NUMERICAL SIMULATION OF THE BLOCKAGE EFFECT IN WIND-TUNNELS

Marek Maciejewski
Wojciech Osmólski
Instytut Maszyn Roboczych i Pojazdów Samochodowych
Politechnika Poznańska
ul. Piotrowo 3
60-965 Poznań
Poland
E-mail: marekm@sol.put.poznan.pl

KEYWORDS

Road vehicles, wind-tunnels, aerodynamic blockage error

ABSTRACT

In the wind-tunnel tests of road vehicles, the great weight has proper reconstruction of actual movement conditions. Unfortunately, certain errors are unavoidable. One of main sources of errors is the blockage effect. This effect is here numerically investigated. As the base for our simulations it has been used the incompressible Navier–Stokes equations solved with the use of the finite volume approximation (for unstructured meshes). The program has been prepared in fully adaptive mesh refinement technique. From simulations we have first determined the computational drag coefficients, and next evaluated the effective coefficients for different heights of wind-tunnels and various boundary conditions. We have also searched for the best parameters for the linear and nonlinear correction factors.

INTRODUCTION

In experimental testing of road vehicles in wind-tunnels, about the reliability of obtained results decides the faithfulness of reconstruction of real movement conditions. In such stationary tests where the inverse movement conditions are realised (a motionless vehicle in the air or other medium stream) and the environment is artificially limited, it occurs a number of errors, which are impossible to remove and thus require accurate investigation and estimation.
For the main error sources of wind-tunnel testing one should include:

- the effect of the model scale which causes the necessity of modification of flow conditions according to the Reynolds number to preserve suitable forms of physical phenomena,
- movement of the medium in relation to the stationary floor of the tunnel results in development of a boundary layer in which the tested vehicle is partially immersed,
- the flow in the wind-tunnel is partially blocked by the vehicle, and the velocity and pressure distribution are inadequate in relation to the real movement conditions.

Other error sources are: the lack of rendition of the flow through the cooling and ventilating systems, the problems with the contact of wheels with the ground, and the lack of the rotary motion of wheels, at least the motion from own driving system.

In our further investigations we have focused on the most important source, namely on the blockage effect. We have assumed that the tested models are faithful copies of vehicles. Moreover, we have also assumed that the floor of wind-tunnel moves in accordance with the air velocity on the inflow. In our consideration we neglect the errors connected with the cooling and ventilating system, with rotation of wheels and their contact with the ground.

METHOD OF THE NUMERICAL SIMULATION

The numerical simulation of the aerodynamic phenomena around a vehicle placed in a wind-tunnel has been carried out on base of the Navier–Stokes equations for incompressible medium. Unfortunately, under such flow description, the lack in the continuity equation a time evolution term causes that this equation can be treated only as divergence-free constraint for velocity. This situation restricts often the solution of the Navier–Stokes equations to different operator (time) splitting method known, as instance, as the fractional step method, the projection method, the pressure correction method or the pressure–Poisson method.
Another possibility is better coupling of incompressible flow equations through the modification of the continuity equation, and then their solution using proper methods to so formulated problem. In the case of this strategy, the modification consists in supplementing the continuity equation by an additional term relevant to pressure. According to the form of this term, it is distinguished two fundamental solution techniques known as the penalty method and the artificial compressibility method.

In our simulation, we have used this last method connected with a special form of time integration (the dual-time integration). Moreover, the solution of the above-defined incompressible flow problems has been realised numerically with the use of the finite volume discretization for unstructured meshes. In this method, searching a solution in computational domain is carried out separately in all finite volumes, and the variation of volume-averaged variables results entirely from the equilibrium conditions on cell faces, i.e. from the flux balance defined by the approximate solution of the Riemann problem.

