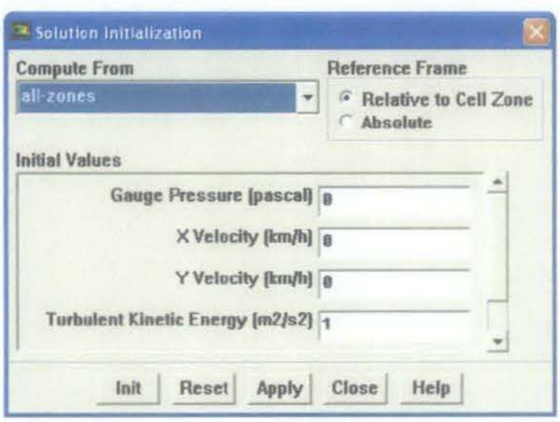
Step 7: Solution

1. Retain default solver settings for initial solution...

   Solve□–→ Controls□–→Solution...

2. Initialize the flow.

   Solve□–→ Initialize□–→Initialize...

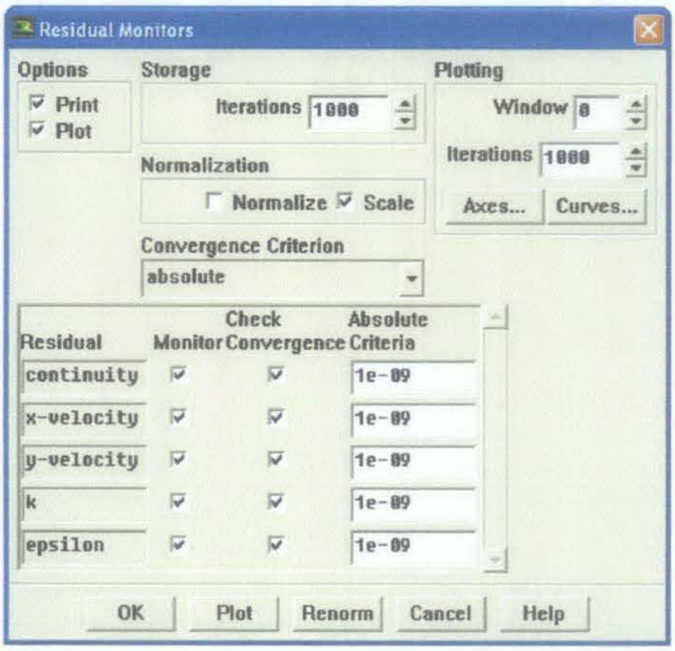


(a) Select all-zones from the Compute From drop-down list.

(b) Click Init and close the Solution Initialization panel.

3. Enable the plotting of residuals during the calculation.

Solve□—→ Monitors□—→Residual...

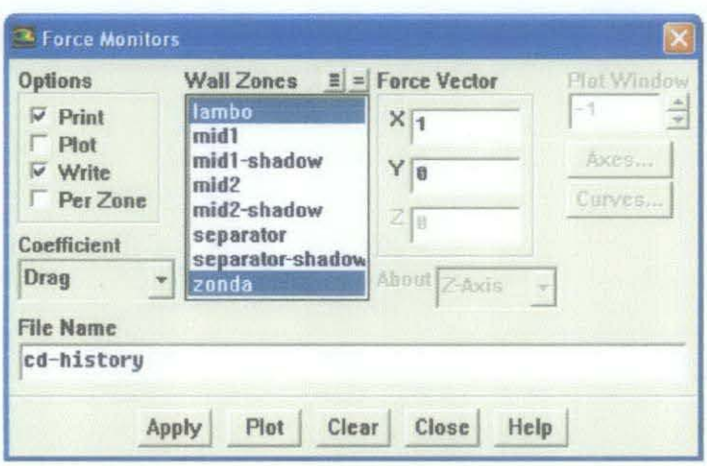


a) Enable Plot from the Options list.

(b) Click OK to close the Residual Monitors panel.

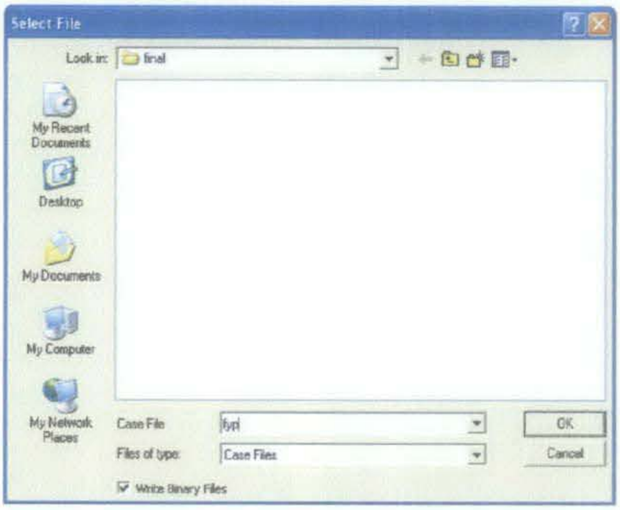
4. Enable the writing of forces (drag and lift) during the calculation.

Solve□—→ Monitors□—→ Force...



