VIETNAM ACADEMY OF SCIENCE AND TECHNOLOGY

Vietnam Journal of MECHANICS

Volume 36 Number 3

ISSN 0866-7136

VN INDEX 12.666

3

2014

Vietnam Journal of Mechanics, VAST, Vol. 36, No. 3 (2014), pp. 235 – 244

ESTIMATION OF CAR AIR RESISTANCE BY CFD METHOD

Phan Anh Tuan*, **Vu Duy Quang**
Hanoi University of Science and Technology, Vietnam
*E-mail: tuan.phananh@hust.edu.vn
Received July 07, 2014

Abstract. Total car resistance is including rolling resistance and air resistance. Rolling resistance comes from car tires when it rolls over the roads with car weight. Air resistance comes from the body when it moves in the air with car body surface area. The air resistance of a car depends upon its shape. The bigger the surface area of a car body, the more air molecules the car will hit and so the larger the air resistance. This paper will mention to estimation of car air resistance by computational fluid dynamics (CFD) method. A 3D car body has used for simulation in ANSYS FLUENT CFD software. The k-ε turbulence model and segregated implicit solver was used to perform computation in this study.

Keywords: Rolling resistance, air resistance, surface, CFD, turbulence.

1. INTRODUCTION

The two main forces that slow the car down are rolling resistance and air resistance. Rolling resistance comes from friction between the car tires and road surface. Air resistance comes from friction between car body surface and air.

In fluid dynamics, air resistance (a type of friction, or fluid resistance, another type of friction) refers to forces acting opposite to the relative motion of any object moving with respect to a surrounding fluid. This can exist between two fluid layers (or surfaces) or a fluid and a solid surface. Unlike other resistive forces, such as dry friction, which are nearly independent of velocity, air resistant force depend on velocity. Most of us do not think of air or wind as a wall. At low speeds and on days when it is not very windy outside, it is hard to notice the way air interacts with our cars. But at high speeds, and on exceptionally windy days, air resistance has a tremendous effect on the way a car accelerates, handles and achieves fuel mileage.

For several decades, cars have been designed with aerodynamics, the study of forces and the resulting motion of objects through the air, in mind. Carmakers also have come up with a variety of innovations that make reduce the effect of the air resistance. Engineers also have developed several ways of doing this. For instance, more rounded designs and shapes on the exterior of the cars are crafted to channel air in a way so that it flows around the car with the least resistance possible.

However, as the passenger car needs enough capacity to accommodate passengers and baggage so there is minimum necessary space for its engine and other components. It is extremely difficult to realize an aerodynamically ideal body shape. The car has a body shape that is not perfect an ideal streamline shape as seen in fishes and birds. Such a body shape is inevitably accompanied by flow separation at the rear end. Two major element that have major influence on the air resistant coefficient of a bluff object are the roundness of its front corners and the degree of taper at its rear end [1].

A variety of studies of the aerodynamic influence of vehicle rear end shapes have been researched. It is well known that the rear-end shape of a car is one of the important elements which governs aerodynamic resistance and lift [2]. An other research studied about aerodynamic of a race car through CFD analysis and experiments. In that research, the authors demonstrated that there is a significant change in the coefficients of lift and air resistance of the model race car when a more streamlined body design is adopted [3]. Some concept cars with good aerodynamic application also have designed based on Autodesk 3ds Max drawing and CFD analysis [4]. Fig. 1 below shows a design of a polygonal concept car by Damjanović et al. [4].

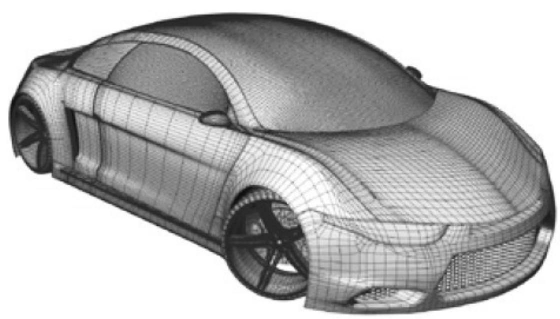

Fig. 1. A design of polygonal concept car

Consequently, having a car designed with airflow in mind means it has less difficulty accelerating and can achieve better fuel economy numbers because the engine does not have to work nearly as hard to push the car through the wall of air. This paper will mention to car air resistance through computational fluid dynamics (CFD) studied method. A 3D car body of Ford Escort MK has been used for simulation in ANSYSY FLUENT software. The k-ε turbulence model and segregated implicit solver was used to perform computation in this study.

