CHAPTER 2

LITERATURE REVIEW

First of all, understanding the basic theory of aerodynamic force is very important. Paper works, journal, engineering books or anything relevant to the project are reviewed to get more knowledge about the project.

2.1 Drag Force

One of the force that come when two car in proximity each others is drag force. When a car is moving, the force a flowing fluid exerts on a body in the flow direction is call drag force. It acts in the opposite direction of the movement of the body. Drag force must to be minimizing because it is undesirable effect like friction that cause some problem when a car is moving. Reduction of drag force in automotive can improved safety and durability of structures subjected to high wind and reduction of noise and vibration.

For this project, the drag force will be discussed when two cars are in proximity each other and this effect can be done using Computational Fluid Dynamic (CFD). Drag force will decrease and increase depends on the velocity and shape of the vehicle. Drag force increases with area. In the context of this model, area is the cross sectional area projected in the direction of motion. Different in area will give different value for the drag force [1].Drag force also increases with speed. An object that is stationary with respect to the fluid will certainly not experience any drag force [1].

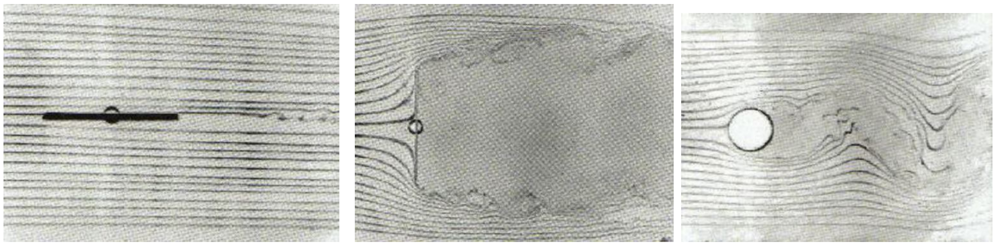

Figure 2.1: Flow of the Drag Force [1].

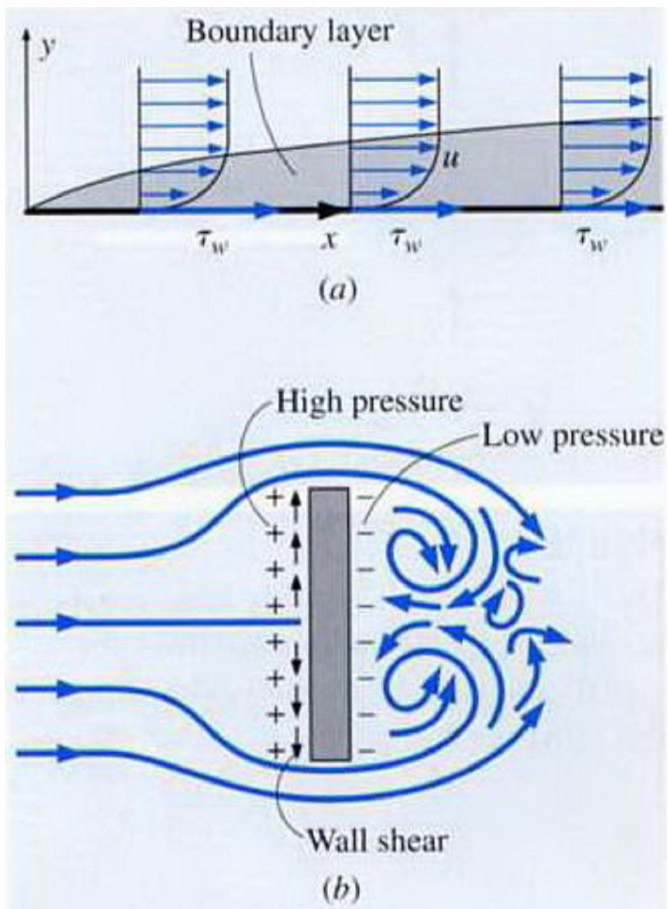

Figure 2.2: (a) The drag force acting on a flat plat parallel to the flow depends on the wall shear [1].

(b) The drag force acting on a flat plat normal to the flow depends on the pressure [1].

2.2 Drag Coefficient

The drag coefficient, in general, depends on the Reynolds number. At higher Reynolds numbers, the drag coefficients for most geometry remain essentially constant [1].This is due to the flow at high Reynolds numbers becoming fully turbulent.