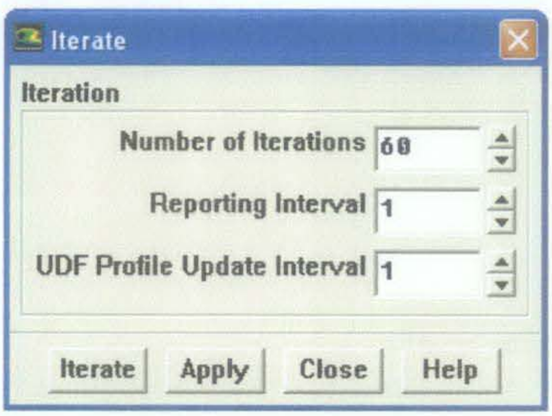
5. Save the case file (`fyp.cas`).

File —→ Write —→Case...



6. Start the calculation by requesting 60 iterations.

Solve —→Iterate...



Unsteady analysis

Write the UDF and compile it

```
#include <udf.h>

DEFINE_CG_MOTION(zonda, dt, cg_vel, cg_omega, time, dtime)
{
    /*real f = RP_Get_Real("motion-par/freq_s");*/
    /*real a = RP_Get_Real("motion-par/amplitude");*/
    real omega[];
    real vel[];
    /*real amplitude = .20; /* in inches */
   /* real conv = 0.0254;*/
    /*omega= 2.0*M_PI*f;*/
   /* vel = amplitude*conv*omega*sin(omega*time);*/
    /*vel = a*omega*sin(omega*time);*/
       vel=2.2;
    cg_vel[0] = vel; /* x-velocity*/
    cg_vel[1] = 0.0;
    cg_vel[2] = 0.0;
    NV_S (cg_omega, =, 0.0); /* no angular motion */
}
```

before you compile your source _le, the udf.h header _le will need to be accessible in your path, or saved locally within your working directory.

The location of the udf.h _le is:

path/Fluent.Inc/fluent6.+x/src/udf.h

For compiled UDFs on Windows systems, two Fluent Inc. files are required to build your shared UDF library: makefile_nt.udf and user nt.udf.

The file user_nt.udf has a user-modifiable section that allows you to specify source file parameters.

The procedure below outlines steps that you need to follow in order to set up the directory structure required for the shared library.

1. In your working directory, make a directory that will store your UDF library (e.g., libudf).

2. Make a directory below this called src.

3. Put all your UDF source _les into this directory (e.g., libudf\src).

4. Make an architecture directory below the library directory called ntx86 for Intel systems running Windows_(e.g., libudf\ntx86).

5. In the architecture directory (e.g., libudf\ntx86), create directories for the FLUENT versions you want to build for your architecture. (e.g., ntx86\2d and ntx86\3d).

6. Copy user nt.udf from

    path/Fluent.Inc/fluent6.+x/src/user nt.udf

to all the version subdirectories you have made (e.g., libudf\ntx86\3d).

7. Copy makefile nt.udf from

    path/Fluent.Inc/fluent6.+x/src/makefile nt.udf

to all the version subdirectories you have made (e.g., libudf\ntx86\3d) and rename it makefile.

After you have set up the directory structure and put the _les in the proper places, you can compile and build the shared library using the TUI.

1. Using a text editor, edit every user nt.udf _le in each version directory to set the following parameters: SOURCES, VERSION, and PARALLEL NODE.

   SOURCES = $(SRC)zonda.c

   VERSION = 2d

   PARALLEL_NODE = none

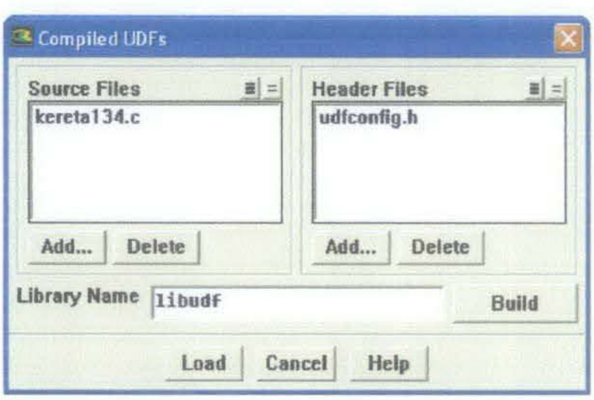
2. In the Visual Studio command prompt window, go to each version directory (e.g., \libudf\ntx86\2d\), and type nmake.

