



ISSN: 2329-8243 (Print)
ISSN: 2329-8235 (Online)



ARTICLE

THE STUDY OF THE AERODYNAMICS OF A ONE-FOURTH MODEL PASSENGER VEHICLE

Harley Kaur*

College of Engineering, Design and Physical Sciences, Brunel University London, Middlesex UB8 3PH, United Kingdom
*Corresponding Author E-mail: Harley@hotmail.com

This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

ARTICLE DETAILS

ABSTRACT

Article History:

Received 16 September 2019
Accepted 17 October 2019
Available online 20 November 2019

The numerical study for the calculation of aerodynamic forces arising in a vehicle has become more conventional. The methods usually employed for the analysis of such forces in vehicles are designed after the devise of prototypes in wind tunnels, which are very expensive demanding a large amount of equipment, facilities and skilled labor. Many researchers have been trying to understand the complex flow patterns around vehicles, developing measurement techniques in wind tunnels and computational efforts. Few of the studies, however, are concentrated in the region of wheelhouse. Generally, the studies include the complete vehicle and indirectly study the region of the wheelhouse. The aim of this study is to identify, through numerical simulations, the effects of internal elements of the wheelhouse, the wheelhouse itself, the wheel and tire assembly in drag force of a passenger vehicle, through the modelling of a quarter car.

KEYWORDS

Aerodynamics, computational fluid dynamics, wheelhouse, vehicle aerodynamics.

1. INTRODUCTION

The numerical study for the calculation of aerodynamic forces arising in a vehicle has become more conventional. The methods usually employed for the analysis of such forces in vehicles are designed after the devise of prototypes in wind tunnels, which are very expensive demanding a large amount of equipment, facilities and skilled labor.

Currently companies have performed three-dimensional construction of their models, thus facilitating processes of tools construction, assembly interference checking and also allowing the numerical analysis.

Most of the articles related to numerical studies of vehicle aerodynamics aim at the vehicle body or phenomena associated with lateral wind generally disregarding components as the wheel geometries, tire, brake assembly (disc and caliper), suspension, shock absorber, stub axle and wheelhouse.

This article aims to determine the flow influence on a passenger vehicle using a one-fourth model of a complete vehicle as seen in Figure 1.

Many researchers have been trying to understand the complex flow patterns around vehicles, developing measurement techniques in wind tunnels and computational efforts. Few of the studies, however, are concentrated in the region of wheelhouse. Generally, the studies include the complete vehicle and indirectly study the region of the wheelhouse.

The aim of this study is to identify, through numerical simulations, the effects of internal elements of the wheelhouse, the wheelhouse itself, the wheel and tire assembly in drag force of a passenger vehicle, through the

modelling of a quarter car.

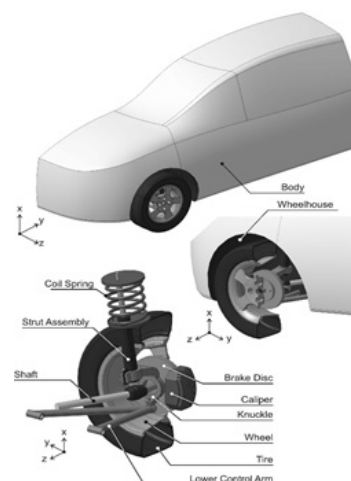


Figure 1: One-fourth model and components to be evaluated.

2. MATHEMATICAL MODEL AND BOUNDARY CONDITIONS

The finite volume method was used to solve the conservation equations applied to an unstructured mesh. The equations solved for the model are respectively, the mass and the momentum conservation equations. It is used the standard $k - \epsilon$ model for the turbulence treatment and the fluid is air at a temperature (T) of 25°C. The flow regime is transient, varying

the inlet velocity (v_{in}) of the fluid and the soil (v_g) from zero to 33.5 m/s ($v_{in} = v_g$). Mach number (M) for the proposed condition in this analysis is less than 0.3 so the flow can also be considered as incompressible and thermal effects are neglected, i.e. isothermal flow. The wheel angular velocity (w) can be expressed as a function of ground speed and the dynamic tire radius (r_d). The boundary conditions are: non-slip flow for the vehicle surfaces, free-slip flow on the top and side walls and symmetry for the zx plane as presented in Figure 2.

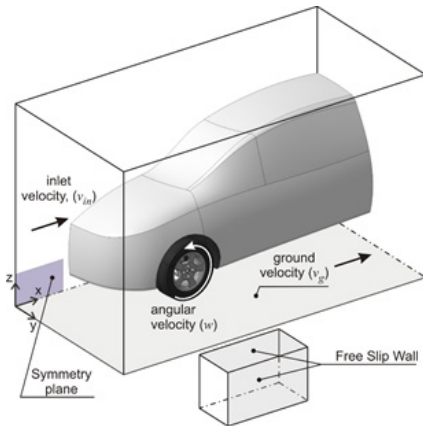


Figure 2: Boundary conditions for the numerical model.

