CFD AND WIND TUNNEL TESTING FOR A REAR CAR SPOILER

AHMAD AZLAN BIN ASRI

(2007271166)

A thesis submitted in partial fulfillment of the requirements for the award of Bachelor of Engineering (Hons) Mechanical

Faculty of Mechanical Engineering

Universiti Teknologi MARA (UiTM)

MAY 2010

ACKNOWLEDGMENT

First of all, praise to Allah the Almighty, for His Most Compassionate and Benecificient, I am able to complete this project for two semester period. Thanks also to all persons who are willing to help me and guide me because these people had gave me inspiration in completing this report. It is my pleasure to acknowledge the assistance of the following individuals who guide me in the way of finishing this case study.

First and foremost, I would like to express my gratitude to my project advisor, Mr Baljit Singh a/l Bhathal Singh who has continuously guidance me and gave a great idea upon completing the FYP 1. Under him, I am well understood about the aerodynamics of the car such as air flow and also the relationship of fluid dynamics in the real application such as drag and lift.

Not to forget my beloved family who always supported me during my study from Diploma until Degree level in UiTM. Their advice and comment had improved me to be the great, strong and disciplined person in life. I also dedicate my thankfulness to my entire colleague for their contribution in helping and understand me when using STAR-CCM+.

Last but not least, I would like to express appreciation to any individual persons in lab or classes who are contribute in term of ideas either directly or indirectly upon completing my final year project. Your kindnesses are never I forgot and May Allah blesses all of you. Thank You.

ABSTRACT

In the way of designing a car, the aerodynamic must taken into consideration so that the car performance and stability will be satisfied and the fuel consumption can be reduced. To achieve this goal, one of the ways is the rear spoilers have to install at the rear end of car. The spoiler can create down force to allow the car grip on the road for better car stability. However, the position of the spoiler also have to chosen correctly to make the car aerodynamic acceptable. So, this study focused on the effect of rear spoiler on the sedan car in terms of drag coefficient and lowering fuel consumption. The method employed for this analysis by using Computational Fluid Dynamics (CFD) software and UiTM Low-Speed Wind Tunnel. The 1/17 scale of car prototype is drawing by using CATIA V5R18. Three variable data were tested such as velocity (v), height of spoiler (h), and angle of attack (α). The car then will simulated without using rear spoiler to compare the effect. Drag Coefficient (Cd) and Lift Coefficient (Cl) were determined during simulation and experimental progress. The result will be investigated by applying the Fluid Mechanics relations to understand the situation. From the results, the comparison will be made either to choose the car without rear spoiler or a car with rear spoiler. After that, the best rear spoiler which can give less drag and fine down force was chosen. For the expected result, the scale car without rear spoiler will give large drag compared the car with rear spoiler. Then, the lower height and negative angle of attack is will generate more drag. The data that give the less drag was considered as achieved the objective of this project which is to reduce fuel consumption.

TABLE OF CONTENTS