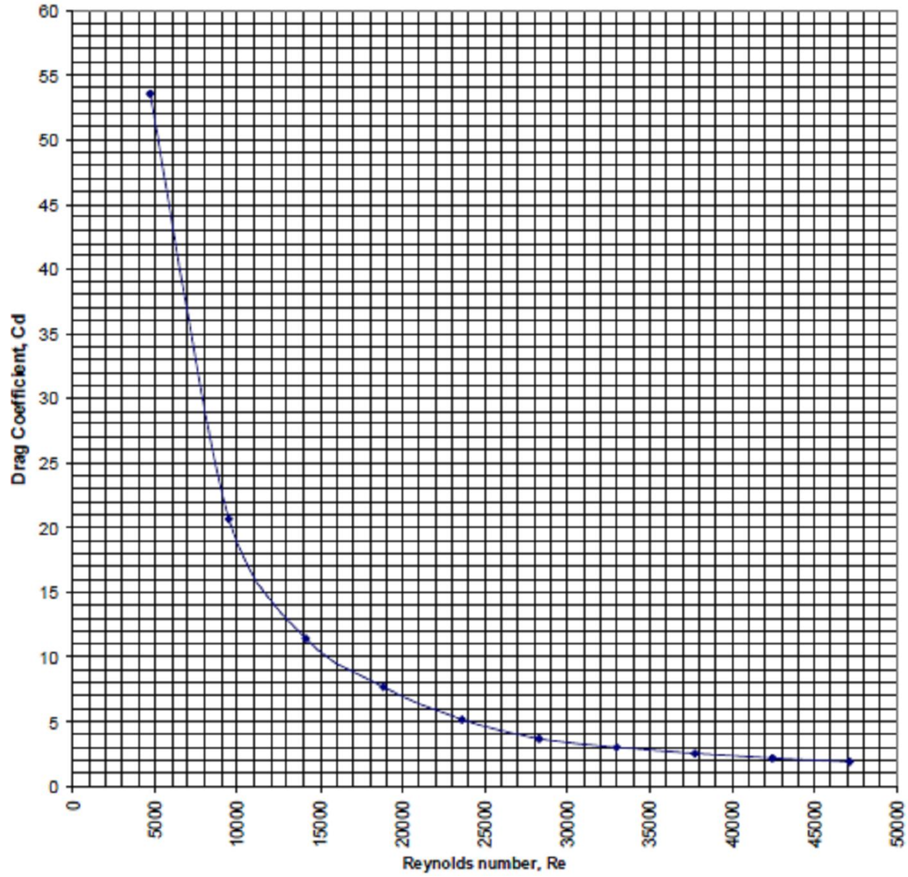

Figure 2.3: Typical graph of Drag coefficient versus Reynolds number [1].

Drag coefficient is inversely proportional to the Reynolds Number. The higher value of the Reynolds Number, drag force becomes smaller. The slope decrease until a certain constant C_D value [1].

Drag coefficient can get from the equation below.

$$C_D = \left[\frac{F_D}{0.5 \rho V^2 A_F} \right] \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (1)$$

C_D = drag coefficient

F_D = drag force

ρ = density of air

V = freestream velocity of the air

A_F = frontal area of the prototype/model

2.3 Lift Force

In the context of fluid flow relative to a body, the lift force is the component of the aerodynamic force that is perpendicular to the flow direction. It contrasts with the drag force which is parallel to the flow. Lift is generated in accordance with the fundamental principles of physics such as Newton's laws of motion, Bernoulli's principle, conservation of mass and the momentum. In automotive field, lift force are caused by difference in pressure acting on a body. Lift depends entirely on the nature of viscous flow past certain bodies in inviscid flow, there is no lift without imposing a net circulation. When there is no flow, there is no lift and the forces acting on the car are zero.

For the experiment, the lift will not be considered as the wind tunnel testing will not be accurate as the lift is associated with the ground effect. However, there will be no ground or road in the experiment. The models of the vehicle will be supported by a metal rod, connecting it to the balance that will measure the forces acting on the model. For this project, the effect of the lift force will discuss when simulating the model using Computer Fluid dynamic (CFD) [1].