A simplification of some elements was adopted for the proposed model given that other geometries exert little influence on the flow. These elements are the hood of the constant velocity joint, wheel bolts case, braking set details, etc. The respective changes can be observed in Figure 3.

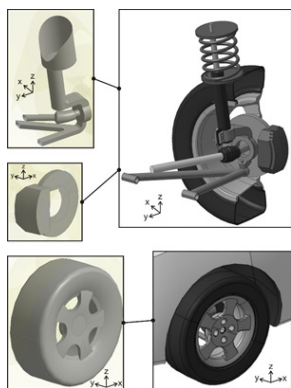


Figure 3: Geometrical simplifications.

The approach presented by Stern and Wilson to define and verify the mesh is used in this paper. They discuss the mesh dependency on the results focusing the element size definition in order to validate the CFD models. The methodology considers an increase of the mesh density for the same boundary condition using predefined ratios. This procedure must be performed in such a way that property variation or small variations are not present. When this condition is satisfied the solution is considered independent of the mesh. The computational domain for the last refinement iteration process has 2.8 million elements. In Figure 4(a) it is possible to observe the surface elements applied to the vehicle body. A greater density of elements has been applied inside the wheelhouse and the tire. Figure 4(b) and Figure 4(d), and an auxiliary view is presented for the volume density elements verification, Figure 4(c).

3. RESULTS AND DISCUSSION

The numerical study was performed in transient regime. In order to demonstrate the highest gradients of the quantities involved, air input speed was chosen to 33.5 m/s (120 km/h). The pressure variation as

a function of the model surface is indicated on the color map of Figure 5(a). Sections in the computational domain were constructed and can be observed in Figure 5(c). Figure 5(b) shows a view (cut) in the xy plane and height (z) equal to r_d . This section region is protected by the vehicle body which contributes to a small pressure variation against the other variations. The sections in the xz plane, Figure 5(d) and Figure 5, detail A, show respectively, the pressure gradient in the section and on the surface geometry tire/wheel [1]. Also in Figure 5 the frontal location of the vehicle shows stagnation regions, this behavior indicates consistency in the numerical results.

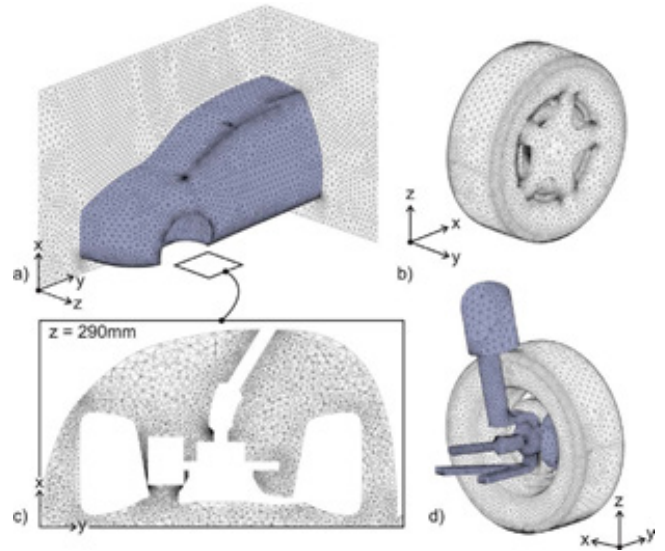


Figure 4: (a) Triangular elements in the region of the vehicle body; (b) Surface elements applied on the wheel geometry; (c) Detail view in the xy plane of the wheelhouse region; (d) Superficial elements inside the wheelhouse.

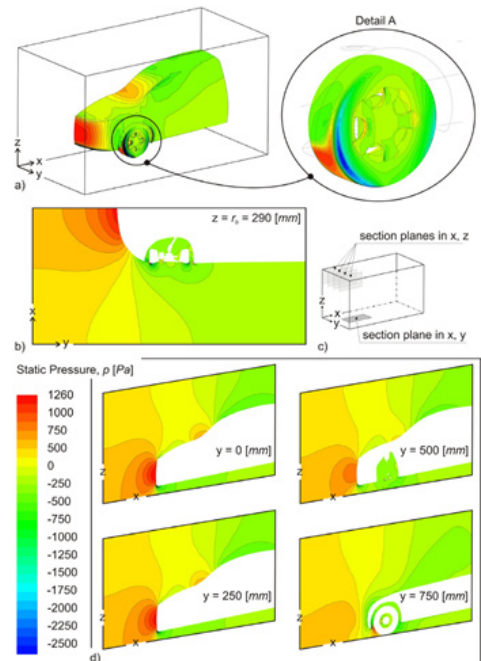


Figure 5: (a) Pressure gradient applied to the vehicle surface; (b) Pressure gradient on xy plane at axis z equal to r_d ; (c) Illustration showing the positions of sections; and (d) Pressure gradient applied to xz plane at various positions.