   C:\users\user_name\work_dir\libudf\ntx86\2d>nmake

Open the fluent software from the visual studio command prompt

Load the UDF into fluent

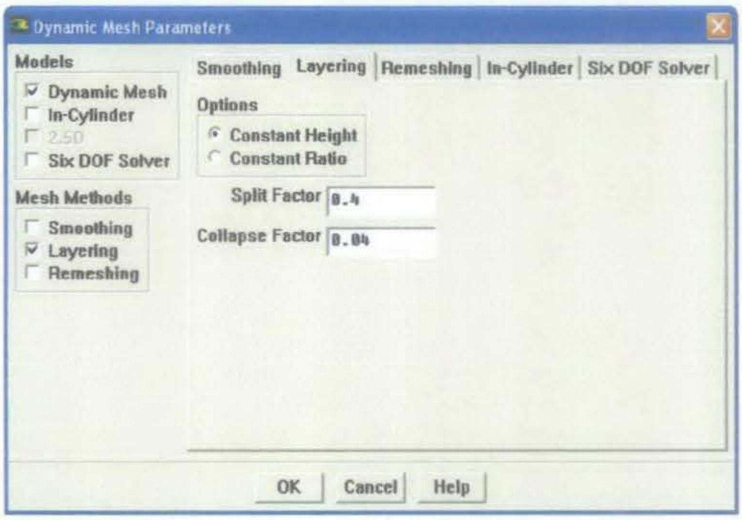
Define□–→ User-defined□–→functions –→ compiled



Add source file and click Build

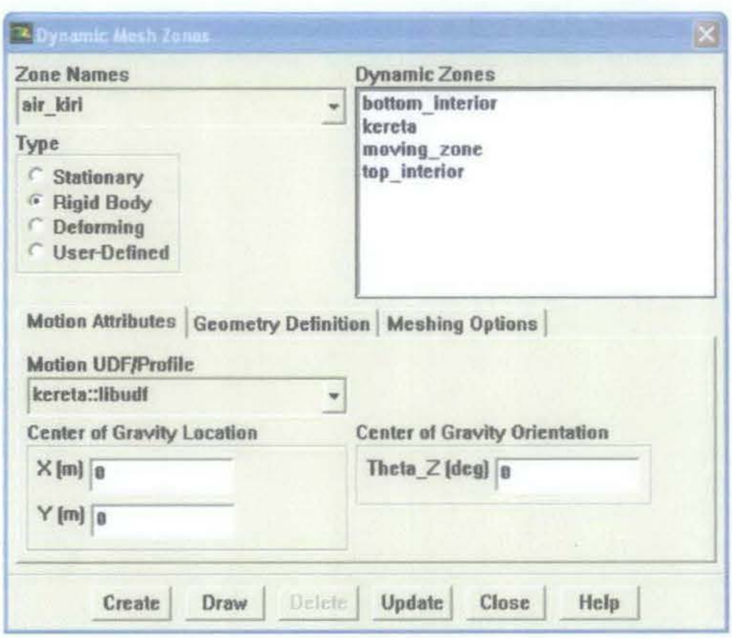
Enable dynamic mesh properties

Define□–→ Dynamic Mesh□–→Parameters...



Define dynamic mesh zones

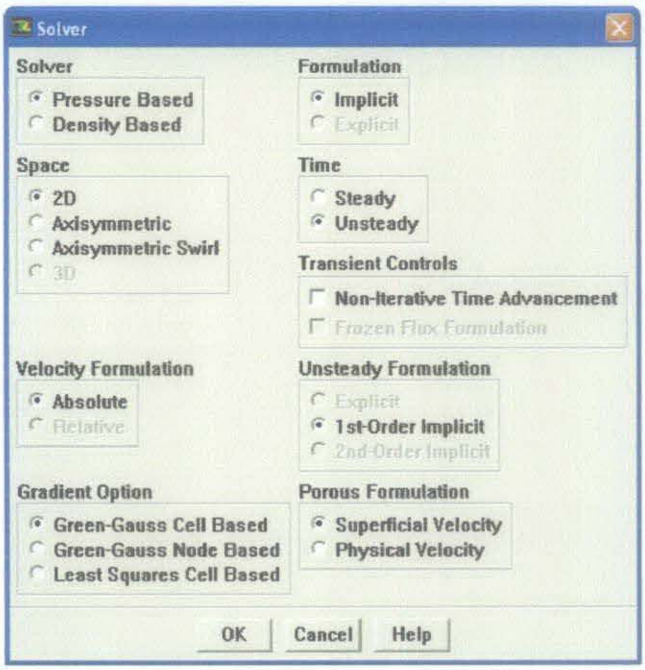
Define□–→ Dynamic Mesh□–→zones...



Repeat same steps as steady analysis, and change:

1. Set the solver settings.

Define□–→ Models□–→Solver...



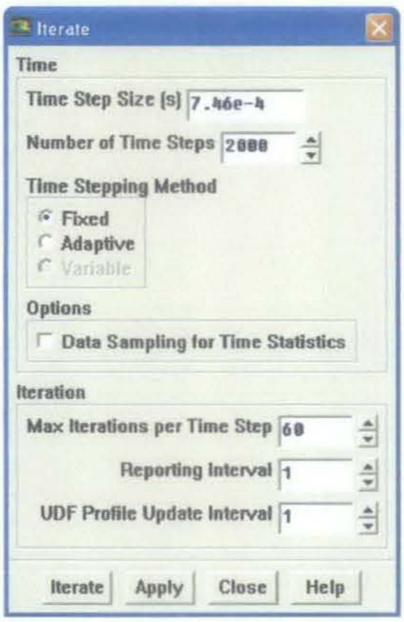
(a) Select unsteady solver.

(b) Click OK to retain the settings.

6. Start the calculation by requesting 60 iterations.

Solve —→Iterate...

