

SIMULATION OF FLUID FLOW THROUGH SEDAN CAR YRS 4 DOORS WITH SPEED VARIATION USING CFD

Syamsuri

Lecturer
Institut Teknologi Adhi Tama Surabaya
Mechanical Engineering
syamsuri@itats.ac.id

Zain Lillahulhaq

Lecturer
Institut Teknologi Adhi Tama Surabaya
Mechanical Engineering

M Yusron

Student
Institut Teknologi Adhi Tama Surabaya
Mechanical Engineering

Aerodynamic forces that occur around the vehicle must be considered since it involves safety, ergonomic, and fuel consumption. To reduce fuel consumption, the vehicle should be built as aerodynamic as possible to minimize drag forces. The vehicle becomes unstable at high speed due to increasing lift force. To balance the vehicle at high speed, a downforce should be generated to keep the tires attached to the road surface. Each type of car has a various value of aerodynamic force due to its design, dimension, and cross-section area. The characteristics of streamflow around the car are discussed in this paper. This research simulated 2D sedan car YRS 4 Doors in the steady condition in various velocities, i.e. 23 m/s, 26 m/s, and 40 m/s. This simulation used the Quad Pave mesh model and run in $k-\epsilon$ implicit turbulence model. The characteristics could be observed from the qualitative and quantitative data. The quantitative data used as measurable data were Coefficient of Pressure (C_p) and Drag Coefficient (C_D). Quantitative data was shown to outline a better visual explanation of the streaming characteristic. The qualitative data used in this paper are path lines, velocity vectors, and contours. The high-velocity stream results in a low value of C_p . When the fluid flowed at high speed through a surface, it had low pressure. The coefficient of drag in the high-speed car decreased as the free stream increased. The value of the coefficient of drag (C_d) from this research was app. 0.567.

Keywords: Sedan Car, Drag Coefficient, Quad Pave Mesh Model, CFD

1. INTRODUCTION

Fluid which flows around the car is causing drag and lift forces on the vehicle. The lift force has an acceleration effect on the vehicle. This is caused by the lift force on the vehicle reducing the friction which occurs between the vehicle wheels and road. Meanwhile, the drag force causes velocity reduction effect in the vehicle. The magnitude of lift and drag force on the vehicle are determined by several factors, including: flow velocity, cross-sectional area, shape and weight of the car. Drag and lift forces are considered as aerodynamic force. Aerodynamic force that occurs around the vehicle must be considered as well, because it involves safety, ergonomic and fuel consumption. To reduce fuel consumption, the vehicle should be built as aerodynamic as possible to reduce drag [1]. The vehicle becomes unstable at high speed according to the increment of lift force. In order to balance the vehicle in high speed, down force should be generate to keep the tires attach to the road surface [2].

Fluid which flows around the car, pass the upward area of the vehicle and separate to flow along the upper and underneath body. The pressure difference of underneath the car and upper body of the car will cause drag and lift force. The stream from underneath and upper of the vehicle body are flow through to the tail area and create secondary flow [3]. In lower velocity flow, the secondary flow rolls behind the vehicle and create two anti-clockwise vortexes. In higher velocity flow, the vortex behind the vehicle swept by stream which flow under the vehicle and the upper stream flow direct to the ground [4].

Accessories such as spoilers, vortex generators or change tire models can add to improve vehicle stability [5]. Spoiler is able to increases tires capability to produce cornering force, stabilizes vehicles at high

speed, Improves braking performance and gives better traction [6]. The vehicle with spoiler resulted higher coefficient of lift along with increment speed of the car. Vehicle with a rear spoiler establish the comfortable and safer condition for driving [7]. Vortex generators can be applied to control separation and adverse pressure gradients [7]. Vortex generators (VGs) which install on the upstream in separation point are able control separation of airflow and improve the aerodynamic characteristics. Vortex generators improve the aerodynamics of vehicle by reducing the drag and lift force [9].

Computational Fluid Dynamics (CFD) and experimental wind tunnel are able to test aerodynamic characteristic of the vehicle. The experimental test in wind tunnel, need huge investment to produce the prototypes. Moreover the accuracy of the wind tunnel measurements is affected by several factors, including blockage, scaling effects, moving road problem, reliability and validity of the data in each case [10]. Numerical simulation research has detailed results qualitatively and quantitatively. The results obtained from the simulation should be then validated first with the results of experiments and other research simulations. CFD can also be simulated various models of turbulence, time step and multiphase flow [11].