In Figure 6 streamlines colored by velocity gradient were constructed in order to make evident the flow behavior. Figure 6 also shows that the vehicle external geometry does not originates vortex structures. This vortex structures occur only in the lower region of the wheel, tire and wheelhouse.

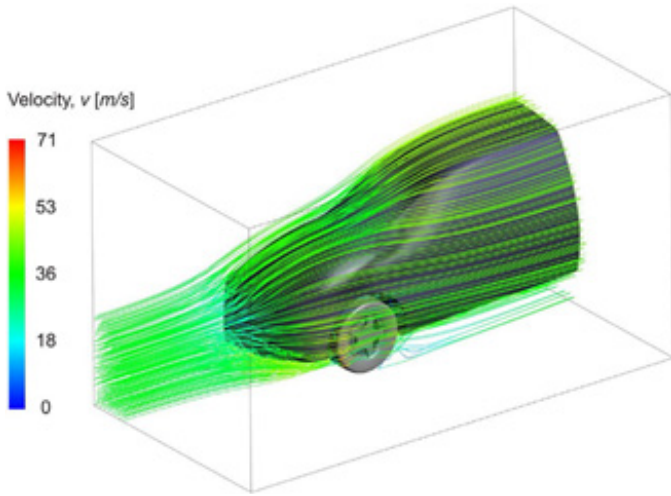


Figure 6: Streamlines in the proximity of the vehicle surface.

Vortex structures appeared depending on the geometry of the wheelhouse as shown in Figure 7. This characteristic in some cases depending on the vortex frequency emission can excite surface so as to promote noise. However, this simulation does not take into consideration such factors. Another problem associated with the formation of these vortex inside the wheelhouse is the decrease of the air flow that cools the brake system. Thus in the wheelhouse design these recirculation structures should be avoided [2].

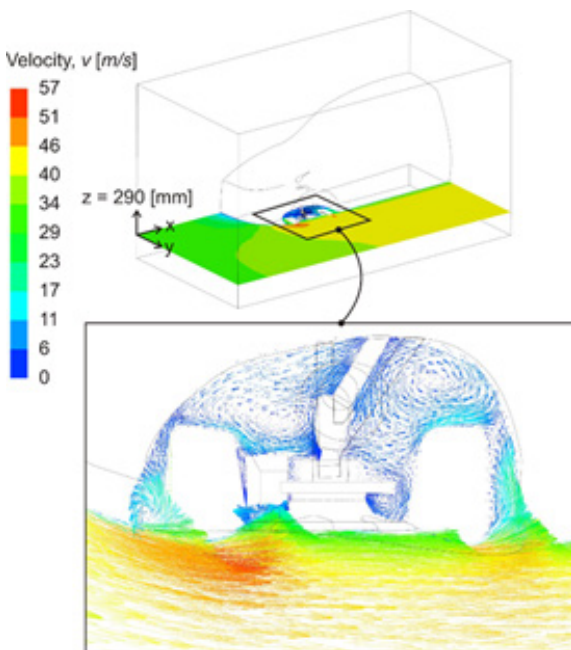


Figure 7: Cross section of the gradient and velocity vectors on the xy plane and z = 290 mm.

The aerodynamic drag force (F_a) according to Fox and McDonald is defined by Equation (1):

where: C_d —aerodynamic drag coefficient, ρ —density and A_f —frontal

area.

$$F_a = C_d \cdot \rho \cdot A_f \cdot \frac{v^2}{2} \tag{1}$$

The aerodynamic drag coefficient is not constant. It depends directly on the flow characteristics. Thus the Reynolds number is defined by Equation (2):

$$Re = \frac{\rho \cdot v \cdot L}{\mu} \tag{2}$$

where: L is the is the characteristic length chosen as $L = \frac{A_f}{P_f}$ and P_f perimeter as function of the frontal area.

The respective areas and perimeters for this model can be observed in Figure 8.

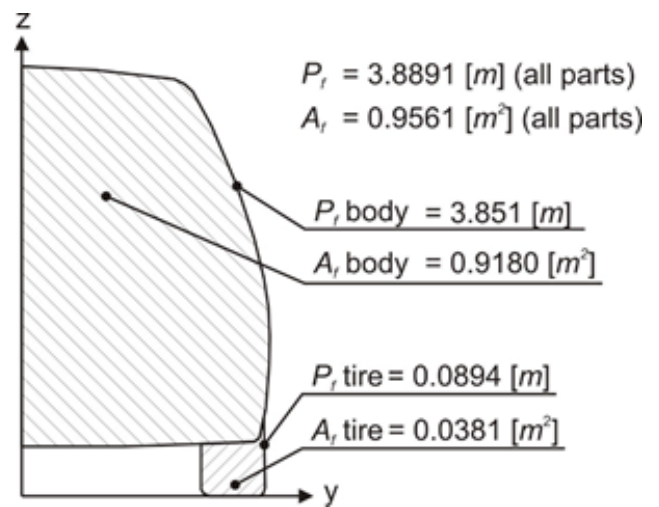


Figure 8: Vehicle frontal projection.

