Validation of CFD Modeling and Simulation of a Simplified Automotive Model

S.Mansor[1, a], N.A.R. Nik Mohd[2, b], C.W.Chung[3, c]

[1,2,3] Department of Aeronautical Engineering, Universiti Teknologi Malaysia,

81310 Skudai, Johor, Malaysia.

[a]shuhaimi@mail.fkm.utm.my, [b]ridhwan@fkm.utm.my, [c]wcchan2@live.utm.my

Keywords: CFD guidelines, Davis model, Vehicle aerodynamics

Abstract. In the early design phase of automotive sector, the flow field around the vehicle is important in decision making on design changes. It would consume a lot of money and time for multiple prototypes development if adopt traditional testing method which is wind tunnel test. Thus, numerical method such as Computational Fluid Dynamics (CFD) simulation plays an important role here. It is very often simulation results been compared with wind tunnel data. However, with various mesh types, meshing methodology, discretization methods and different solver control options in CFD simulation, users may feel low confidence level with the generated simulation results. Thus, a robust modeling and simulation guideline which would help in accurate prediction should be developed due to the industry's demand for accuracy when comparing CFD to wind tunnel results within short turnaround time. In this paper, a CFD modeling and simulation study was conducted on a simplified automotive model to validate with wind tunnel test results. The wind tunnel environment was reproduced in the simulation setup to include same boundary conditions. Meshing guidelines, turbulence model comparisons and also the best practice for solver setup with respect to accuracy will be presented. Overall, CFD modeling and simulation methods applied in this paper are able to validate the results from experiment accurately within small yaw ranges.

Introduction

In recent years, the influence of global issues includes environmental pollution and oil crisis has grown rapidly and has raise awareness among the public about the importance of energy conservation. For car manufacturers, their major concern is the fuel consumption. Past studies on car aerodynamics discovered that vortices around the road vehicle strongly influence the drag, lift, side force and yaw moment. The aerodynamic drag is the main contributor of the high fuel consumption of vehicles.

Nowadays, numerical simulations, wind tunnel and road tests are jointly used for the aerodynamic study in the automotive industry. Wind tunnel experiments are known to be expensive and preparation time is high due to test model build up. In contrast to wind tunnel testing, numerical method such as Computational Fluid Dynamics (CFD) require lesser running costs and useful in understanding the flow [1]. Thus it becomes important to use simulation in product development stage. These days, CFD simulation is not just focus on accuracy but also on shorter turnaround time and reliability with repeated design modifications. Reynolds Averaged Navier Stokes Equation (RANS) approach has been maturely developed in recent years. It enables a fast and cost-effective estimation of aerodynamics for ground vehicles compare to other more expensive approach such as Detached Eddy Simulation (DES) and Large Eddy Simulation (LES) used in other researches [2,3]. Thus, RANS is chosen as the turbulence modeling approach used in the simulations.

In this paper, thorough comparison and guidelines in terms of CFD modeling strategy for several turbulence models, and solver settings are presented on a simplified road vehicle model called Davis model. All the simulation results have been correlated with available wind tunnel data.

Experiment Approach

The two Davis model configurations used in this paper have been previously used by Joshua [4] and Mansor [5] which allowing for direct comparisons of the simulation results presented in this paper with their previously published experimental works. Both works were conducted in the Loughborough University 1/4 scale wind tunnel. The wind tunnel has a closed working section, 1.9 m wide by 1.3 m high and can deliver maximum wind speed up to 45 m/s. The test model used in both the wind tunnel tests was originally developed by Davis [6] which constructed from fibreglass and the model dimensions are shown in Fig. 1. The steady test is conducted in the yaw range of +/-20 degrees.

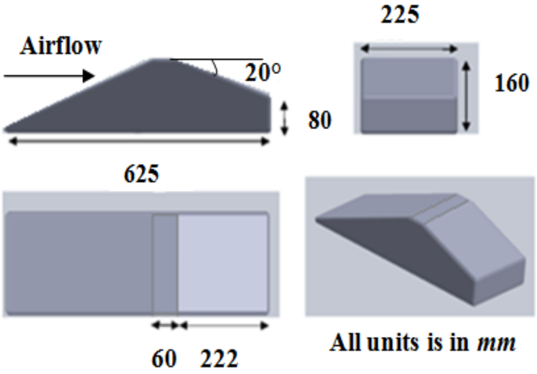

Fig. 1. Davis model geometry.

