POSMEC 2014 – Simpósio do Programa de Pós - Graduação em Engenharia Mecânica
Faculdade de Engenharia Mecânica – Universidade Federal de Uberlândia
26 a 28 de Novembro de 2014, Uberlândia - MG

INVESTIGATION OF CFD SETUP FOR AUTOMOTIVE APPLICATIONS

Renan Francisco Soares, renanfs@mestrado.ufu.br
Francisco José de Souza, fjsouza@mecanica.ufu.br
Universidade Federal de Uberlândia – Av. João Naves de Ávila 2121 - Campus Sta. Mônica | Uberlândia - MG - Brazil

Abstract. *An investigation of setup for CFD simulations of automotive applications is presented in this paper. As a test case, the DrivAer Fastback car model has been chosen. All simulations were performed in two dimensions. Thus, the profile in the symmetry plane DrivAer Fastback (with smooth underbody) car is adopted. Three of the most important factors are chosen for numerical simulation analysis, namely: (i) turbulence models, (ii) mesh base sizes and (iii) gradient methods. It shown that the drag prediction is very sensitive to mesh base size. As the surface base size is reduced, the simulation results for the different gradient methods tend to the same value. The conclusion is that the mesh base size has the major importance among the factors investigated. Regarding the turbulence model, SST k-ω turbulence model overpredict the drag in comparison to Realizable k-ε under larger mesh base size. On the other hand, under smaller mesh base size, the SST k-ω turbulence model underpredict the results with Realizable k-ε results. Such behaviour is very clear when using 2nd-order gradient methods. It leads to the conclusion that the first turbulence model is very sensitive to mesh base size, while the second shows smaller variations over different mesh base sizes. Regarding the gradient calculation method, they provide similar results under larger mesh base size. However, the 1st-order Green-Gauss has discrepant results from 2nd-order methods as the mesh base size decreases. In summary, the main contribution of this paper is the following best practices to set in CFD applications: (i) Realizable k-ε two-layer as turbulence model (robustness),(ii) smaller mesh base size over the car surface (better prediction of drag), and (iii) Green-Gauss or Hybrid LSQ in 2nd-order for gradient calculation method (numerical accuracy).*

Keywords*: Aerodynamics, Automotive, DrivAer, CFD, RANS.*

1. INTRODUCTION

The formation of vortex structures, as well the separation of boundary layer and respective turbulent wake, gain importance in applications in aeronautical and motorsport, as a result of the its influence with lift and drag forces.

In the automotive industry, the size of the turbulent wake is related to fuel consumption. Once the flow detaches behind over the rear of the vehicle, it increases the intensity of turbulent wake. It results in an increase of the drag force over the car, although it has non-linear relations to turbulent wakes (Hucho, 1998). An alternative method to evaluate the drag over a car is employing numerical investigation via computational fluid dynamics (CFD).

Among the many parameters for numerical setup, the knowledge of how such features affect the solution process is crucial to achieve results with satisfactory accuracy. For instance, to have a suitable mesh density for a specific problem with suitable CPU time and numerical precision has remained a main theme in the CFD literature. The flow pattern around a vehicle is very complex. To be able to resolve the wakes behind the vehicle, a fine mesh is required. However, to reduce the computation time and memory, a strategy has to be established to refine the mesh only in particular areas within the flow domain (Ahmad *et al.* 2010). Besides the mesh characteristics, there are other significant parameters as turbulence models, gradient methods and solver algorithm.

Therefore, the motivation of this paper is to determine an ideal numerical setup to require a moderate level of computational resources.

2. BACKGROUND

Among several two-equation turbulence models, the k-ε and k-ω turbulence model families are often used in numerical studies for industrial applications, like automotive aerodynamics. A comparison of two-equation turbulence model (Browand *et al.*, 2009) shows that few models have reasonable accuracy for drag prediction: SST k-ω (0.8%), High-Reynolds Number k-ε (1%), and RNG k-ε (2.3%), comparing to experimental result. Nonetheless, according to Olander *et al.* (2011), the realizable k-epsilon turbulence model is used in the standard procedure at Volvo cars throughout their computational aerodynamics development.