The numerical simulations and experimental results reported in the literature indicate that the drag coefficient values vary slightly from a given speed. In such cases, the value stabilizes, even with the increased speed and allows it to be defined as aerodynamic characteristics of the body. Gillespie, 1992 presents average values tables for the drag coefficient as a function of the vehicle category which is proper only for estimates [3,4].

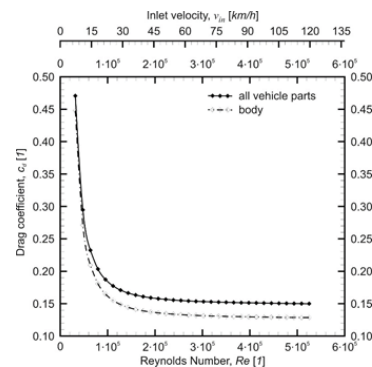


Figure 9: Inlet velocity and Reynolds number as a function of the drag coefficient.

For the verification of such feature a graph of velocity versus drag coef-

ficient was built. The vehicle body and the whole vehicle body have different drag coefficients, but show the same trend as shown in Figure 9.

Figure 10 presents the drag force projected in the x direction as a function of velocity. As expected the greatest strength comes from the vehicle body, nevertheless the geometry effect of the tire, wheel and wheelhouse are significant and together come to 16.3% of the total drag force in this direction at a speed of 33.5 (m/s) and is directly proportional to the velocity.

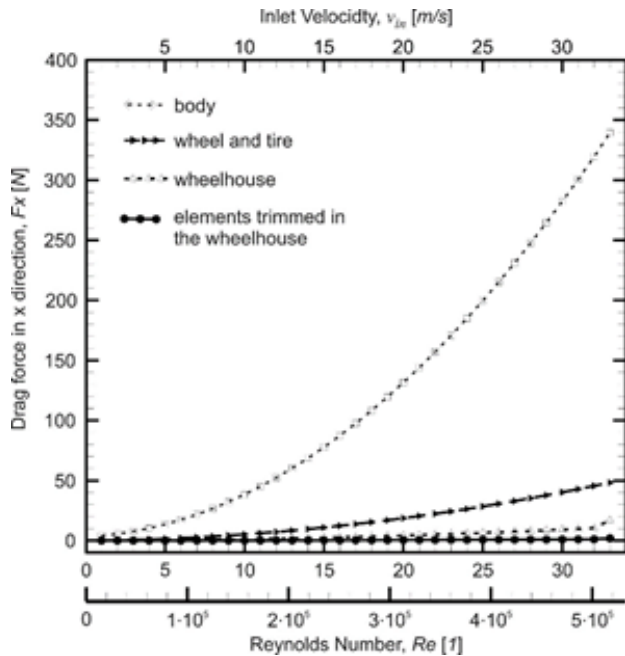


Figure 10: Inlet speed as a function of the strength's projection in the x direction.

The internal elements of the wheelhouse exert a small influence on the total drag force of the vehicle. It is about 0.4%, so that adopted simplification compared with the order of magnitude of other elements is very much reasonable.

4. CONCLUSIONS

The main conclusions of the study are:

- 1) It is possible the use of mathematical models for predicting aerodynamic loads;
- 2) Flow inside the wheelhouse is complex and presents recirculation regions;
- 3) Mechanical elements within the wheelhouse had simplified geometry and were incorporated into the model; however, the contribution to the drag force of such elements is quantitatively less than that of the wheelhouse and assembly wheel and tire;
- 4) According to the simulations, presented in paper, neglecting the effects of the box on wheels and wheel tire set is not suitable in aerodynamic vehicle research, except at low speeds where the aerodynamic loads are low for any vehicle.

REFERENCES

- [1] Cogotti, A. 2008. Evolution of Performance of an Automotive Wind Tunnel. *Journal of Wind Engineering and Industrial Aerodynamics*, 96, 667-700.
- [2] Guilmineau, E. 2008. Computational Study of Flow around a Simplified Car Body. *Journal of Wind Engineering and Industrial Aerodynamics*, 96, 1207-1217.
- [3] Regert, T., Lajos, T. 2007. Description of flow field in the wheelhouses of cars. *International Journal of Heat and Fluid Flow*, 28, 616-629.
- [4] Gillespie, T.D. 1992. *Fundamentals of vehicle dynamics*. Society of Automotive Engineers Warrendale, 400.

