ANALYSIS OF CAR BODY AERODYNAMICS WITH VARIATION OF SPEED USING CFD SOFTWARE

Bagus Gde Didit Citra Budgeta
Udayana University
Korespondensi: diditca7@gmail.com

Abstract

The car is one of the most common vehicles used to meet transportation needs around the world. In the development of technology in cars, one of the influencing factors is the body design of the car. When designing a car, there are several aspects that need to be considered, one of which is the drag coefficient. In this study, a car body shape will be designed for further simulation and analysis with the help of the ansys 2020 software. Comutational fluid dynamics can analyze or predict the fluid flow in the car body. The formation process includes Preprocessing, Solving, and Postprocessing. The analysis was carried out on three-dimensional (3D), steady, turbulent flow. The variable is used to analyze the speed of the wind that will be passed by the car body, with a range of 15m/s, 30m/s, and 45m/s.

Keywords: Aerodynamic, CFD, Ansys, Car

INTRODUCTION

Vehicle is a means of transportation that is driven by system equipment that is on the vehicle. Vehicles have several types, such as light vehicles, heavy vehicles, to vehicles used for sports or racing. The vehicle consists of various components in it, some of the main components consist of the drive or engine, the frame or vehicle frame, and the vehicle body.

The automotive world has experienced rapid development, both in terms of developments in the machines used which include engine performance, fuel consumption and emissions, developments in the applied technological features, as well as in terms of forms that have progressed for the sake of vehicle aesthetics and aerodynamics.

Over time, automotive manufacturers are competing to make vehicles with the most aerodynamic body shape possible to get a low drag force and coefficient of drag (Cd) in order to improve performance and also fuel consumption, but also adjusted to the function of the car being produced.

The shape of the body that is engineered in such a way will produce different fluid flow characteristics and greatly affect the function of the body shape.

METHOD

Aerodynamic Drag

Objects that move through the fluid will experience interactions on the surface of the object with the fluid flowing or passing through it. These interactions are forces and moments originating from the shear stresses caused by viscous effects, and the normal stresses caused by the pressure distribution.

Drag on an object can be broken down into two parts, namely drag caused by friction and drag caused by pressure.

Drag can be written in Coefficient of Drag (Cd), which is the comparison between drag and the Dynamic Pressure Freestream which can be written as follows:

$$Cd = \frac{D}{1/2\rho u^2 A} \quad (1)$$

CD =Coefficient of drag
D = Drag force
ρ = Density of air
u = air speed
A =*Reference Area*

Computational Fluid Dynamics

Computational Fluid Dynamics or commonly called CFD is a way to analyze a system involving fluid flow, heat transfer, chemical reactions and other physical phenomena based on computer-based simulation. [4]

At present CFD has become one of the approaches taken in seeking answers to an engineering problem, especially in the field of fluid mechanics and heat transfer. An example of one CFD software is Ansys 2020

Research Stages

In this study, there are three main stages of the method that must be carried out, namely: preprocessing, solving or processing, and postprocessing. Due to the limitations of visualizing the flow across the car body, this study uses a numerical method with the help of Ansys Workbench 19.2 software using a CFD-solver, so that it can help make it easier to carry out a gradual process because it is already in sequential form and if there is an unresolved process, the next process cannot be completed. next. This study also includes a 3D appearance of the car body.

a) Preprocessing

Preprocessing is the first step in building and analyzing a computational model (CFD). This stage includes several sub-stages including: creating geometry, determining domains, creating meshing and determining the parameters used.

Parameter	Dimensions
C (Chords)	4017.96mm
W (Widht)	1850mm
H (Height)	1408.2mm

Table 1. Car dimensions

Figure 1. 3-dimensional car design

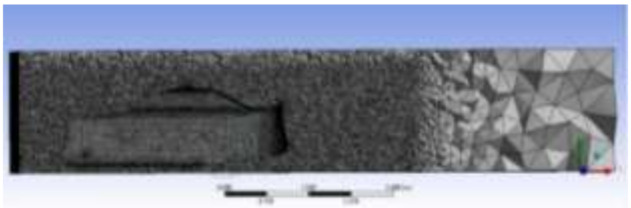

Figure 2. Meshing

b) Processing or Solving

With the help of the CFD-Solver Manager software, the conditions that have been set during preprocessing in the CFD-Pre software will be calculated (iterated). If the convergence criterion is reached with the convergence criteria 10-6, then the stage is continued in post-processing and if it is not reached, the stage will move backwards to the stage of making meshing or by reducing the convergence criteria.

c) Post Processing

Post-processing in the CFD-Post software is the display of the results and analysis of the results obtained in the form of qualitative data and quantitative data. Quantitative data in the form of pressure coefficient distribution, drag force and lift force. While qualitative data is in the form of flow visualization by displaying pathlines in the form of velocity magnitude.

