

CHAPTER 3

METHODOLOGY

For completing this project, there are several methodologies need to be done. The project will start by literature review, analysis of Wind Tunnel Experiment, CFD simulation, calculation for drag and lift forces, and comparing computational result with experimental result. Softwares that used for this project were Fluent, Gambit, Microsoft Word and Microsoft Excel. Figure 3.1 below illustrated the flow chart of the project.

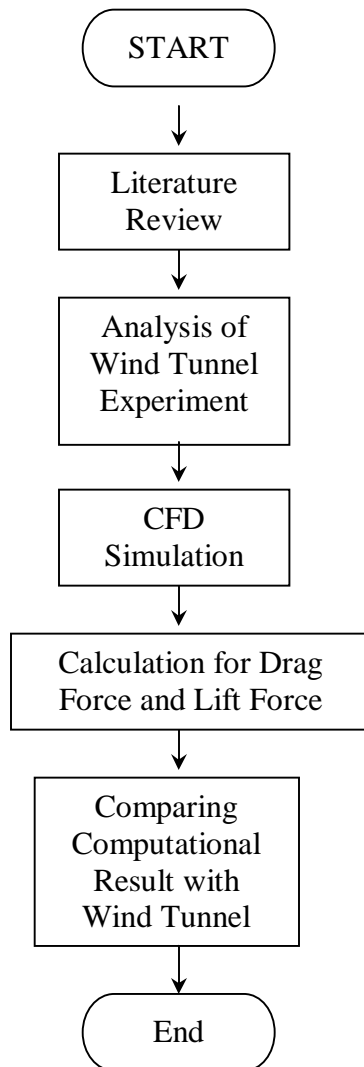


Figure 3.1: Flow chart of the project

3.1 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.

3.2 Analysis Wind Tunnel Experiment

In order to make CFD simulation for this project, the wind tunnel experiment was reviewed. The purpose of this analysis is to know the value of the velocity, distances between two cars and the dimension of the car that used in experiment. The CFD simulation must use same value with experimental because at the end of this project, both result will be compared.

3.3 Computational Fluid Dynamic (CFD) Simulation

To investigate the force that react when two cars in proximity to each other, some simulation using computer were done. At this stage, there were several steps need to be done before run the simulation.

- a) Building the model - Building the model within Gambit software using same dimension with wind tunnel model.
- b) Meshing process - The model will separate into small pieces and it will show the force that react on the model. Beside that, the mesh will be examined to check the quality of resulting mesh.
- c) Boundary Condition - There are several options at the boundaries through which fluid enters the computational domain (inflow) or leaves the domain (outflow). At a velocity inlet, the velocity that occurs is the velocity of the incoming flow along the inlet face. Beside velocity inlet, there are other boundaries which are pressure outlet and wall.

- d) Decide separating distances – The separating distances that used in simulation were 6.25 cm, 12.5 cm, 18.75 cm, and 25 cm.
- e) Decide velocity value – the speed of the air in simulation for all cases were same with speed in experiment which are 5m/s,10m/s,15m/s,20m/s,25m/s, 30m/s,35m/s,40m/s,45m/s, and 50 m/s.

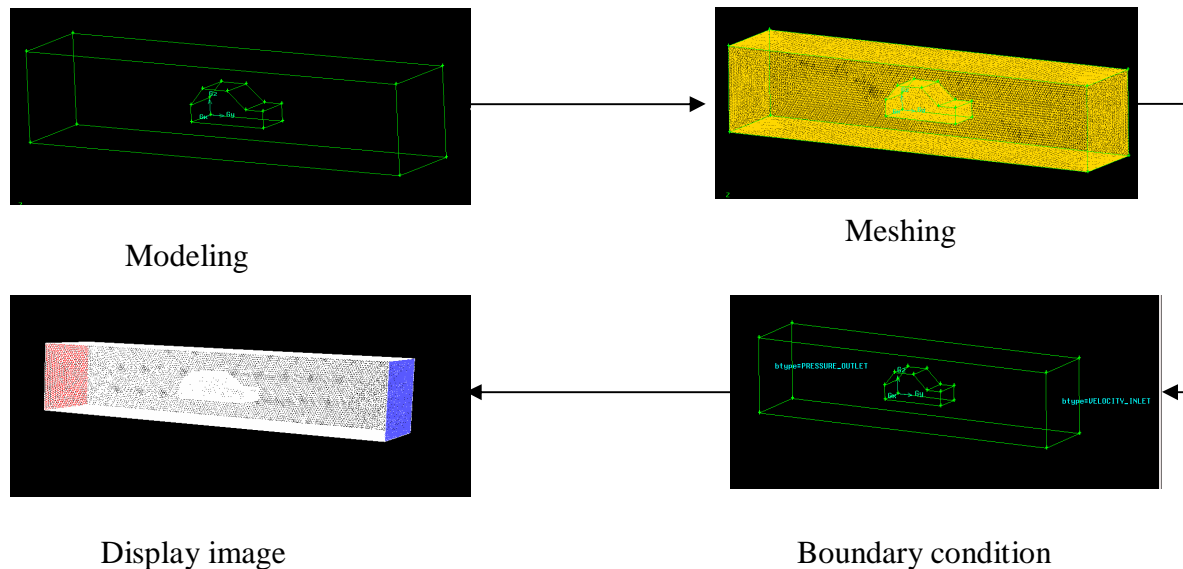


Figure 3.2: CFD procedure

3.4 Calculation for Drag and Lift Force

Drag force and lift force were calculated after known the value of drag coefficient and lift coefficient. From this value also, the Reynolds number will be calculated

3.5 Comparing CFD Result with Wind Tunnel Result

The last method that will use is comparing the result from computer simulation with wind tunnel experiment. Result from the simulation using computer should give more accurate result comparing with wind tunnel result. If the result in the simulation is same, this project will continue with the different value of the distances and various size of the car.