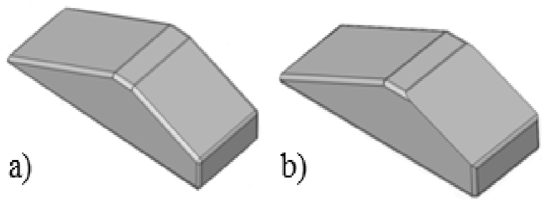

Fig. 2. Davis model configurations: a) round edge model and b) square edge model.

In order to investigate the ability of the CFD modeling method to predict the changes of model's geometry, two configurations of the same model with different rear pillar geometries are used in this study which includes round edge rear pillars and square edge rear pillars. The round edge model has all the edge with 10 mm radius. The square model is identical to the round edge model except the rear pillar edges modified to square edge as shown in Fig. 2. All the tests are conducted at 40 m/s with Reynolds number of 1.7×10^6 based on the model length. As the ratio of model frontal area to tunnel test section area (tunnel blockage) is 1.2% which is less than 5 %, no corrections is required for the wind tunnel test data in this study because the blockage effect is very small [7].

CFD Simulation Approach

The external aerodynamics simulation process consists of four important steps which include model geometry setup, mesh generation, numerical iterations and postprocessing.

Mesh Generations. Hybrid mesh generation are chosen in this study. This is because prism layer in hybrid mesh can resolves the boundary layer efficiently. Other than prism layers, hybrid mesh expands the rest of the computational domain with tetrahedral mesh. The effectiveness of prism layer in obtains a more accurate result also demonstrated in previous research [8].

Patch independent tetrahedral mesh method is used in the meshing. The Patch Independent mesh method for tetrahedrons ensures mesh refinement where necessary, but also maintains larger elements where possible which allowing for quicker computation [9]. There are two refinement zones created by using box shape in the mesh to help smooth transition of mesh from the refinement zone to the domain of wind tunnel. The total number of element after the refinement is around 4.48 million with maximum skewness less than 1.0 which is approximately 0.88.

Further refinement is applied near to the wall of the model. The size of the grid cell nearest to the surface (value of $y+$) is very important and the $y+$ value depend on the modeling approach chosen.

In this study, k-ω SST Model was chosen to simulate the turbulence condition and a near wall grid resolution of at least $y+=1$ are required [10]. In practice, maintaining a prescribed value of y+ in wall-adjacent cells throughout the domain for is very challenging and Enhanced Wall Treatment which is default to be used for k-ω model is very helpful because it makes the model relatively insensitive to the $y+$ value of the wall cell [9]. Moreover, in the case of a bluff-body flow, such as the flow around a car with massive separations and regions of recirculating flow, the influence of the near-wall structures on the main flow is smaller [11]. So that, the requirement of the critical small $y+$ can be ignored and the accuracy of the simulation did not affected even $y+$ more than 1 had been used. In this paper, near wall meshes with y+ of 15 are generated. Grid generation has a strong impact on model accuracy. In order to generate high quality CFD grids, the most important one is that the relevant shear layers should be covered by at least 10 or more cells normal to the boundary layer. Thus, 10 layer of prism cell are applied near the walls of Davis model in the simulation. First aspect ratio inflation method was applied in the meshing.

Mesh Independence Study. Mesh Independence Study is conducted to ensure that the solution obtained is independent of the mesh resolution. This study is carried out for round edge model with various mesh sizes. The results show that there is only 0.5% of differences in drag coefficient with mesh size increment after 4.48 millions elements mesh size. Thus 4.48 millions elements mesh size was chosen in the CFD analysis and similar mesh scheme was adopted for square edge model.