From the computational modelling point of view, gradients are used in numerous places within the transport equation solution methodology. For instance, it is applied in reconstruction field values at the cell faces, secondary gradients for diffusion terms, pressure gradients for pressure-velocity coupling in the segregated flow model, and strain-rate and rotation-rate calculations for turbulence models (CD-adapco, 2014).

In regards to computational mesh sensibility, Ahmad *et al.* (2010) presents mesh parameters analysis for the simplified MIRA car model. According to them, as the car surface (base) mesh size becomes smaller, the drag coefficient tends to reduce and approaches the experimental value, but this followed by an increase in the total number of cells as well as the simulation run time and the computer memory.

3. NUMERICAL SETUP

This section aimed to identify the optimum combination among three numerical parameters, as shown in Tab. (1).

Table 1. Numerical setup factors and levels.

Factors	*Number of Levels*	*Description*
Mesh base size	4	40mm; 20mm; 10mm; 05mm
Turbulence model	2	SST k-ω; Realizable k-ε two-layer
Gradient method	3	Green-Gauss (1^{st}- and 2^{nd}-order); Hybrid Gauss - LSQ (2^{nd}-order)

The DrivAer model has been chosen in this investigation of the numerical setup. Due to time constraints, the computational domain of all simulations were two-dimensional. The symmetry section of DrivAer Fastback model (Heft *et al.*, 2012a,b) of 1:1 scale has adopted for studies. The inlet of rectangular domain is positioned 2L upstream of the car profile, the ground is 12L long, and is 0.065L from the front wheel axis position.

As demonstrated in previous studies of Soares *et al.* (2013), the pressure contours shown that typical height of three-dimensional domain might compromise the two-dimensional results, since the pressure is not constant at the top boundary, as seen at Fig. (1). In the 3D domain, such height (and width at same size) provide a blockage ratio of approximately 1%. Hence, to minimize an equivalent 2D blockage ratio[1], a height of 4.4L seems a suitable choice. An example of final mesh is shown at Fig. (2).

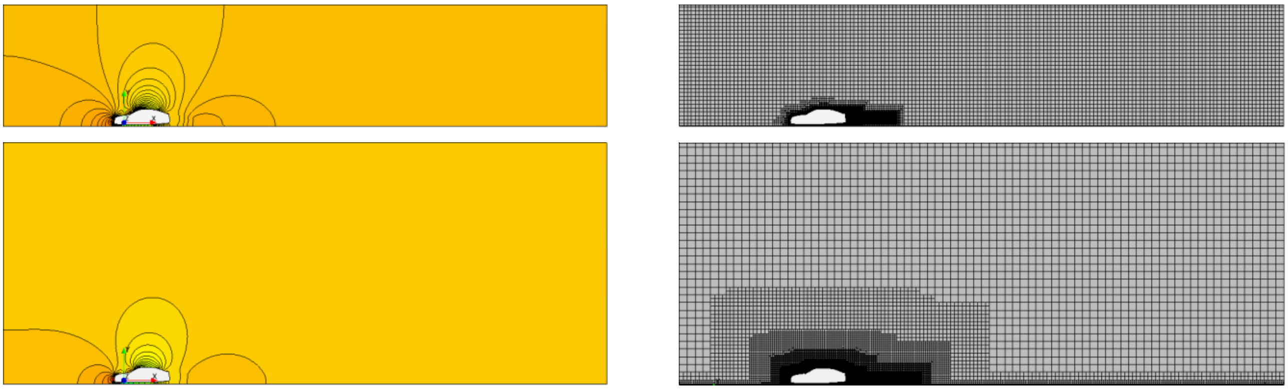

Figure 1. Comparison of pressure contours between the domain of 2.2L (upper) and of 4.4L (lower) heights.

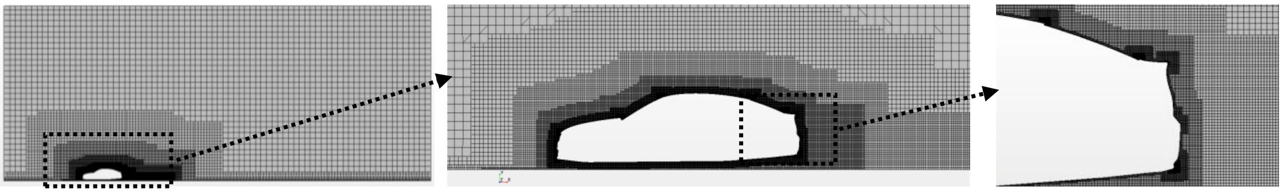

Figure 2. Example of numerical mesh over DrivAer Fastback profile model, under mesh base size of 5 mm and refinement regions.