Vehicle has complex shape to model in numerical computation. Simplified vehicle shape such as Ahmed body is built to and generates better understanding of such streamline flows [12]. The accuracy of the results of numerical simulation depends on the turbulence model [13]. The k-ε standard and SST model result some difference compared with the experiment, especially in flow separates and reattaches area. DES simulations based on a modified version of the SST-DES formulation, resulted in a significant improvement of the solution compared to the SS-TRANS model especially in separation zone and strong trailing vortices [14].

Each type of car resulted different value of aerodynamic force due to the design, dimension and cross section area. The characteristics of stream flows around the sedan car YRS 4 Doors are discussed in this paper. The characteristics can be observed from the qualitative and quantitative data. The quantitative which used as measurable data are Coefficient of pressure (CP) and Coefficient of pressure (CD). Quantitative data is shown to get better visual explanation of the stream characteristic. The qualitative data used in this work are path line, velocity vectors and contours

2. EQUATIONS

Simulation data results are represented in non-dimensional graphics to show the aerodynamic characteristic of the car. The non-dimensional numbers that represent the aerodynamic characteristic is Coefficient of pressure (CP), and Coefficient of Drag (CD), as shown in equation 1 and 2.

$$C_P = \frac{P_C - P_\infty}{\frac{1}{2} \rho U^2} \quad (1)$$

$$C_D = \frac{F_D}{\frac{1}{2} \cdot \rho \cdot A_S \cdot U^2} \quad (2)$$

with,

- C_P : Pressure coefficient
- P_C : Static pressure in a specific point
- P_∞ : Static pressure free stream
- ρ : Density [kg/m³]
- U : Freestream velocity [m/s]
- C_D : Torque drag
- A_S : Frontal area [m²]

3. METHODS

This research simulated 2D sedan car YRS 4 Doors in steady condition, shown in Figure 1. The domain meshing also shown in Figure 2. The car simulated in various numbers of velocities in 23 m/s, 26 m/s, and 40 m/s. The design of the car is drawn in commercial drawing program. Dimension of 2D sedan car YRS 4 Doors in this paper refer to previous experimental study [4]. The dimensions of the car and test section are shown in Table 1.

Table 1: Dimension of car and test section.

AREA	DIMENSION	VALUE
Car	Length (L)	4.3 m
	Width (W)	1.69 m
	Height (H)	1.46 m
Test Section	L1	8.6 m
	L2	21.5 m
	H1	5 m

The boundary area is drawn in complete form using computational program with dimension in Figure 2. Meshing is a substantial process in this numerical study. This simulation uses Quad Pave mesh model. Meshing which applied in this study shown in Figure 3. Boundary condition which used in this simulation is shown in Table 2.

Table 2: Boundary Condition.

BOUNDARY	INPUT	VALUE
Fluid	ρ	1.225 kg/m ³
	μ	1.86 x 10 ⁻⁵ N.s/m ²
Inlet	Velocity inlet	13, 26, 40 m/s
	Turbulence length	0.8 %
	Length scale	1 m
Outlet	Outflow	
Turbulence model	k-ε implicit	



Figure 1. Geometry of sedan car YRS R4 Doors.

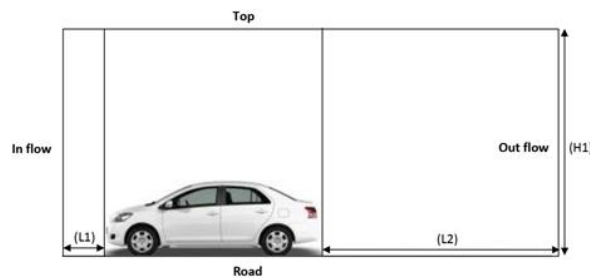


Figure 2. Computational domain.

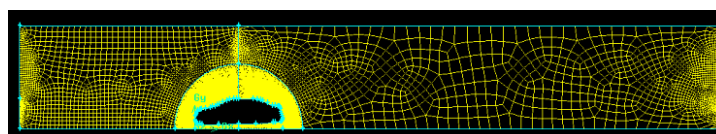


Figure 3. Computational mesh.

4. RESULT & DISCUSSIONS

The numerical simulation result is compared with previous experimental study [4] to validate the result. In validation process, numerical simulation using ideal gas velocity inlet 40 m/s and $Re = 10^6$. The previous experiment result total $C_D = 0.574$ and this present numerical simulation is 0.567. This simulation has 1.2% of error compared with C_D value of previous experiment.