Comparisons of Turbulence Models. Most common turbulence models including k-ε standard, k-ε realizable, k-ω standard and k-ω SST are tested in this study. Simulation generated centerline pressure distribution over the top surface of the round-edged model had been compared with data collected using Electronic Pressure Scanners as in reference [4]. Results from k-ε standard and k-ε realizable turbulence model correlated poorly against the experimental results compare to the two k-ω models and it was observed that k-ω SST model provided the closest agreement in pressure coefficient values against the experimental test data as in Fig. 3. Hence, k-ω SST turbulence model has been chosen for all CFD simulation in this paper.

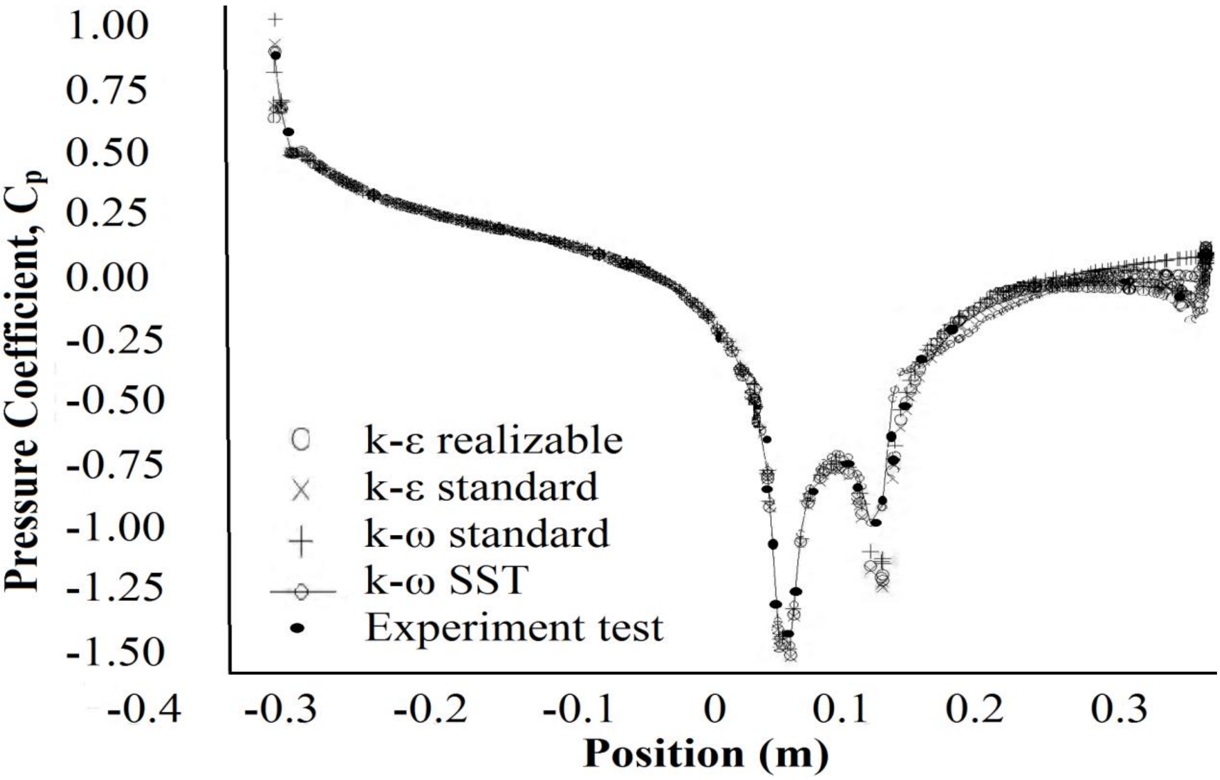

Fig. 3. Pressure coefficient, C_p plot of different turbulence models.

Boundary Conditions and Solver Setup. CFD modeling of the wind tunnel test section with Davis model is shown in Fig. 4. The set-up of the boundary conditions of numerical simulation matches with the conditions of the experiment. The inlet of the domain is defined in Fluent as velocity-inlet with the same velocity of wind and turbulence intensity from the experimental wind tunnel and the outlet of the domain is set as pressure-outlet. The domain is set at an upstream distance of 1.7 m from the inlet to ensure that the flow to be fully developed and ends at about 1.9 m downstream to the outlet to capture the separation and wake generation. No-slip wall condition is used for the two walls in the analysis which include Davis model and the floor. The other walls of the domain are modeled as symmetry.