2. APPLICABLE THEORY

2.1. Car resistances

Two parts of car resistance, rolling resistance and air resistance could be obtain as in following equations.

2.1.1. *Rolling resistance*

The force resisting the motion when a car rolls on a road is called the rolling resistance or rolling friction. Fig. 2 below shows an explanation of car rolling resistance.

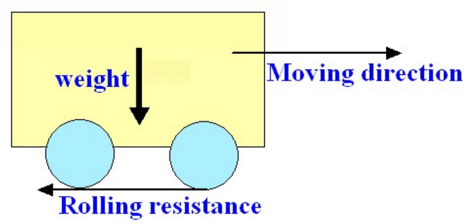

Fig. 2. Car rolling resistance explanation

The rolling resistance can be expressed as

$$F_R = Cmg, \tag{1}$$

where F_R is rolling resistance, C is rolling resistant coefficient, m is mass of car body, g is acceleration of gravity ($9.81\,\mathrm{m/s^2}$). The rolling resistance can alternatively be expressed as

$$F_R = C_l W r, \tag{2}$$

where W is weight of the car ($W = mg$), C_l is rolling resistant coefficient with dimension length, r is radius of tire. For cars, rolling resistant coefficients for a pneumatic tires on dry roads can be calculated as follow

$$C = 0.005 + \frac{1}{p}\left[0.01 + 0.0095\left(\frac{v}{100}\right)^2\right], \tag{3}$$

where p = tyre pressure, v = velocity.

2.1.2. *Air resistance*

The air resistance of a car moves on road is given by the following expression

$$F_A = \frac{1}{2} A\rho C_A v^2, \tag{4}$$

where C_A is the air resistant coefficient, A is the projected frontal area of the car, ρ is the density of air, v is the velocity of the car relative to the air.

Eq. (4) shows that to calculate air resistance we need to know three things: the air resistant coefficient (C_A); the frontal area of car (A) and the velocity of air past the car (v). This equation shows important point-aerodynamic forces are proportional to the square of the velocity. That means we quadruple the air resistance when we double the velocity. Since velocity is never the item that is pretended to be decreased and the density of air is not even possible to change, the aerodynamic resistance can only be reduced or minimized by manipulating the car's design characteristics, obtaining lower air resistant coefficients (C_A), or even reducing the car's frontal area.

2.2. Continuous equations

The continuity and momentum equations, Navier-Stokes equations, with a turbulence model are used to solve the airflow as in the following equations [4, 5].

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0, \tag{5}$$

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\frac{1}{\rho}\left(\frac{\partial \tau_{xy}}{\partial y}+\frac{\partial \tau_{xz}}{\partial z}\right)+F_x, \tag{6}$$

$$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\frac{1}{\rho}\left(\frac{\partial \tau_{yx}}{\partial x}+\frac{\partial \tau_{yz}}{\partial z}\right)+F_y, \tag{7}$$

$$u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial z}+\frac{1}{\rho}\left(\frac{\partial \tau_{zx}}{\partial x}+\frac{\partial \tau_{zy}}{\partial y}\right)+F_z, \tag{8}$$

where u, v, w are axis components of velocity vector, ρ is density of air, p is static pressure, τ is shear stress, $\tau_{xy}=\tau_{xy}\tau_{xz}=\tau_{zx}\tau_{yz}=\tau_{zy}$ and F_x, F_y, F_z are body forces.

2.3. *k*-ε turbulent model

The k-ε turbulence model is the most common model used in computational fluid dynamics to simulate turbulent conditions. It is a two equation model which gives a general description of turbulence by means of two transport equations. The original impetus for the k-ε model was to improve the mixing-length model, as well as to find an alternative to algebraically prescribing turbulent length scales in moderate to high complexity flows. The first transported variable determines the energy in the turbulence and is called turbulent kinetic energy, k. The second transported variable is the turbulent dissipation, ε, which determines the rate of dissipation of the turbulent kinetic energy.