DISCUSSION RESULT

Residuals and Iterations

In this study, the simulation is targeted to achieve an error value of 10-4 on the residual and an error value close to 0 on the simulated mass flow rate. As can be seen in Figure 1 and Figure 2, convergent conditions have been reached.

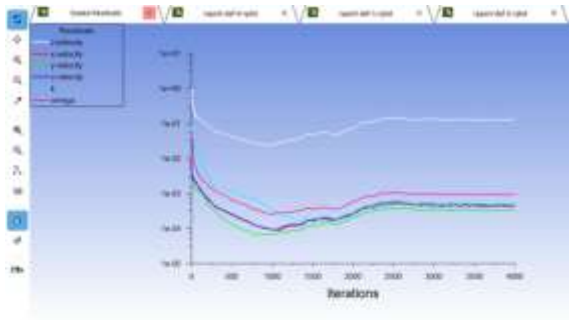

Figure 3. Residuals and iterations

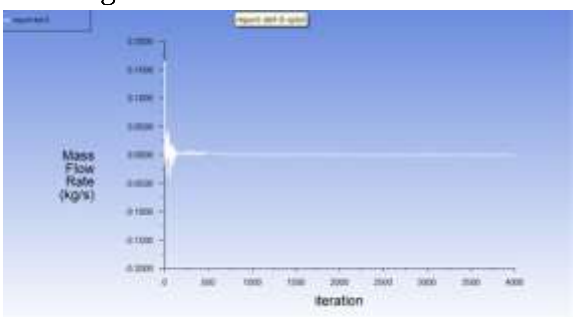

Figure 4. Mass flow rates

Streamlines

Streamline Velocity obtained after post processing simulation is shown in Figure 3. In Figure 3, the redder the color of the streamlined line is, the higher the wind speed. It can be seen from the front bumper that the streamlined car body failed to split properly, this is marked by a high drop in speed and a streamlined shape that turns steeply.

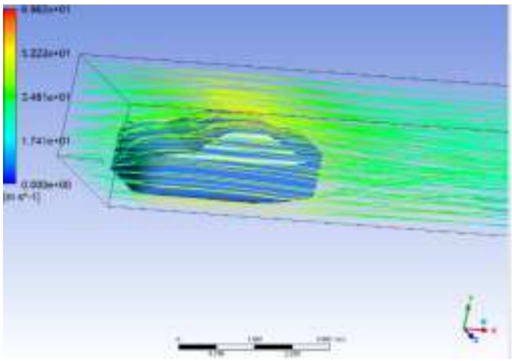

Figure 5. Velocity streamline on the car body

Contours on the XY Plane

A plane is made on the XY plane to obtain the pressure distribution and also the velocity in the horizontal direction around the car body. The pressure contour shows a high distribution on the front of the car and also on the lower back, this can affect the high value of the drag force and the coefficient of drag (CD) on the design.

The speed contour shows the speed value in the horizontal direction around the car. It can be seen that at the front of the car there is a significant decrease in speed, indicating that there is resistance to wind flow by this design, besides that there is also a considerable wake at the rear of the car.

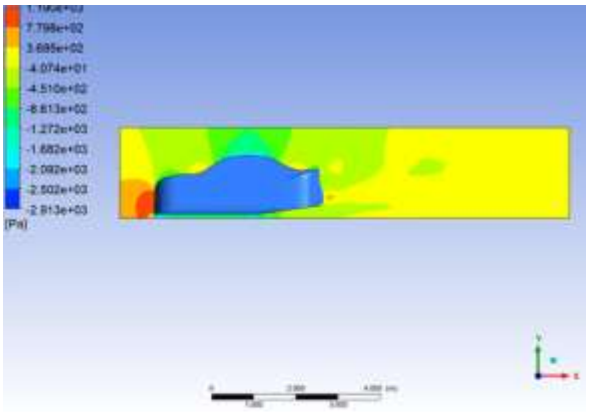

Figure 6. Pressure contour on the XY plane

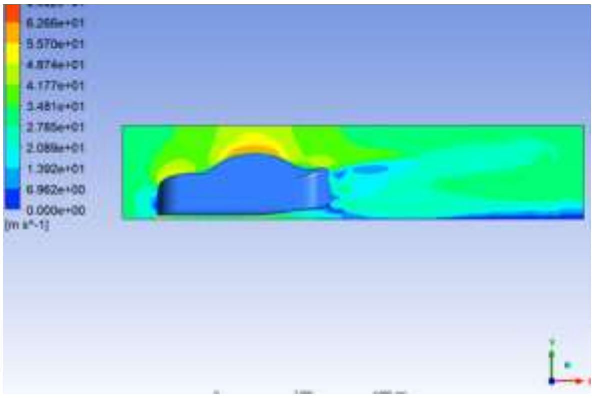

Figure 7. Contour velocity on the XY plane

Vortex Core Region

The vortex core region can show flow phenomena that occur immediately after contact with the body. shown in the results of the flow shape after contact with the car there is a wake on the rear and also high pressure and low speed of almost 0 on the front.