Proceedings 14th European Simulation Symposium
A. Verbraeck, W. Krug, eds. (c) *SCS Europe BVBA, 2002*

Taking into account the essential influence of mesh arrangement onto accuracy of results, and the fact that the optimal mesh shape can be variable and a priori unknown, the program for numerical flow simulation of vehicles in wind-tunnels has been prepared in an adaptive version. It means that the program performs a suitable standard analysis, determines the subdomains of the mesh where the solution is not sufficiently accurate, and improves the discretization in those regions until the assumed accuracy criterion has been achieved. Thus our adaptive algorithm, in comparison with the standard ones, contains two new elements: an a posteriori error estimator and a mesh refinement procedure. This adaptive procedure changes the mesh density through the bisection of the longest cell faces, and thus also finite volumes, in indicated regions.

RANGE AND CONDITIONS OF THE SIMULATION

The realisation of the numerical estimation of the blockage effect in wind-tunnels requires always the detailed formulation of the range and conditions of our considerations. First of all we shall concisely classify the wind-tunnels. However, we are not interested in overall configuration of wind-tunnels, i.e. whether it is closed- or open-return tunnel, but in the working sections of tunnels. Generally, we can distinguish the following test sections:

- closed,
- open,
- porous, i.e. with slotted walls.

In general feeling, the open-jet and slotted-wall tunnels are better solution in aspect of the blockage error reduction, but each from above mentioned solutions is not entirely good. Practically the scatter of results between different tunnels can be significant and in each case it is required a suitable correction of measured forces and calculated drag (or also lift) coefficients.

How this question looks in aspect of numerical flow simulation around a vehicle placed in a working section, for different sizes and construction solutions of test sections, i.e. for different forms of boundary conditions, will be in detail presented in relation to one vehicle model.

MODEL AND INITIAL DISCRETIZATION

As the object of our aerodynamic analysis it has been assumed a very simplified shape of a passenger car with three rear end forms: the hatchback, the squareback and the notchback. However in further descriptions in this article we will mainly refer to only one from tested shapes, namely with the squareback rear. Generally, it has no essential meaning but permits to observe more clearly some phenomena in the flow domain than in the case of more streamlined shapes.

The full aerodynamic simulation of the flow around a vehicle in wind-tunnel requires three-dimensional analyses. Since such simulations are very labour and time consuming, it is difficult to carry out wide comparative investigations. In this situation we have decided on considerations in two-dimensional space. Moreover such assumption is advantageous because the blockage effects are from the nature much visible just in results of two-dimensional simulations.

The creating simplified vehicle models (Fig.1) and their placement in a wind-tunnel has defined the configuration of computational domains. Exemplary, one of many, view of a computational domain has been presented in Fig.2. Concretely, in our investigations the heights of wind-tunnels were 3, 4, 5, 6, 12 and 25 meters. The tunnel length and the car position were always identical. The ground for discretization of computational domains has been explicit defining of boundaries through nodal points placed on profiles of a car model and wind-tunnel. Exemplary forms of initial discretization have been shown in Fig.2.

The boundary conditions (for each specific computational domain) have been defined in the range of mentioned below possibilities:

- on the vehicle profile: motionless solid boundary,
- on the left- and right-side of the tunnel: inflow and outflow boundaries, respectively,
- at the top of wind-tunnel, according to the problem under consideration, it can distinguish:
  - motionless solid boundary conditions,
  - moving solid boundary conditions,
  - symmetric (far field) boundary conditions,
  - outflow (far field) boundary conditions,
  - mixed (partly solid and partly outflow) conditions,
- at the bottom of wind-tunnel: moving (with inflow speed) solid boundary.

Finally, we shall add that the air velocity at inlet always was 20 m/s, and the medium properties have been defined as for the air. Control parameters for the solver and adaptive procedure were identical in all computations.

ABOUT CORRECTION OF DRAG COEFFICIENT

Aerodynamic resistance forces and coefficients obtained directly from wind-tunnel tests (or from their numerical simulations) require a correction cancelling the blockage effect. The vehicle model blocking the flow in wind-tunnel causes that the average air speed in the immediate vicinity of the model is increased. The blockage can be additionally enlarged due to a wake behind the vehicle. Thus, the directly measured or computed drag force and coefficient are usually overestimated. Moreover, one should remember that the blockage influences a longitudinal buoyancy effect and thereby increases the total error of measurements or computations.