2.3.1. *For turbulent kinetic energy k*

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\varepsilon-Y_M+S_k. \tag{9}$$

2.3.2. *For dissipation*

$$\frac{\partial}{\partial t}(\rho \varepsilon)+\frac{\partial}{\partial x_i}(\rho \varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}\left(G_k+C_{3\varepsilon}G_b\right)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\varepsilon, \tag{10}$$

where,
μ_t is turbulent viscosity and μ is fluid viscosity.
G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients.
G_b is the generation of turbulence kinetic energy due to buoyancy.
Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate.
$C_{1\varepsilon}, C_{2\varepsilon}, C_{3\varepsilon}$ are constants.
σ_k, σ_ε are the turbulent Prandtl numbers for k and ε, respectively.
S_k and S_ε are user-defined source terms.

2.3.3. *Turbulent viscosity*

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}, \tag{11}$$

where, C_μ is constant.

3. SIMULATION PARAMETERS

3.1. Car model and domain

In this study, for estimation car air resistance, a Ford Escort MK car has been used for simulation in CFD software. The principle parameters of the car are 4.3 m in length, 1.7 m in width and 1.4 m in height. It is a five seats sedan car. Tab. 1 and Fig. 3 show the principle dimensions and the 3D picture of the Ford Escort MK.

Table 1. Parameters of Ford Escort MK

Length	4.3 m
Width	1.7 m
Height	1.4 m
Body weight	1048 kg
Engine	1.3 L
Passenger	5 seats
Fuel	gasoline
Power net	59 HP/5000

A 3D Ford Escort MK car has been modeled for simulation in CFD software. The car model is created by a 3D drawing software. It is a full scale model. The domain for CFD simulation in this study is a virtual wind tunnel of a semi cylinder with 7 m in radius and 39 m in length. Fig. 4 shows the domain for CFD simulation with the Ford Escort MK car.

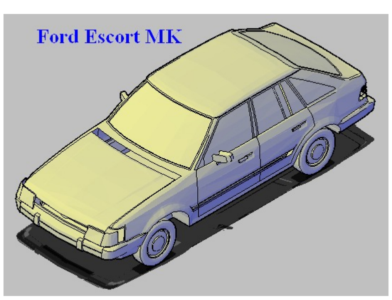

Fig. 3. 3D picture of Ford Escort MK

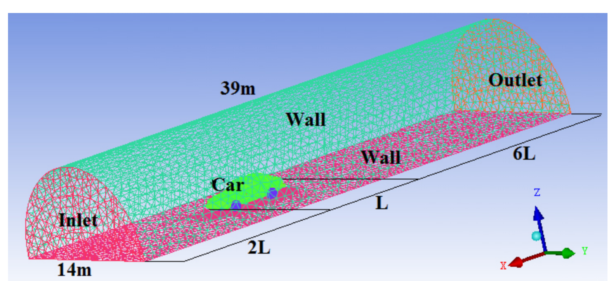

Fig. 4. Domain for CFD simulation

3.2. Boundary conditions

Boundary condition for FCD simulation are: car and virtual wind tunnel are stood and set as walls, airflow is blew from inlet with a range of velocities from 1.39 m/s to 44.44 m/s (equal with velocity range from 5 km/h to 160 km/h), temperature is set at 27^0C ($\approx 300^0$K). The outlet boundary condition is set to pressure outlet with the gauge pressure of 0 Pa. The density of air is set as 1.225 kg/m^3 and the viscosity of air is 1.7894×10^{-5} kg/(ms). Coefficients for k-ε turbulence model are standard and could be obtained from ANSYS FLUENT user handbook such as $C_{1\varepsilon} = 1.44, C_{2\varepsilon} = 1.92, C_\mu = 0.09$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.3$, etc. Tab. 2 below shows the velocity of the airflow at the inlet boundary conditions.

Table 2. Velocity of the airflow at the inlet boundary conditions

km/h	5	10	20	40	60	80	100	120	140	160
m/s	1.4	2.8	5.6	11.1	16.7	22.2	27.8	33.3	38.9	44.4

4. RESULT AND DISCUSSION

4.1. Velocity distribution

Fig. 5 shows the velocity vector distribution around the car in simulation that obtained from simulating with different car velocities.