Once the domain dimensions have been selected properly, four levels of mesh base size over the car are generated. All meshes display the same wake refinement region and prism layer properties (providing y^+ between 30 and 90).

Wall functions are employed to represent the near-wall effects. They allow a reduction of mesh cell size and, as consequence, a proportional reduction in the computational cost. Eq. 4 shows the dimensionless y^+. In general, wall function are valid when y^+ is between 30 and 500. In other words, within the range of log-law layer (Olander *et al.*, 2011). The law of the wall is presented in Eq. 2, as following:

[1] Widely used in wind tunnel tests, the blockage ratio is calculated as $\varphi = A_{ref}/(H.W)$. In two-dimensional cases, there is no such width of the domain, and the reference frontal area (A_{ref}) must be simplified to the reference height (h_{ref}). Therefore, the two-dimensional blockage ratio may by written as $\varphi = h_{ref}/H$.

$$y^+ = \frac{\rho u_t y}{\mu} \quad (1)$$

$$u^+ = \frac{\bar{u}_p}{\bar{u}_\tau} = \frac{1}{\kappa}\log y^+ + 5.1 \quad (2)$$

The drag coefficient (C_D) is given by:

$$C_D = \frac{F_D}{\frac{1}{2}\rho A_{ref} U^2} \quad (3)$$

Steady-state simulations are carried out using the STAR-CCM+ code employing two RANS turbulence models: Realizable k-ε two-layer model, and SST k-ω model. Each model have been simulated with the three different gradient methods mentioned in Tab. (1). Since there is no experimental data for this 2D profile, there is not any specific Reynolds number to be set. Hence, the free stream of U_∞=40 m/s is chosen to better fit a real car operation. The air density (ρ) is 1.18415 kg/m³ and its dynamic viscosity (μ) is 1.85508x10^{-5} N.s/m². In summary, the physical model setup is shown in Tab. (2) and the boundary conditions are presented at Tab. (3).

Table 2. Physical model setup.

Domain	***Time***	***Fluid***	***Flow solver***	***Equation of State***	***Flow Regime***	***Turbulence model***
Two-dimensional	Steady-state	Air	Segregated	Constant density	Turbulent	SST k-ω; Realizable k-ε two-layer

Table 3. Boundary conditions.

Boundary	***Condition***
DrivAer car profile	wall: no slip
Inlet	velocity-inlet: U_∞ = 40 m/s, I = 0,1%, μ_t/μ = 10
Outlet	pressure-outlet
Top	symmetry
Ground	wall: no-slip, ground movement of U_w = 40 m/s

Once the physical model and boundary conditions are chosen, the simulations are performed. Regarding the four mesh base sizes, two turbulence models, and three gradient methods, the total of twenty-four parameters combinations are created. Figure (3) shows the results for the drag coefficient obtained with the Realizable k-ε and SST k-ω turbulence models.

Figure 3. Comparison of drag coefficient of all mesh base size and gradient methods, by Realizable k-ε two-layer (left) and SST k-ω (right) turbulence models.

As expected, as the mesh base size is reduced, the simulation results for the different gradient methods tend to the same value. However, it is noticed that the SST k-ω shows a higher variance and the 1st-order Green-Gauss also did not follow the results using 2nd-order gradient methods, known as more accurate.