Generally, the correction of the drag coefficient is defined as follows:

$$C_{DE} = w \cdot C_{DO}$$

where C_{DE} and C_{DO} are the effective and computational (or measured) drag coefficients, respectively. The correction factor w describes interdependence between both drag coefficients, and its simple, coarse description is usually based on the continuity equation.

Neglecting the fact that the flow is not uniformly speeded all along the vehicle model, and referring to the maximum cross-section of the vehicle, we can define the forms of correction factor as follow:

$$w = 1 - 2 \cdot (\frac{A_N}{A_T}) + (\frac{A_N}{A_T})^2 \qquad \text{(in 3D space)},$$

$$w = 1 - 2 \cdot (\frac{h_N}{h_T}) + (\frac{h_N}{h_T})^2 \qquad \text{(in 2D space)},$$

where A_N and A_T are frontal projection areas of the vehicle and wind-tunnel, and h_N and h_T – the heights of the body and tunnel, respectively. Other, simpler and more popular expressions for the correction factor are:

$$w = 1 - k \cdot (\frac{A_N}{A_T}) \qquad \text{or} \qquad w = 1 - k \cdot (\frac{h_N}{h_T}) \quad ,$$

where k is a constant parameter. In numerical simulations presented in further sections, we will also search for the parameters k_L and k_N in the following nonlinear formula for the correction factor:

$$w = 1 - k_L \cdot (\frac{h_N}{h_T}) + k_N \cdot (\frac{h_N}{h_T})^2$$

DIRECT RESULTS FROM SIMULATIONS

All results presented in this article refer to the numerical simulations in the two-dimensional space. The blockage effect is here much more visible than in the case of 3D simulations (with the same height of wind-tunnel). Moreover, the 2D computations are significantly more fast and less expensive (not require parallel machines), and thus the material for comparative studies is considerably more extensive. We must here add that the procedure for searching of correction factors is identical for the 2D and 3D simulations.

To illustrate the results from numerical flow simulations, we present first the exemplary final meshes of computational domains for different heights of wind-tunnels. The chosen meshes (Fig.3) refer to the computational domains with motionless solid (wall) boundary conditions at the top of wind-tunnels, and the tunnel heights 3 and 12 meters, only.
On account of the place limitation, we omit here the presentation of the pressure and horizontal components of velocity, and place only the instantaneous (from tenth second of each simulation) distribution of vertical component of velocity for three above-mentioned cases of discretization (Fig.4). This can help us to observe the differences in the forms of vortices in wakes behind vehicles according to the height of wind-tunnels.

COMPUTATIONAL AND EFFECTIVE DRAG COEFFICIENTS

Besides the comparison of direct results, i.e. distributions of variables in different computational domains and simulation moments, we have carried out also the comparisons of indirect quantities as the aerodynamic drag and lift forces or the drag and lift coefficients. These quantities characterise concisely all aerodynamic phenomena occurring around a vehicle. In further consideration we will busy oneself with drag coefficients only.

First of all we shall introduce a special symbolic description of particular simulations. We use the following pattern:

- first character denotes the rear end forms: h (hatchback), n (notchback) or s (squareback),
- second and third digits denote the height of wind-tunnel, e.g. 05 (5 meters),
- fourth character is always – (the dash),
- fifth digit denotes the type of boundary conditions at the top of wind-tunnel:
  - 2 the symmetric (far field) boundary conditions,
  - 4 the outflow (far field) boundary conditions,
  - 5 the moving (with inflow speed) solid boundary conditions,
  - 6 the motionless solid boundary conditions.

We have used also the mixed boundary conditions, e.g. 646 (motionless solid – outflow – motionless solid boundary conditions), but in this presentation they are omitted.

Different boundary conditions on the upper boundary of computational domains do not show essential influence on results, except the outflow boundary conditions which cause significant differences in drag coefficients for low (3, 4, 5 and 6 meters) wind-tunnels and no differences for high (12 and 25 meters) ones. It concerns all tested vehicle models. In this situation, in further presentation we will restrict oneself only to the motionless solid boundary conditions at the top of wind-tunnels.