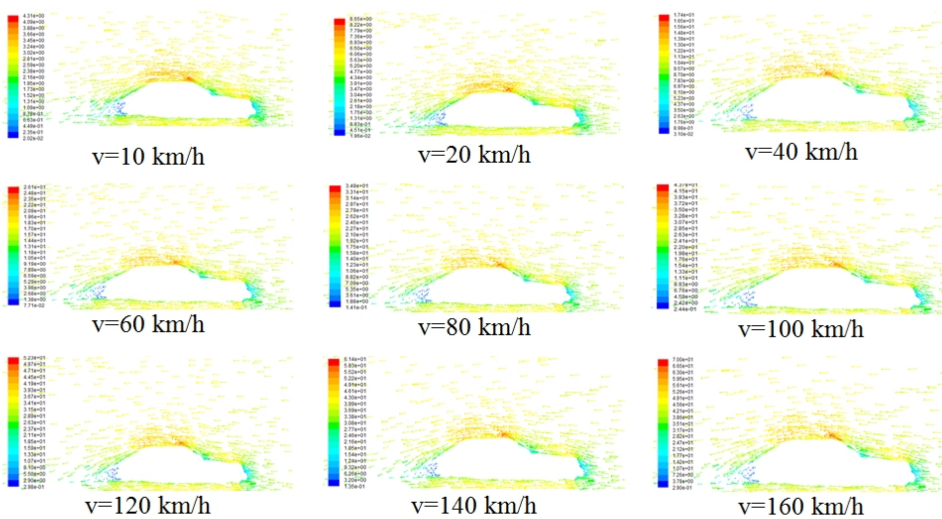

Fig. 5. Velocity distribution in simulation

The velocity vector distributed pictures show that there are four zones that appear turbulent flows around the car body. They are head zone (I), bonnet-front glass intersection zone (II), front glass-ceiling intersection zone (III) and rear zone (IV). As shown in Fig. 6, the biggest turbulent magnitude around the car body during it moving is the rear zone.

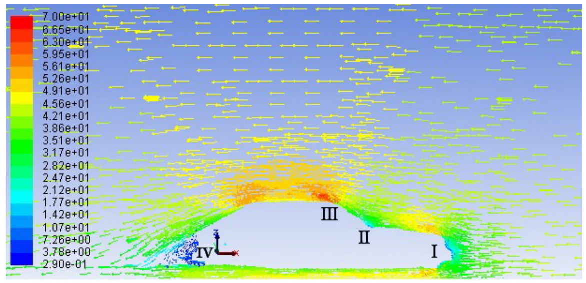

Fig. 6. Turbulent areas around car body

4.2. Pressure distribution

Fig. 7 shows the pressure distribution around the car in simulation that obtained from different car velocities.