Coefficient of pressure around the car is shown in Figure 4. The red line represents C_P for upper body of car while the black line shows C_P for underneath area. The C_P value in upper body of the car has higher value than underneath area. This condition is briefly explained by velocity contour in Figure 5. Free stream flow from the inlet area and collide the front area of the car at the point of stagnation. The stagnation is a wake area which has lower pressure value below the atmosphere pressure. Then the free stream separate and flow through upper and underneath area. Upper area is an open area and free stream contact and flow through the upper body smoothly. The upper area stream has low velocity because of friction between the air molecules. There is no extreme pressure drop in upper body. Extreme pressure different occur when the free stream flows in underneath area, because the area between the car and the road is vacuum. The narrow area creates suction effect and let the fluid flow in high velocity. The underneath stream has high velocity and lowest pressure. This condition creates high pressure difference for underneath free stream compered by atmosphere condition.

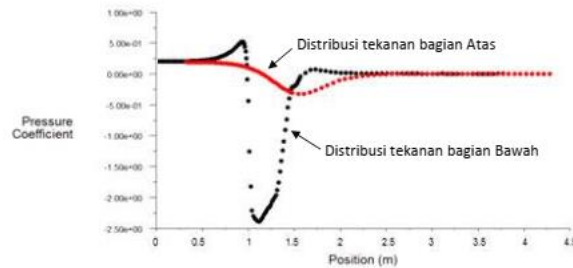


Figure 4. The coefficient of pressure around the car in 40 m/s

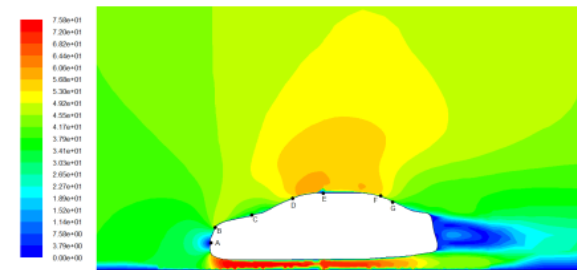


Figure 5. The velocity contour around the car in 40 m/s

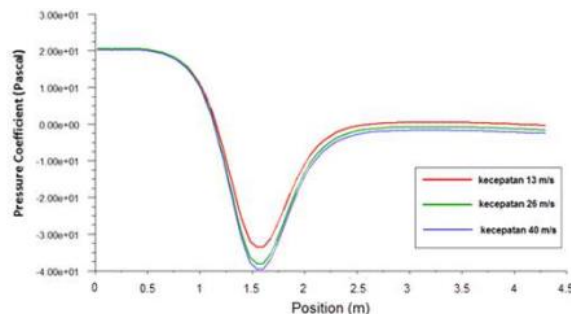


Figure 6. The coefficient of pressure around the car in 13, 26 and 40 m/s

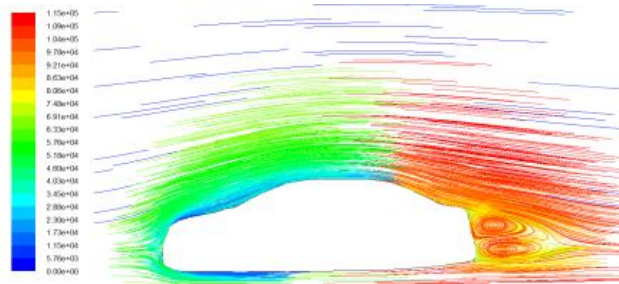


Figure 7. The velocity contour around the car in 40 m/s

Table 3. The Length of twin Vortex in various velocity inlet

Velocity (m/s)	Length of twin vortex (m)
13	0.974
26	0.985
40	0.998

The underneath coefficient of pressure at various velocities of the car is shown in Figure 4. High velocity stream results the lowest value of C_p . When the fluid flows in high speed through a surface, it has low pressure according to the Bernoulli law. Free stream flow from the inlet area and hit stagnation point in front of the car. The stream separated and flows to upper and underneath body. Then stream from upper and underneath area flow back through wake area in the tail of the car. The back flow is shown as twin vortex in Figure 7. The twin vortex has different length in various velocities as shown in Table 3. The longest twin of vortex length resulted by velocity 40 m/s.

Table 3 can explain if the velocity increasingly rise, the length of the twin vortex will also increase. Visually the flow traces created from each different speed are similar as marked by the formation of 2 vortices. But length of twin vortex has a different appearance on each of them with speed. The length of vortex which generated by 40 m / s stream is 0.977 m. While fluid flow with a velocity of 13 m / s produces length of vortex 0.974 m. The fluid which flow at 26 m/s has 0.985 m length of vortex. So, the higher velocity of the vehicle is resulted longer length of vortex. The drag force is an aerodynamic force which causes velocity reduction effect in the vehicle. The magnitude of drag force on the vehicle is determined by flow velocity, cross-sectional area, shape and weight of the car.