No. of time steps to complete simulation at every 0.01m is 2000 time steps with step size (7.46 e-4 sec)



Results and Discussion

In this chapter the author will discuss the assumptions, the model design and the mesh design.

Assumptions

The assumptions made in this research are:

- No crosswind.
- The flow due to the 3D profile of the car does not affect the flow around the top view 2D profile.

Results

Steady Analysis

In this part of the report, the author will discuss the results obtained from the quasi static CFD simulation and clarifies how these results represent the real life although 3D and transient analyses were not investigated.

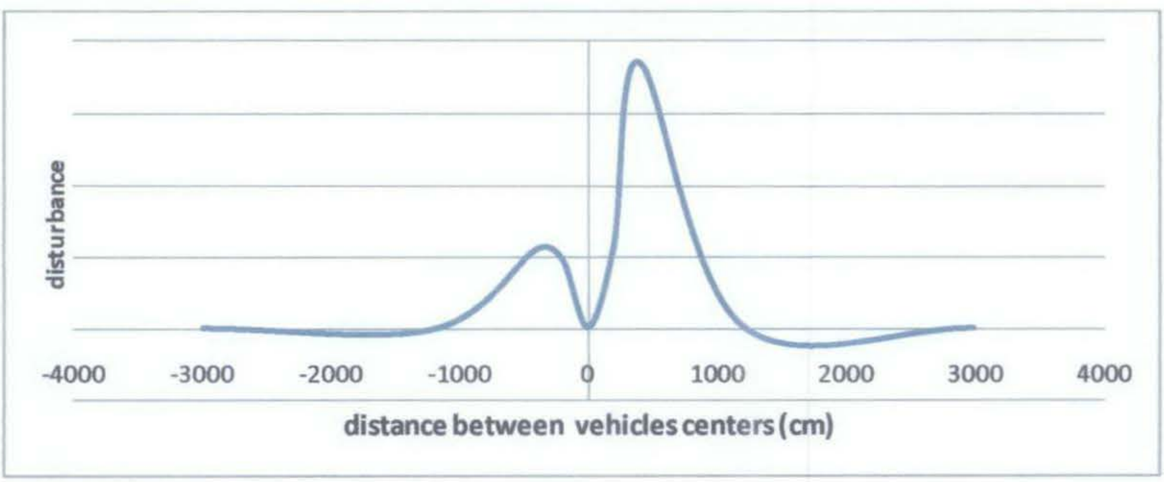


Figure 4: relation between the car disturbance and the vehicles positions

As one car approaches another car, trying to do an overtaking maneuver, the car faces some instability due to aerodynamic disturbances. As shown in figure 4, the disturbances starts to increase as the car get close to the front vehicle, and it starts to drop as soon as the overtaking vehicle (car 1) reaches the overtaken vehicle (car 2) till they reach the neck and neck position.

As car 1 tends to get in front of car 2, the disturbance rises rapidly until car2 2 is overtaken. Finally the disturbance decay naturally till it reaches the stable condition.

The disturbance magnitude is affected by the speed of the cars, the relative speed between the cars and the size ratio of the cars, so it is advised to hold the steering wheel with both hands when driving fast (race conditions) to be able to correct the instability and to avoid serious injuries.

In the following pages the contours of the static pressure and velocity magnitude is shown to provide the necessary visual aid to visualize the theoretical data which lead to the engineering analysis of the instability phenomena during vehicle overtaking.