First order upwind convection scheme was used in FLUENT during the start of calculation. After achieving convergence with the first order scheme, the calculation was continued using second-order upwind scheme. The second-order scheme reconstructs the face pressure in the manner used for second-order accurate convection terms. This scheme may provide some improvement over the standard and linear schemes, but it may cause some errors if it is used at the beginning of a calculation with a bad mesh [12]. There are options to solve the flow problem in either a segregated or coupled manner in Pressure-based solver. Coupled approach provides some advantages over the segregated approach which the coupled scheme obtains a robust and efficient single phase implementation for steady-state flows with good performance [12]. So that, coupled scheme option had been chosen in this study.

For pressure-based coupled case, Courant number is set to 200 and the Explicit Relaxation Factors for Momentum and Pressure is set to 0.75 by default. However, when higher order schemes for momentum and pressure are used such as in this study, the explicit relaxations need to be reduced. In the simulation, explicit relaxation of 0.25 had been used based on recommendation from Fluent user's guide which stated that the calculations can be stabilized by further reduction of the explicit relaxation factor to 0.25 for cases with much skewed meshes. For other under-relaxation factor, the default values may be too aggressive especially at the beginning of the calculation. So that, 0.8 was set in turbulent viscosity for First Order Upwind and 0.95 for Second Order Upwind to facilitate convergence. Detailed settings are shown in the Table II, III and IV.

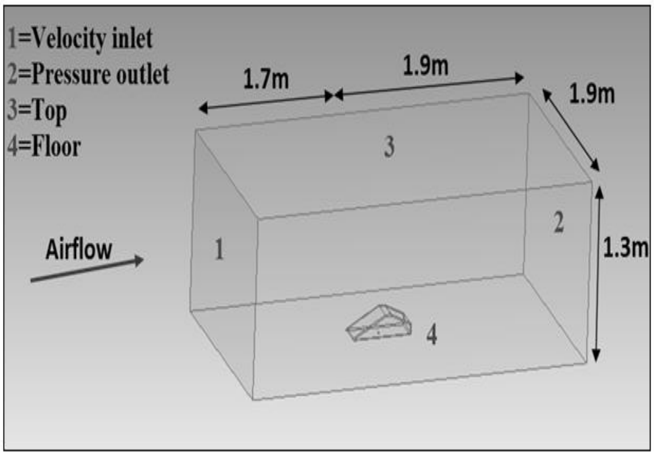

Fig. 4. CFD domain setup.

Table II. References values settings.

Reference Values	Settings
Area (m^2)	0.04 at zero yaw angle (Front model area facing the inlet airflow)
Density (kg/m^3)	1.225 (At sea level and at 15°C)
Length (m)	0.625 (Reference length of test model)
Velocity (m/s)	40 (Velocity of airflow to be applied)
Turbulent Intensity (%)	0.15 (Same value with reference wind tunnel)

Table III. Solution methods settings.

Pressure-Velocity Coupling	Settings
Scheme	Coupled
Gradient	Least Squares Cell Based (Default)
Pressure	Standard
Momentum	First and Second Order Upwind
Turbulent Kinetic Energy	First and Second Order Upwind
Specific Dissipation Rate	First and Second Order Upwind

Table IV. Solution control settings.

Solution Controls	Values
Flow Courant Number	50
Explicit Relaxation Factors	**Values**
Momentum	0.25
Pressure	0.25
Under-Relaxation Factors	**Values**
Density	1 (Default)
Body Forces	1 (Default)
Turbulent Kinetic Energy	0.8 (Default)
Specific Dissipation Rate	0.8 (Default)
Turbulent Viscosity	0.8 for First Order Upwind, 0.95 for Second Order Upwind

Results and Discussions

Forces and Moments Data. Drag coefficient, C_d comparison of round edge model shown in Fig. 5 and the differences of C_d summarised in Table V. Simulation C_d correlates very well with experimental value at zero yaw angle which only generates difference of 2.98%. However, small discrepancy exists along increasing yaw angle between all the simulation results and experiment results with an average of 12% differences. It would be interesting to compare this result with the square edge model but unfortunately experimental result for this configuration is unavailable.