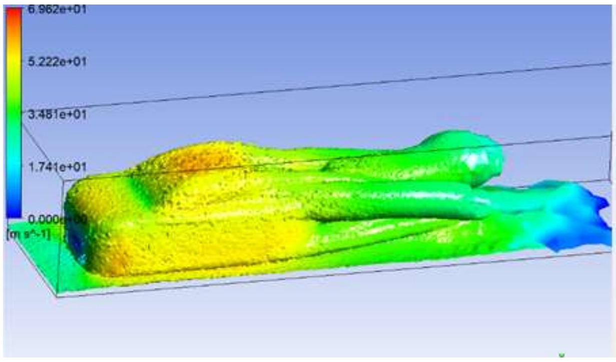

Figure 8. Vortex core region with velocity contour

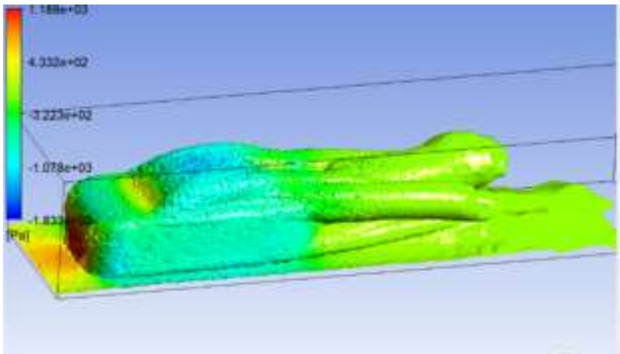

Figure 9. Vortex Core Region with Pressure Contour

Drag and Drag Coefficient

In this study, a report definition was designed to monitor drag and coefficient of drag values at each iteration. after the convergent simulation the drag values can be taken, the drag values obtained at each wind speed are as shown in the following graph.

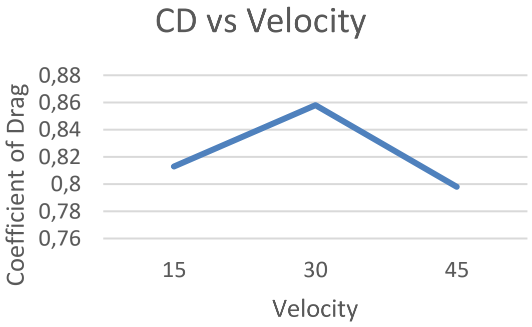

Figure 10. Comparison between CDs at each airspeed

In the graph it can be seen that the highest CD value experienced by the design during the simulation with a speed of 30m/s. The results of 3 simulations at speeds of 15 m/s, 30 m/s, 45 m/s each experienced a CD of 0.813; 0.858; 0.798.

CONCLUSION

Analysis of fluid flow on the car body shows the occurrence of high air resistance at the front of the car. There was a big enough wake on the back of the car. the highest pressure value on the car body is the front bumper. the highest drag value occurs in the design when given a flow of 30 m/s with a coefficient of drag value of 0.858

REFERENCES

Gusti Muttaqin, I. ., Sucipta, M. ., & Suarda, M. . (2022). Simulasi Computational Fluid Dynamic Pada Model Turbin Vortex Variasi Kecepatan Rotasi Runner. Sibatik Journal: Jurnal Ilmiah Bidang Sosial, Ekonomi, Budaya, Teknologi, Dan Pendidikan, 1(8), 1445–1454. https://doi.org/10.54443/sibatik.v1i8.188

Prihadnyana, Y. (2017). Analisis Aerodinamika Pada Permukaan Bodi Kendaraan Mobil Listrik Gaksi (Ganesha Sakti) Dengan Perangkat Lunak Ansys 14.5. JJPTM, Vol. 8, No. 2.

Yogatama, M., & Trisno R (2018). Studi Koefisien Drag Aerodinamika Pada Model Ahmed Body Terbalik Berbasis Metode Numerik. JTM, Vol. 7, No. 1.

Stople, Remi André. 2011." Testing Efficiency And Characteristics Of A Kaplan- Type Small Turbine". Trondheim. NTNU. Hal 5-7.

H. K. Versteeg dan W Malalasekera, 2007. "An Introduction to Computational Fluid Dynamics", Vol. II.

Winda, M., Herny Susanti, P., & Ayu Putri Trarintya, M. (2022). The Role of Commitment to Mediate Effect of Motivation on The Performance of Waste Bank Managers in The City of Denpasar. International Journal of Social Science, Education, Communication and Economics (Sinomics Journal), 1(2), 115–130. https://doi.org/10.54443/sj.v1i2.12