CHAPTER 1

INTRODUCTION

1. INTRODUCTION

1.1 Background of the Study

When two vehicles are driven in close proximity, the wind changes can be felt around each other and under certain conditions they can generate severe force. This situation is due to the changes of aerodynamic flow around the vehicles. The aerodynamic changes are much more significance when the size and speed of vehicles increase. As one vehicle passes another on the road, flow fields around the two vehicles will generate transient aerodynamic forces. These forces can have an adverse effect on vehicle handling and stability. For this situation, Computational Fluid Dynamic (CFD) will build a computational model that represents a system or device to analyze and then apply real word physics and chemistry to the model. Beside that CFD will provide the images and data which predict the performance of the design.

1.2 Problem Statement

The forces like drag force and lift force can affect the other vehicle which can alter their road holding and thus result in safety problem. Small size of the vehicle will certainly feel more of the wake from larger heavier vehicles. To simulate the experiment about this project, software Gambit and Fluent will use. Gambit will be used to make geometry setup like a model of a car while Fluent will be used to run the simulation. This project used same dimension and same distance with the real experiment which were carried out previously at UTP. The result will show either same dimension will give same result or not.

1.3 Significant of Project

This project is mainly to investigate the reaction of the aerodynamic forces that react on vehicle in tailing each other. It is significant to the drivers to know the exact distance when they following the other vehicle. Studying the effect of these aerodynamic changes could help in minimizing the risk of an accident.

1.4 Objectives

- To make simulation using Computational Fluid Dynamic (CFD)

    - Model for two vehicles in proximity each other will be simulated using Gambit and Fluent software.

- To study the effect of velocity of vehicles on aerodynamic force.

    - The different speed of the model will show different effects on aerodynamic force.

    - Studying this effect will give more knowledge about the speed for the driver when proximity with another car.

- To study the effect of separating distance on aerodynamic force.

    - Different Drag and Lift forces will get from different distances between two models

1.5 Scope of the Study

- Study the effect velocity on aerodynamic force.

  - When a car is moving in different speed, the aerodynamic force that reacts on the car is different.

- Make a Computational Fluid Dynamic (CFD) simulation for two cars in proximity each other at different distances and analysis the force around the vehicles.

  - The effect of the different distance between two models will be analysis using CFD simulation.

- To validate the CFD model when simulating the experiment.

  - The result from CFD simulation will be compared with result from wind tunnel experiment.

1.6 Feasibility of Study

This project was compared the result between wind tunnel experimental which were carried out previously in UTP with simulation using CFD. Based on the research, the simulation should show the same result because simulation using CFD must use same dimensions and same steps with experimental.