The influence of the tunnel height on the obtained (computational) drag coefficients is substantial. It is distinctly visible in the case of all tested vehicle models. Exemplary, in Fig.5 has been presented the computational drag coefficients of the squareback model for tunnel heights: 3, 4, 5, 6, 12 and 25 meters. Large cyclic fluctuations of drag coefficients are here connected with large vortices in the wake behind vehicles. The vortices in low wind-tunnels are larger than those in high tunnels (see also Fig.4).

In order to eliminate the influence of the wind-tunnel height on drag values, we must correct the obtained computational drag coefficients and search for a correction factor ensuring that the drag coefficients will be covered to the highest degree. The classical, very crude blockage correction has only linear character, and the independent coefficient undergoing the changes is here the parameter k, which for the value $k = 1.55$ satisfied our criterion best (Fig.6). Next, for defining the effective drag coefficients with use of the fully nonlinear formula for the correction factor, we must previously determine the most suitable parameters k_L and k_N. Unfortunately, our experiences with the use of the mathematical optimisation procedures show that it is difficult to find univocally (the local extremes) these parameters. Exemplary, the effective drag coefficients determined with parameters $k_L = 2.0$ and $k_N = 1.0$ (Fig.7) are similar coefficients obtained with $k_L = k = 1.55$ and $k_N \cong 0.0$ (Fig.6), and so on. Moreover, for different examined models the optimum parameters are different, and in general, they are function of the vehicle geometry.

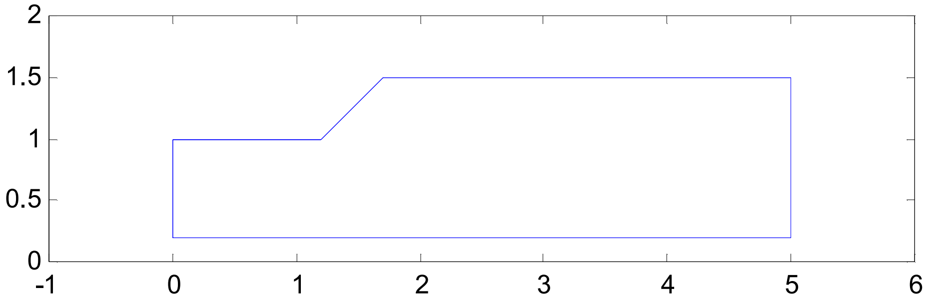

Fig.1. One of simplified vehicle models (the squareback)

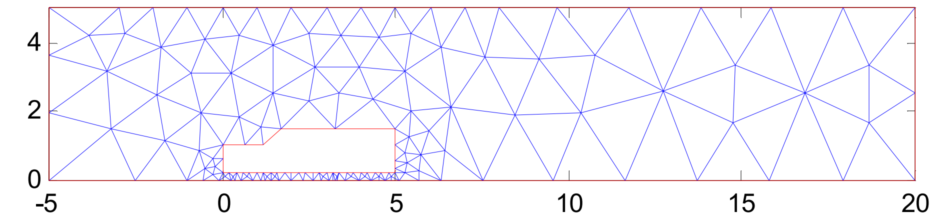

Fig.2. Initial discretization for the squareback model and the tunnel height 5 meters