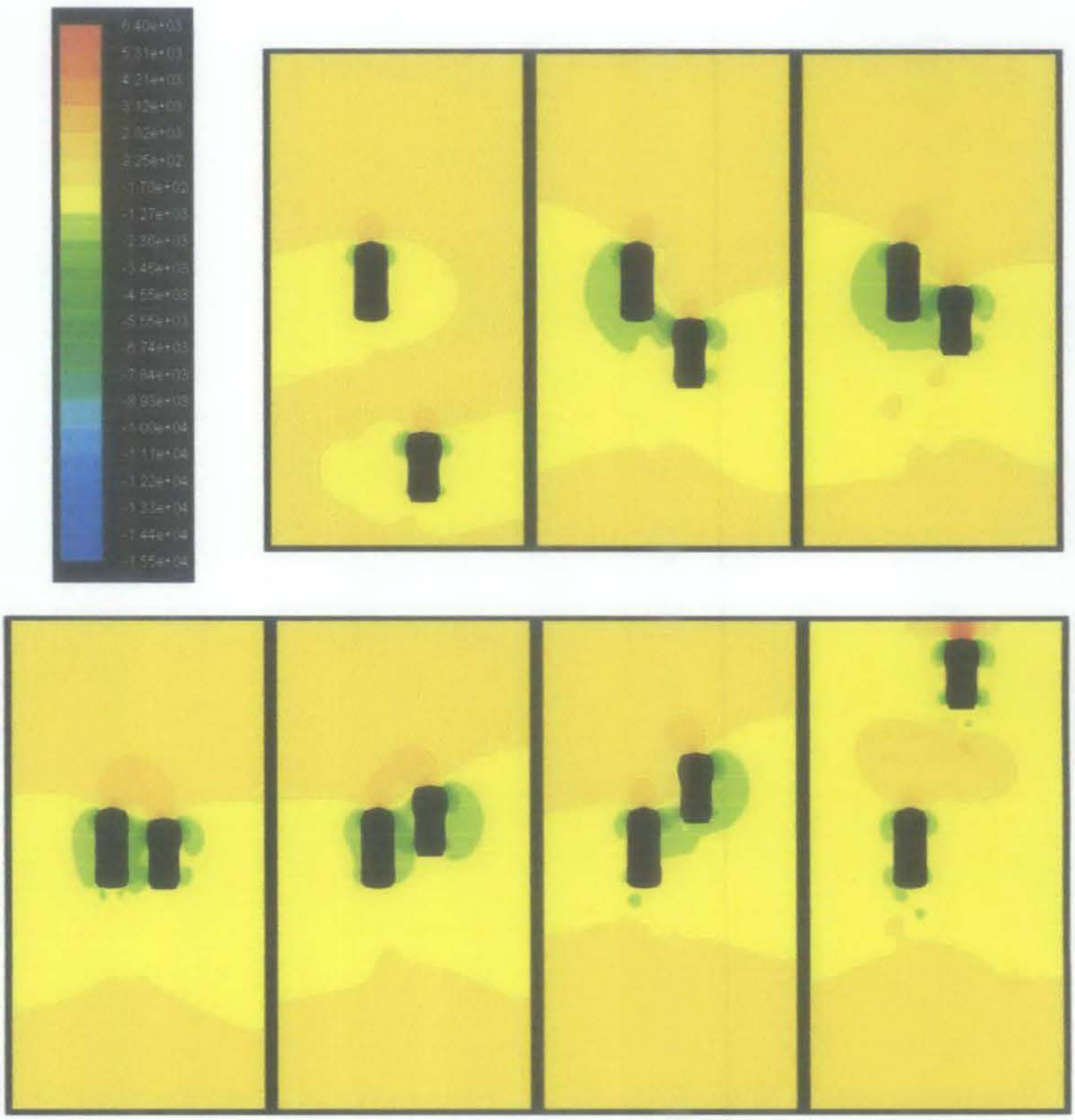


Figure 5: Pressure contours for steady analysis

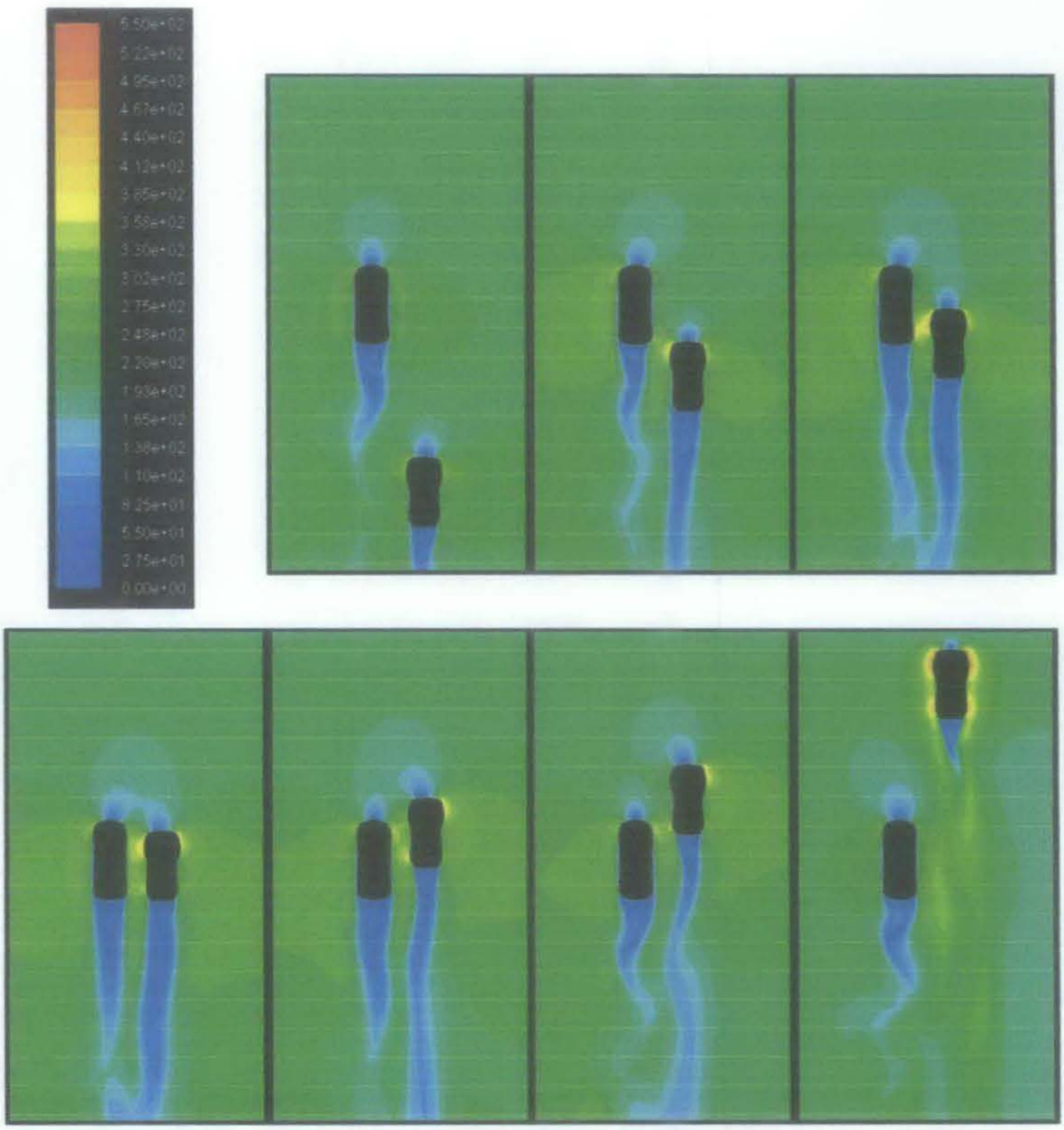


Figure 6: Velocity contours for steady analysis

Problems with unsteady (transient) analysis

As the research in transient analysis was being conducted, some problems were faced that stopped the continuance of the study in an unsteady state. The main problem was the existence of non positive volumes in the mesh, weather this problem is faced and bypassed, it always reappears. This reason is caused by high skewness of the elements due the mesh motion and deformation. To overcome this problem and run a dynamic mesh successfully it is advised to use Fluent Tgrid software which specializes in moving unstructured large meshes and it is able to fix the skewness problems

In fluent software:

Negative volumes problem appeared as the UDF started working, it was then corrected by smoothing the mesh, and the problems kept reappearing until it was not able to be fixed.