The optimum numerical setup should be close to the finest mesh base size, while both 2nd-order Green-Gauss and Hybrid LSQ gradient methods predict drag almost at same level, as shown above. Thus, one additional simulation has

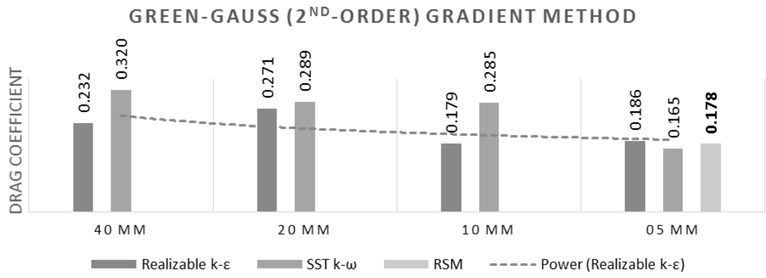

Figure 4. Drag coefficient prediction, by 2nd-order Green-Gauss gradient method, over different mesh base sizes and turbulence models.

been performed with such combination, with the Reynolds Stress (RSM) turbulence model. Figure (4) compares the C_D prediction with the 2nd-order Green-Gauss gradient with the different mesh base sizes and turbulence models.

Notice that the RSM model provides C_D prediction between the results with the Realizable k-ε and SST k-ω turbulence models, as displayed at Fig. (4) above. Although RSM turbulence model generally suffer from numerical stability issues, it provides more accurate prediction than two-equation models. Hence, such convergence among the prediction by these turbulence models indicates that the numerical setup is optimised enough to deal with reasonable prediction.

Another important point is the stability behaviour of drag prediction using Realizable k-ε two-layer. While SST k-ω match the other models only with the finest mesh, the k-ε based-model achieved the similar results with slightly coarse mesh. This feature allows a saving computational time and/or resources.

4. CONCLUSION

An investigation of setup for CFD simulations of automotive applications is presented in this paper. Turbulence models, mesh base sizes and gradient methods have been compared in two-dimensional domain, in a total of twenty-four parameters combinations.

It shown that the drag prediction is very sensitive to mesh base size. As the surface base size decreases, the drag prediction tend to decrease the variance prediction. The conclusion is that mesh base size has the major importance among the factors investigated.

In the turbulence model perspective, SST k-ω turbulence model overpredicts the drag in comparison to Realizable k-ε under larger mesh base size. On the other hand, under smaller mesh base size, the SST k-ω turbulence model underpredicts the results with Realizable k-ε results. Such behaviour is very clear when using 2nd-order gradient methods. It leads to the conclusion that the first turbulence model is very sensitive to mesh base size, while the second shows smaller variations over different mesh base sizes.

Regarding the gradient calculation method, they provide similar results under larger mesh base size. However, the 1st-order Green-Gauss has discrepant results from 2nd-order methods as the mesh base size decreases.

As the main contribution of this paper, the following best practices to set in CFD setups is suggested: (i) Realizable k-ε two-layer as turbulence model (robustness),(ii) smaller mesh base size over the car surface (better prediction of drag), and (iii) either Green-Gauss or Hybrid LSQ in 2nd-order for gradient calculation methods (numerical accuracy).

5. REFERENCES

AHMAD, N. E. *et al.* 2010. Mesh Optimization for Ground Vehicle Aerodynamics. CFD Letters, Vol.2.

BROWAND *et al.* 2009. The Aerodynamics of Heavy Vehicles II: Trucks, Buses, and Trains. Lecture Notes in Applied and Computational Mechanics 41. Springer, Heidelberg.

CD-ADAPCO. 2014. User Guide STAR-CCM+ Version 9.04.

HEFT, A. I. *et al.* 2012b. Experimental and Numerical Investigation of The DrivAer Model. Rio Grande, Puerto Rico: ASME 2012 Fluids Engineering Summer Meeting.

HUCHO, W-H. 1998. Aerodynamics of Road Vehicles. Fouth Edition. Society of Automotive Engineers Inc.

OLANDER, M. *et al.* 2011. CFD Simulation of the Volvo Cars Slotted Walls Wind Tunnel. 53f. Master Thesis, Chalmers University of Technology.

SOARES, R. F. *et al.* 2013. Estudo Fluidodinâmico Computacional (CFD) Aplicado à Aerodinâmica do Esporte Automobilístico. 149 f. Trabalho de Conclusão de Curso (Bacharel em Engenharia Mecânica), Universidade Tecnológica Federal do Paraná - UTFPR.

6. RESPONSIBILITY FOR THE INFORMATION

The authors are responsible for information included in this paper.