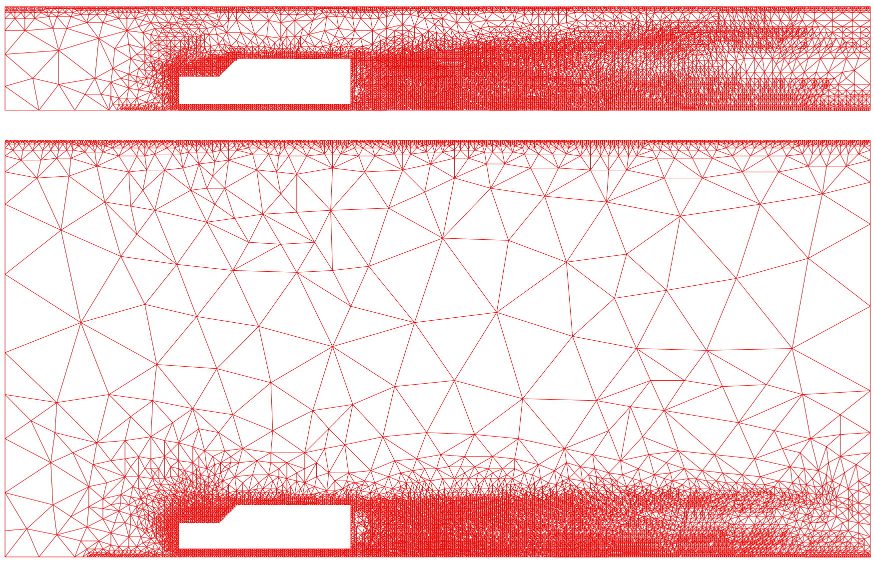
Fig.3. Exemplary final meshes for the tunnel heights: 3 and 12 meters

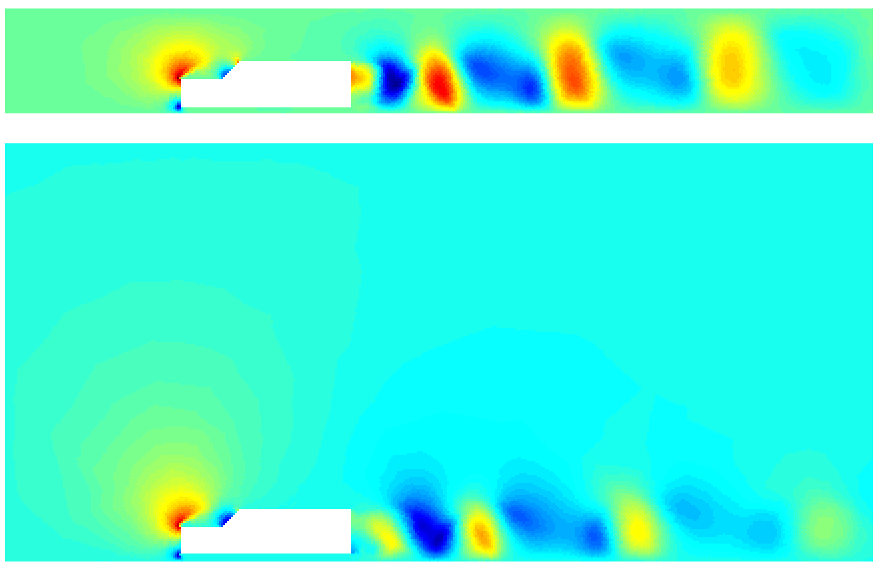
Fig.4. Distributions of vertical velocities for the tunnel heights: 3 and 12 meters.