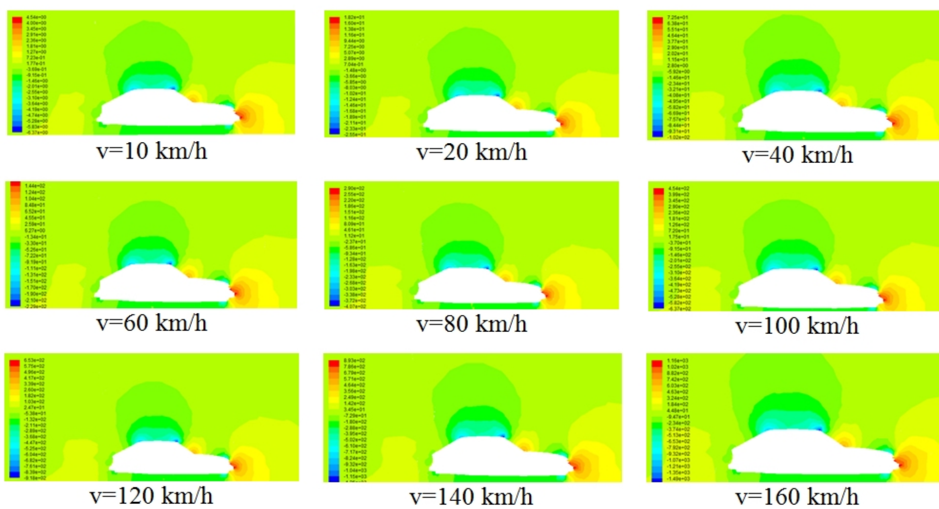

Fig. 7. Pressure distribution around car body

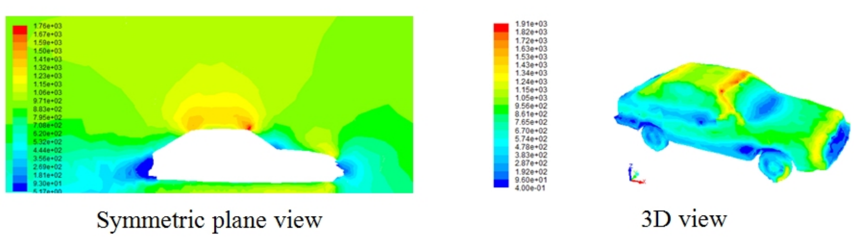

Fig. 8. Dynamic pressure distribution around car body

Fig. 8 displays the result of simulation of dynamic pressure distribution around car body at speed of 140 km per hour. This figure shows that when the car moves forward, the high pressure appears at the ceiling plane. The highest pressure appears at front glass-ceiling intersection area (zone III with orange color).

4.3. Air resistance

4.3.1. *Theoretical calculation*

Air resistance of the Ford Escort MK could be calculated by Eq. (4) as follow

$$F_A = \frac{1}{2} A\rho C_A v^2,$$

where $A = 1.8$ m^2, $\rho = 1.225$ kg/m^3, $C_A = 0.39$.

After calculation, we have the result of air resistance of the car as shown in Tab. 3.

Table 3. Air resistance in theoretical calculation

Velocity (km/h)	5	10	20	40	60	80	100	120	140	160
Velocity (m/s)	1.4	2.8	5.6	11.1	16.7	22.2	27.8	33.3	38.9	44.4
F_A (N)	0.8	3.3	13.3	53.1	119.5	212.3	331.8	477.7	650.3	849.2

4.3.2. *Simulation result*

Tab. 4 shows the simulation result of air resistance of the Ford Escort MK in CFD with ANSYS FLUENT software.

Table 4. Air resistance in simulation

Velocity (km/h)	5	10	20	40	60	80	100	120	140	160
Velocity (m/s)	1.4	2.8	5.6	11.1	16.7	22.2	27.8	33.3	38.9	44.4
F_A (N)	0.9	3.5	13.9	55.0	123.0	217.9	337.8	486.2	663.5	860.5

Fig. 9 shows the result of air resistance in theoretical calculation and in simulation with CFD software. In this figure, the vertical axis is the values of air resistance (N) and the horizontal axis is car velocity (km/h). In this figure, there is a little difference between the result of theoretical calculation and the result of simulation. However, the trend (response of air resistance with car velocities) are almost similarly.

Beside getting the total air resistance, with CFD method, we could also obtain the component air resistances, pressure air resistance and viscous air resistance. Fig. 10 shows the graphs of the component air resistances and the total air resistance. In this figure, the vertical axis is the values of component air resistance F (N) and the horizontal axis is car velocity v (km/h).

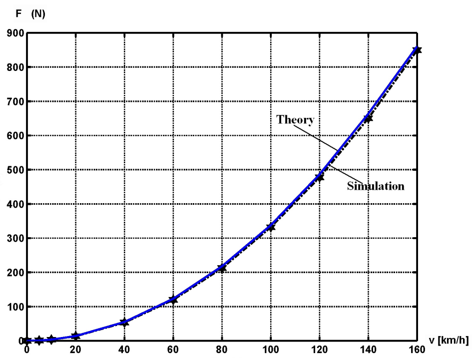

Fig. 9. Air resistance in theoretical calculation and in simulation

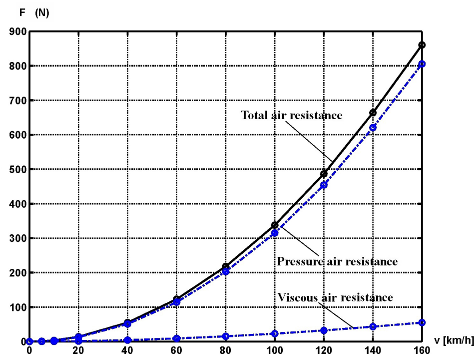

Fig. 10. Graphs of component air resistances

4.4. Lift force

Fig. 11 displays the graph of lift force, F_L, against car velocity, v. In this figure, the vertical axis is the values of air lift force F_L(N) and the horizontal axis is car velocity v (km/h). It shows that the lift force of the car is not so big comparison with the weight of the car (10280 N without passenger loading). It means the car is stable during it moves on roads.