Table 4. Coefficient of drag for velocity inlet

Velocity (m/s)	Coefficient of Drag (C_D)
40	0.5673

5. CONCLUSION

Based on the simulation results regarding the effect of the variation of velocity on the flow characteristics of car that have been carried out numerically, it can be concluded that the area between the car and the road is a vacuum and creates suction effect to let the fluid flow in high velocity. The underneath stream has high velocity and lowest pressure. This condition creates high pressure difference for underneath free stream compared by atmosphere condition. This work informs also that high velocity stream results the lowest value of C_p . When the fluid flows in high speed through a surface, it has low pressure. The coefficient of drag in this research is about 0.567. The error of C_d from this study compared to previous research study is about 1.2%.

6. ACKNOWLEDGEMENT

The author would like to express their gratitude for the support given by ITATS. The authors are grateful to Prof. Ming Jyh Chern, Prof. Nyoman PA and Prof. Susilo A for their helpful discussions as well.

7. REFERENCES

- [1] MOHAMMED-KASSIM, Z. & FILLIPONE, A., “Fuel savings on a heavy vehicle via aerodynamic drag reduction”, *Transportation Research Part D: Transport and Environment*, v. 5 n. 15, pp. 275–284, 2010
- [2] NORWAZAN, A. R., KHALID, A. J., ZULKIFFLI, A. K., NADIA, O., Fuad, M. N., “Experimental and numerical analysis of lift and drag force of sedan car spoiler”, *Applied Mechanics and Materials*, v. 165, pp. 43–47, 2012
- [3] KUREC, K., REMER, M., “The influence of different aerodynamic setups on enhancing a sports car’s braking” *International Journal of Mechanical Science*, v. 164, pp. 105-140, 2019
- [4] HU, X. & Wong, E. T. T., “A Numerical Study On Rear-spoiler Of Passenger Vehicle”, *World Academy of Science, Engineering and Technology*, v. 57, pp. 636–641, 2011
- [5] SHARMA, R. B. & BANSAL, R. “Aerodynamic Drag Reduction of a Passenger Car Using Spoiler with VGs”, *International Journal of Engineering Research and Applications (IJERA)*, pp. 256-263, 2014
- [6] KUMAR, V. N., NARAYAN, K. L., RAO, L. N., SRI RAM, Y., “Investigation of Drag and Lift Forces over the Profile of Car with Rears spoiler Using CFD”, *International Journal of Science and Research* v. 1 n. 8, pp. 331-339, 2013
- [7] BEZAVADA, S. & KODALI, S. P., “Numerical simulation of air flow over a passenger car and the Influence of rear spoiler using CFD” *International Journal of Advanced Transport Phenomena*, v. 1 n. , pp. , 2012
- [8] KUYA, Y., TAKEDA, K., Zhang, X., “Computational investigation of a race car wing with vortex generators in ground effect”, *Journal of Fluids Engineering, Transactions of the ASME* v. 132 n. 2, pp. 0211021–0211028, 2010
- [9] ISLAM, M. R., Hossain, M. A., Mashud, M., Gias, M. T. I., “Drag Reduction of a Car by Using Vortex Generator”, *International Journal of Scientific & Engineering Research*, v. 4 n. 7, pp. 1298 - 1302, 2013.
- [10] SELVARAJU, P., PARAMMASIVAM, M., SHANKAR, D., “Analysis of drag and lift performance in sedan car model using CFD”, *Journal of Chemical and Pharmaceutical Sciences*, special issue 7, pp. 429-435, 2015.
- [11] STAMSURI, S., SUHANI, S., MAULANA, H. S., “Simulasi numerik kombinasi perpindahan panas konveksi alami dan radiasi pada square cavity”, *Turbo : Jurnal Program Studi Teknik Mesin* v. 8 n. 2., pp. 208-213, 2019
- [12] GUILMINEAU, E., “Computational study of flow around a simplified car body”, *Journal of Wind Engineering and Industrial Aerodynamics*, v. 96 n. 6–7, pp. 1207–1217, 2008
- [13] LILLAHULHAQ, Z. & MAULANA, H. S., “Pengaruh Model Turbulensi Aliran Terhadap Simulasi Numerik Aircurtain” *MEKANIKA: Jurnal Teknik Mesin*, v. 5 n. 2, 40–45, 2020
- [14] MENTER, F. R. & KUNTZ, M., “Adaptation of Eddy-Viscosity Turbulence Models to Unsteady Separated Flow Behind Vehicles”, *The Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains*, Berlin, Springer Verlag, pp 339-352, 2004