3D testing Results

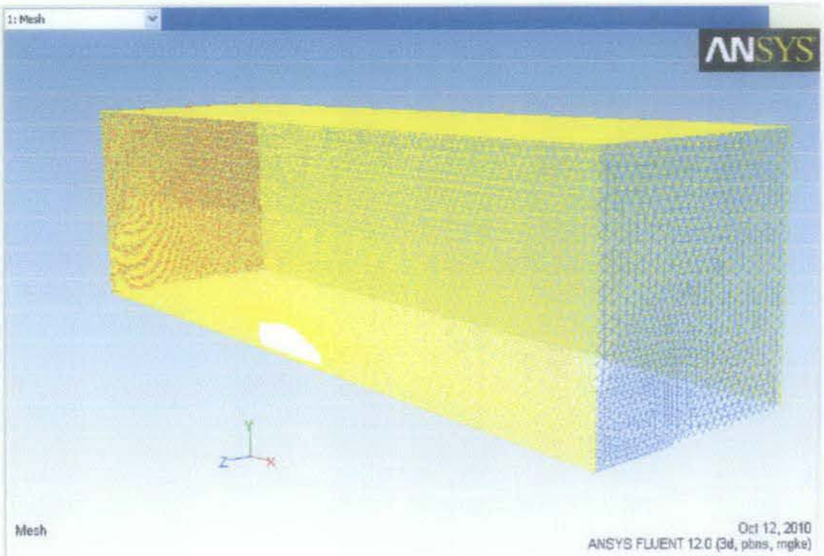


Figure 7: 3D mesh of lamborghini model, half car

Figure 8: residuals for 3D analysis

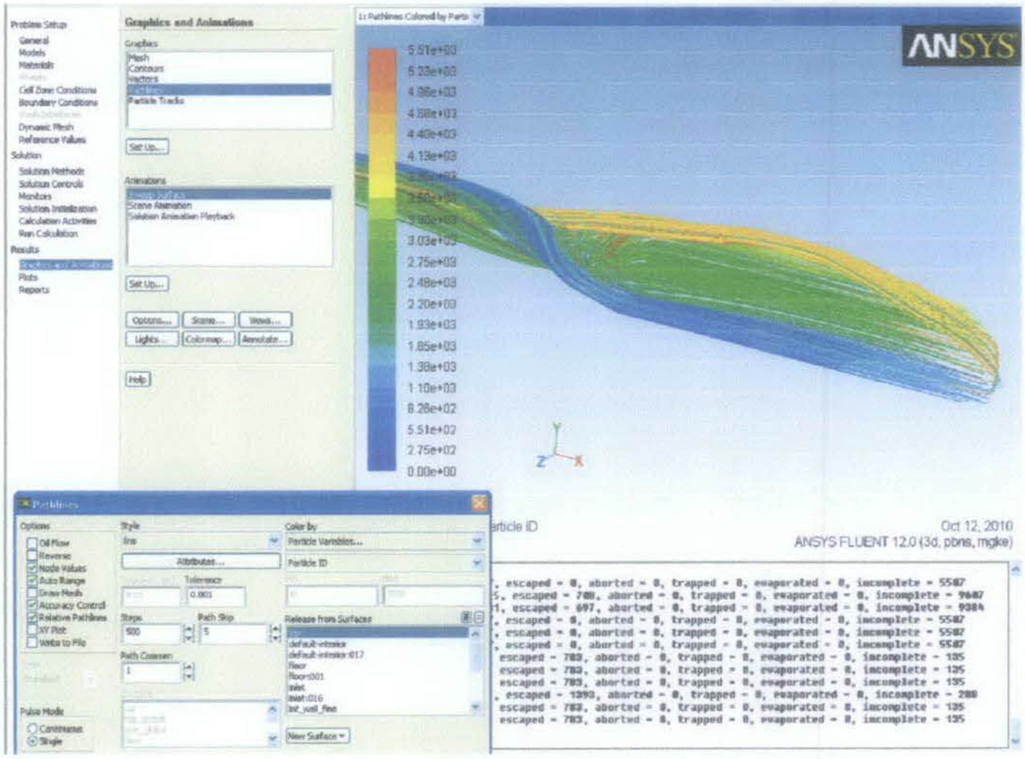


Figure 9: particles motion around 3D half model analysis

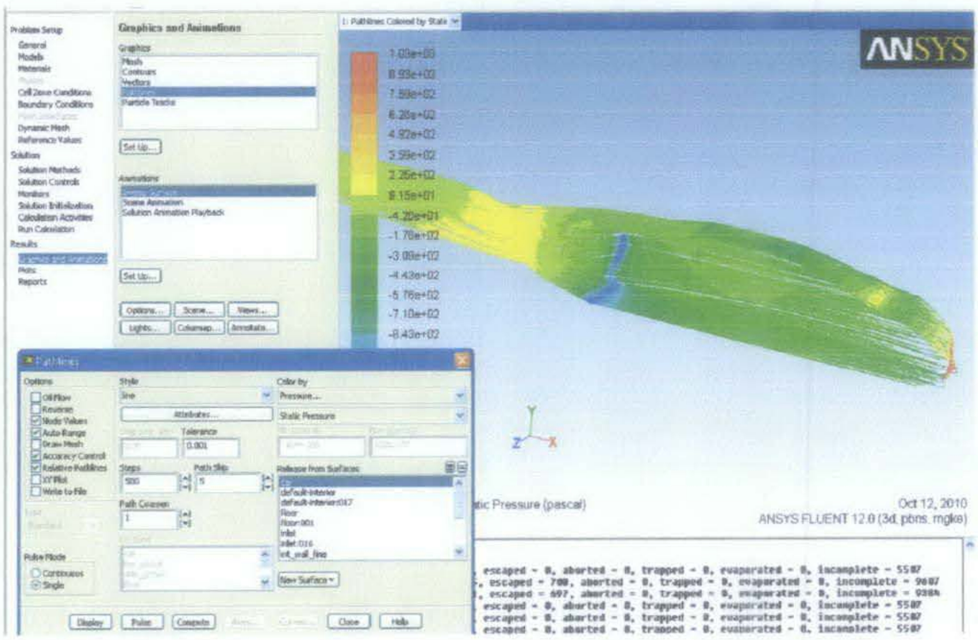


Figure 10: pressure pathlines for the 3D half model analysis

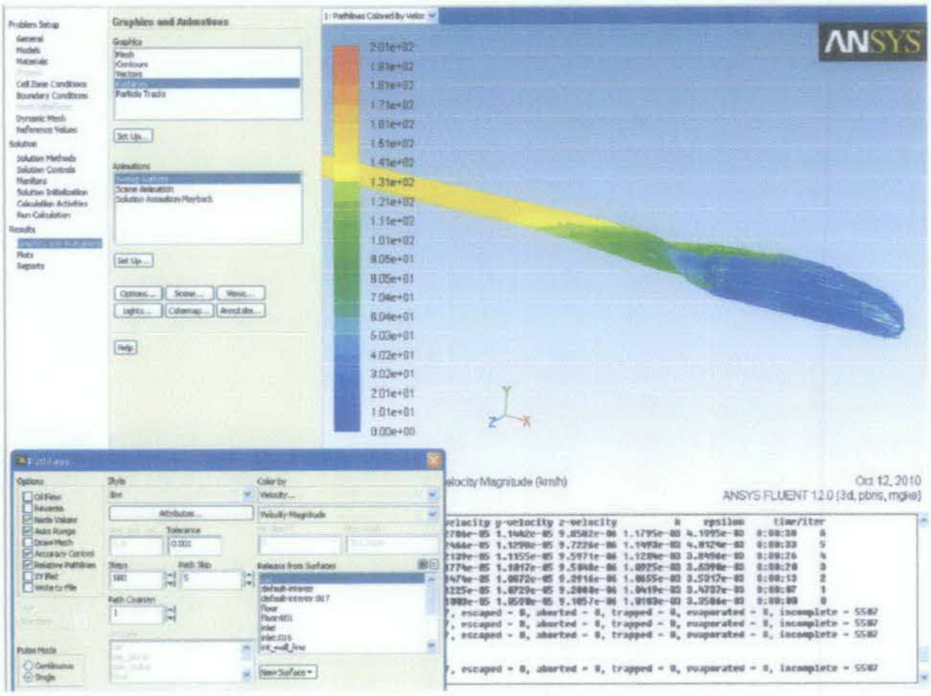


Figure 11: velocity pathlines for the 3D half model analysis

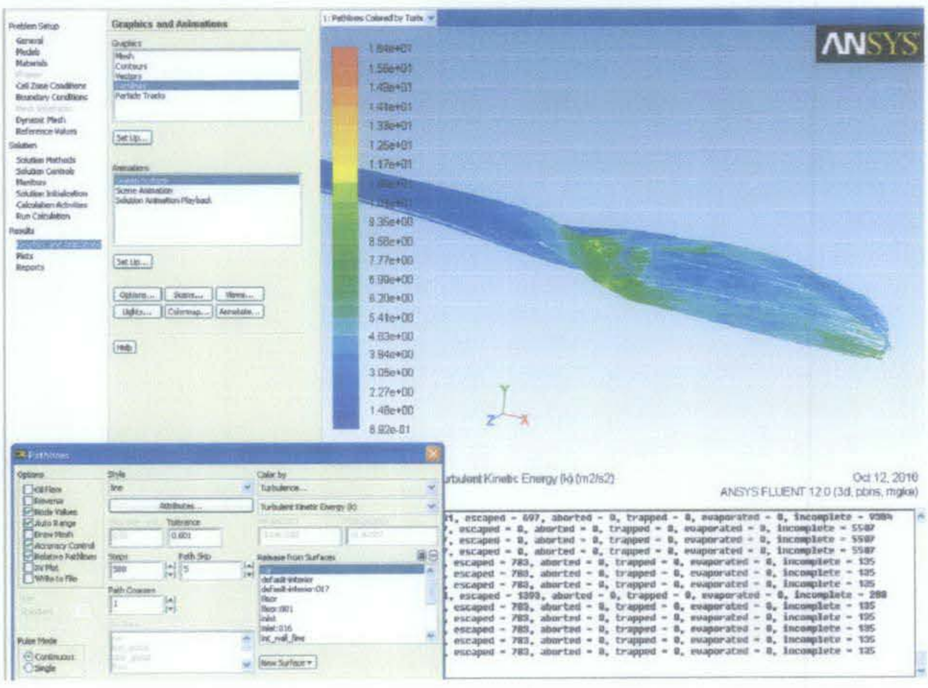


Figure 12: turbulance pathlines for the 3D half model analysis

The study was conducted on a half car model (Lamborghini Gallardo) at 150 Km/h. the grid consists of about 700,000 elements, so the results has computational errors as the number of the elements needs to be more than 4,000,000 elements to get a solution independent from the grid size. And to upgrade for this study we need about 15milion elements.

The pictures show the residuals, drag and lift convergence. It also shows the pathlines for the pressure, particles and velocity around the vehicle.

The grid was formed in gambit software and the analyses were conducted with fluent 6.3 and fluent 12 software.

This is a steady analysis with K-epsilon viscous model on a full scale 3D car

Drag coefficient $C_D = 0.7$,

Lift coefficient $C_L = 1.4$.

Conclusion and Recommendations

Conclusion