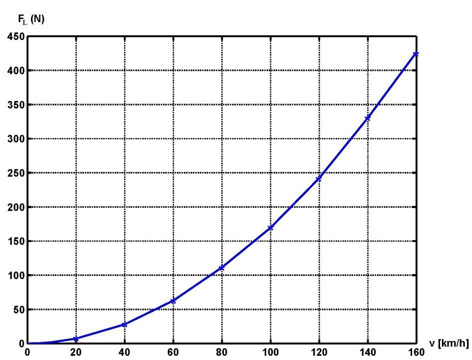

Fig. 11. Graphs of lift force

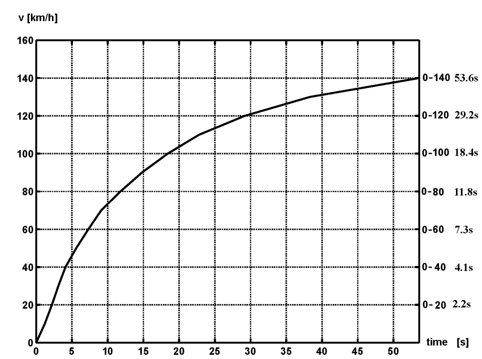

Fig. 12. Graph of car acceleration

4.5. Car accelerations

Fig. 12 displays car accelerations of the Ford Escort MK. In this figure, the vertical axis is the values of reaching velocities of the car from its standing and the horizontal axis is time period (sec). For example, the acceleration of the car for reaching velocity of 100 km/h from its standing is 18.4 seconds.

5. CONCLUSIONS

Calculation car air resistance by CFD method is a modern and popular research method on the world. By using CFD method, we could obtain an overview of car air resistance from velocity distribution contour and pressure distribution contour.

In theoretical calculation, it is very difficult to solve the complicated boundary layer airflows around the car that come from multi surface sections. In simulation, the advantage is the software has solved the boundary layer airflows for us. Other addition advantage in simulation is we could also obtain lift force and component air resistances, not only get the total air resistance.

The velocity distribution contour and the pressure distribution contour in Fig. 5-8 show that there are four zones appear turbulent airflows. They are head zone (I), bonnet-front glass intersection zone (II), front glass-ceiling intersection zone (III) and rear zone (IV). These turbulent flow will increasing the air resistance of the car. Because of this factor, in modern cars, the bodies have been designed with aerodynamic forms.

As shown in Fig. 10, the results of theoretical calculation and simulated calculation of air resistance of the car are similar. There is a little different between the result of theoretical calculation and simulated calculation. This difference may come from the chosen air resistant coefficient C_A. In theoretical calculation, the air resistant coefficient is set as a constant, $C_A = 0.39$ for all of car velocities but in simulation this coefficient is different base on car velocities and it is more accurate with the form of the car body.

Fig. 12 shows that the acceleration of the Ford Escort MK car is not so quick as other modern car with similar engine power. This means the Ford Escort MK has a not good performance car body. The car body was designed with only multi joined planes, not aerodynamic body form. Since a car designed with airflow in mind will have less difficulty accelerating and can achieve better fuel economy numbers because the engine does not have to work nearly as hard to push the car through the wall of air, the Ford Escort MK should be renew its body with an aerodynamic form for increasing its performance.

REFERENCES

[1] A. A. Karamallah and A. K. Wahab. Experimental determination of drag coefficient on different automobiles geometry. *Eng. & Tech. Journal*, **29**, (2011), pp. 3043–3057.

[2] V. Sirenko, R. Pavlovsky, and U. S. Rohatgi. Methods of reducing vehicle aerodynamic drag. In *the ASME 2012 Summer Heat Transfer Conference*, (2012), pp. 97–102.

[3] A. Chainani and N. Perera. CFD investigation of airflow on a model radio control race car. In *proceedings of the World Congress on Engineering 2008*, Vol. II, London, UK, (2008).

[4] D. Damjanović, D. Kozak, M. Zivić, Z. Ivandić, and T. Baskarić. CFD analysis of concept car in order to improve aerodynamics. In *International Scientific and Expert Conference TEAM 2010*, Kecskemot, (November, 2010).

[5] Y. Nakayama and R. Boucher. *Introduction to fluid mechanics*. Butterworth-Heinemann, (2000).

VIETNAM ACADEMY OF SCIENCE AND TECHNOLOGY
VIETNAM JOURNAL OF MECHANICS VOLUME 36, N. 3, 2014

CONTENTS