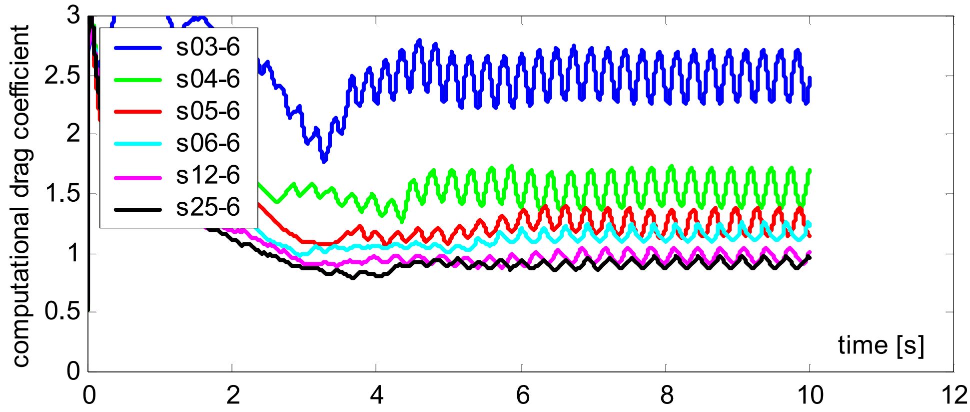

Fig.5. Computational drag coefficients for different tunnel heights

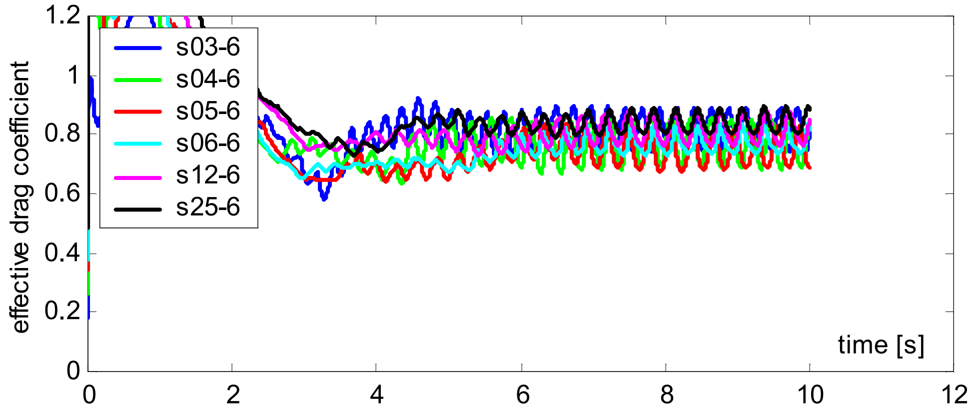

Fig.6. Effective drag coefficients ($k = 1.55$) – the linear correction

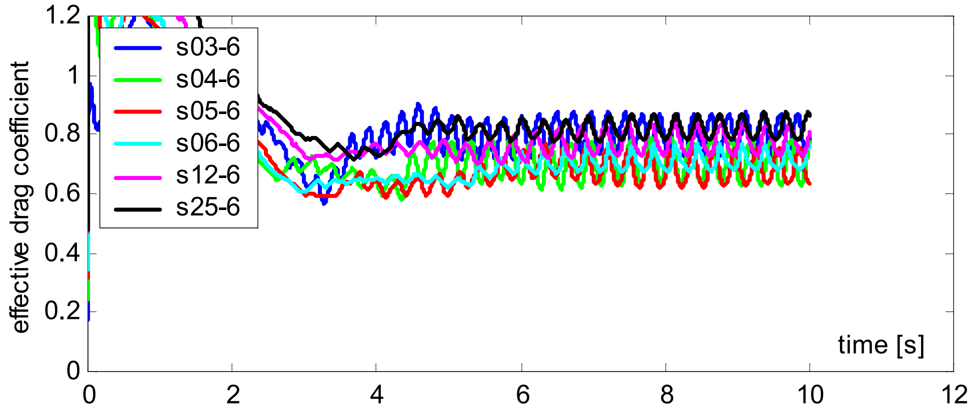

Fig.7. Effective drag coefficients ($k_L = 2.0$ and $k_N = 1.0$) – the nonlinear correction

CONCLUSION

In real wind-tunnel tests it is indispensable to carry out a correction of obtained results (aerodynamic drag forces and coefficients). To this end one should use a suitable nonlinear formula for correction and select properly the parameters. These parameters can be obtained from numerical simulations. However, one should also remember that the value of those parameters is also dependent on the shape of examined vehicle model.

In the case when we calculate the aerodynamic drag coefficient only by means of numerical simulations, it is possible to avoid the result correction through the assumption of sufficiently large proportion between the tunnel height and the car body height. In the case the two-dimensional analyses and the squareback model, the minimum height ratio was here about 20. Accordingly, for the simulations in the three-dimensional space, one may determine in like manner a limit for the ratio of the cross-sectional tunnel area to the projected frontal area of a vehicle model.

Finally one should also notice that applying (in computations) the adaptive procedures leads to increase of mesh density only in immediate vicinity of a vehicle model and within the range of its wake. Thus, considerable enlargement of the computational domain in numerical simulations does not cause the visible increase of computational cost.