Since the topic chosen was confirmed, research has been done through journals and reference books in the internet and the library. Drag is one of the most important forces for affecting the vehicle movement as well as pressure forces.

Different mesh sizes were created and tested to obtain a grid independent solution. After selecting the suitable mesh size, several meshes were created with different positions between the vehicles. After the quasi static CFD analysis was conducted, the researcher tried to deepen the research field or put the basics for further investigations; a 3D half car model was generated to test the feasibility of 3D analysis, but the grid was too big for the computer and software resources.

Then transient analysis study was started, and done with good progress, but due to lack of time and resources, this study is pending for further investigations.

This study was a success in analyzing the flow between two vehicles, and in making the start for the 3D and transient studies.

Recommendations

For UTP:

- To install Fluent 12 and Microsoft Visual studio on the computers in building 17
- Install Tgrid

For the project:

- Upgrade to transient analysis

  As the quasi steady approach describes the flow and the disturbances due to vehicles positioning, the transient approach fine-tunes these results as it takes the relative velocities between the vehicles into consideration. Fluent Tgrid softwareis needed for this approach.

- Considering crosswind

  This study is conducted assuming normal wind conditions, but for more accuracy crosswind should be implemented in the study to mimic the real life situation.

- 3D flow analysis.

  The flow around the vehicles is affected by the 3D profile of the vehicle not by the 2D profile only; further investigations should be carried on 3D analysis. Minimum of 15 million element grid should be constructed to get a good mesh for grid independent results.

References

[1] Clancy, L.J.; Aerodynamics

[2] Batchelor, George (2000); An introduction to fluid dynamics.

[3] Yunus A. Cengel, John M. Cimbala; Fluid Mechanics.

[4] R.J. Corin , L. He, R.G. Dominy; A CFD investigation into the transient aerodynamic forces on overtaking road vehicle models.

[5] Chien-Hsiung Tsai , Lung-Ming Fu , Chang-Hsien Tai , Yen-Loung Huang , Jik-Chang Leong; Computational aero-acoustic analysis of a passenger car with a rear spoiler.

[6] Tayfun E. Tezduyar; CFD methods for three-dimensional computation of complex flow problems.

[7] Simon Watkins_, Gioacchino Vino; The effect of vehicle spacing on the aerodynamics of a representative car shape.

[8] W. Kieffer, S. Moujaes, N. Armbya; CFD study of section characteristics of Formula Mazda race car Wings.

[9] Fluent user guide, tutorial manual and UDF manual