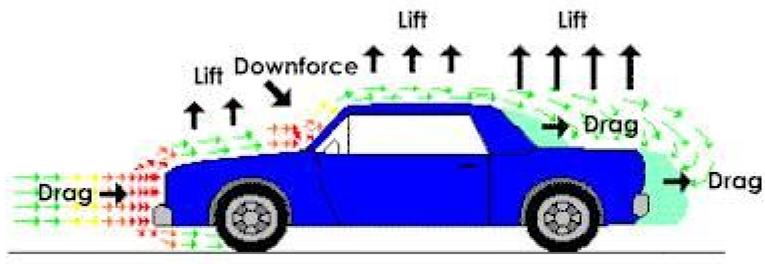

Figure 2.4: Lift force

2.4 Lift Coefficient

The lift coefficient is a number that aerodynamicists use to model all of the complex dependencies of shape, inclination, and some flow conditions on lift. Lift coefficient means the dimensionless quantity that describes the characteristic of the lift on the body. The lift coefficients vary along the surface as a result of the changes in the velocity boundary layer in the flow direction [1].The equation that used to get calculated lift coefficient is:

$$C_L = \left[\frac{F_L}{0.5 \rho V^2 A_P} \right] \qquad (2)$$

C_L = lift coefficient

F_L = lift force

ρ = density of the air

V = velocity of the air

A_P = top/platform area (parallel to the flow) of the object

2.4 Dimensional Similarity

In order to gain accurate accuracy result, the model that will build from computer must have similarities to the real world. These similarities are the concept of technique called Dimensional Similarity. Dimensional Similarity is introduced so that the test can be done on scale model rather than a prototype. There are three conditions that are needed to complete the similarity which are:

- Geometric similarity - model has the same shape with the size being scaled.
- Kinematics similarity – velocity at any point in the model flow must be proportional to the velocity at the corresponding point in the prototype flow.
- Dynamic similarity – all forces in the model flow are scaled by a constant factor to the corresponding force in the prototype flow.

Size the model that will use in the computer must be same with the model of car that use in wind tunnel. In wind tunnel, to determine the appropriate size, some calculation of dimensional analysis needs to be done [1].

$$\left[\frac{\rho VL}{\mu}\right]_{prototype} = \left[\frac{\rho VL}{\mu}\right]_{\mathrm{mod}\,el} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots \text{(3)}$$

Where:

ρ = density of air

V = freestream velocity of the air

μ = kinematic viscosity

L = Length of the model

2.5 Meshing

Meshing is an integral part of the CAE analysis process. The mesh influences the accuracy, convergence and speed of the solution. More importantly, the time it takes to create a mesh model is often a significant portion of the time it takes to get results from a CAE solution. Therefore, the better and more automated the meshing tools, the better the solution. From automatic meshing to highly crafted mesh, ANSYS, Inc. provides the ultimate meshing solution. ANSYS provides powerful pre- and post-processing tools for mesh generation from any geometry source, to produce almost any element type, for nearly any physics, for virtually any application [5].

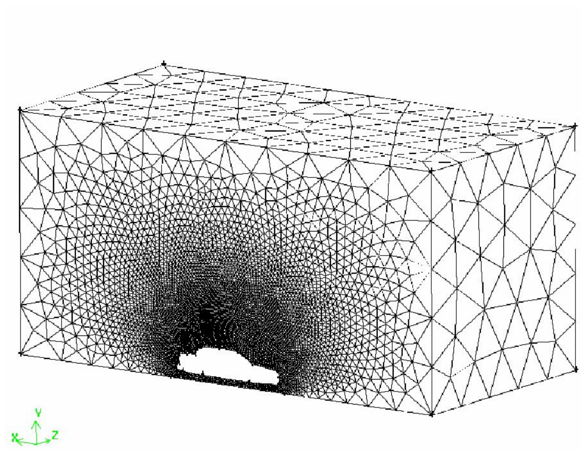

Figure 2.5: Typical portion of the volume mesh [5].

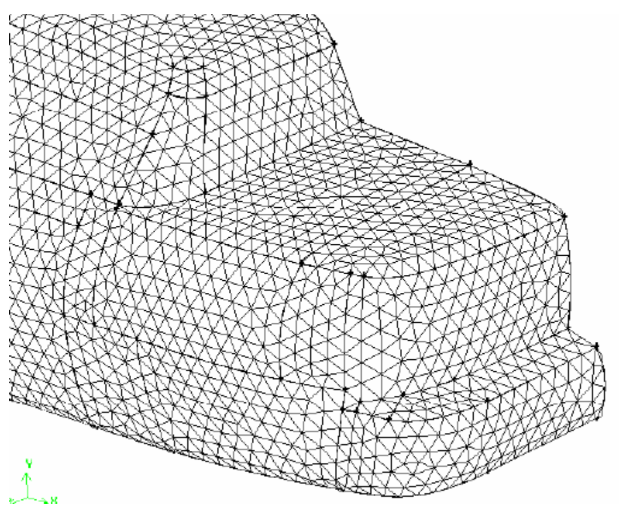

Figure 2.6: Typical surface mesh on rear of car body