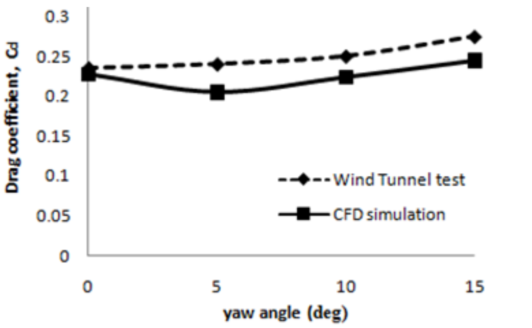

Fig. 5. Drag coefficient, C_d comparison of round edge model.

Table V. C_d Difference (%) between experimental and simulations data.

Yaw angle (°)	C_D (Wind Tunnel Test)	C_D (CFD Simulation)	C_D difference (%)
0	0.235	0.228	2.98
5	0.240	0.205	14.58
10	0.250	0.224	10.4
15	0.275	0.245	10.9

The validations of side force and yaw moment between CFD simulations and experimental results are illustrated in Fig. 6 and Fig. 7. These results indicate that yaw angle sensitivity test on Davis model is generally validated in small yaw angle range. Simulation results start deviate from the experiment data at large yaw angle beyond 10°. This shown that CFD modeling and simulating method used in this study is very successful in predicting flow around ground vehicles where flow is attached. Forces and moments cannot be predicted accurately at large yaw angle due to limitation of RANS-based turbulence model to capture wide range of scale of the complex turbulent flow which also illustrated as in [13]. Results based on Fig. 6 and Fig. 7 also demonstrate that CFD simulations could offer adequate accuracy for investigations of geometry changes such as rear pillar modifications in this study.

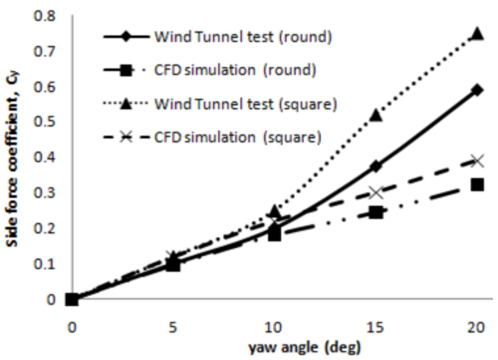

Fig. 6. Side force coefficient, C_y comparison of round edge model and square edge model.

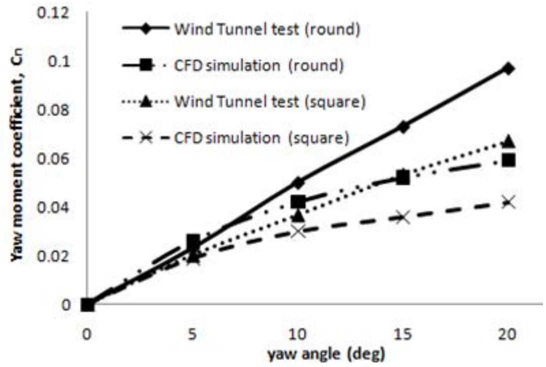

Fig. 7. Yaw moment coefficient, C_n comparison of round edge model and square edge model.

Flow Field Structure. In CFD Simulation, flow's behavior can be analyzed by identify the vortex structures with the help of plotting Isosurfaces of the second invariant of the velocity gradient, so called “Q-criterion” as shown in Fig. 9. The round rear pillars allow acceleration of the flow from the sides towards the model centreline. The relatively weak and low trailing vortices mix with the wake behind the model base, creating trailing vortices that are move inboard to the centreline upon exiting the backlight. The changes to the flow field are significant for a relatively small modification of rear pillar geometry. The trailing vortices generated on the square edge model do not mix with the wake behind the base and the vortices extend less inboard toward the centreline of the model compare to the round edge model. All the significant trends of the flow structure on the experimental results are well predicted by CFD Simulation. Thus, CFD simulation generated flow structure has been validated with the experimental results.

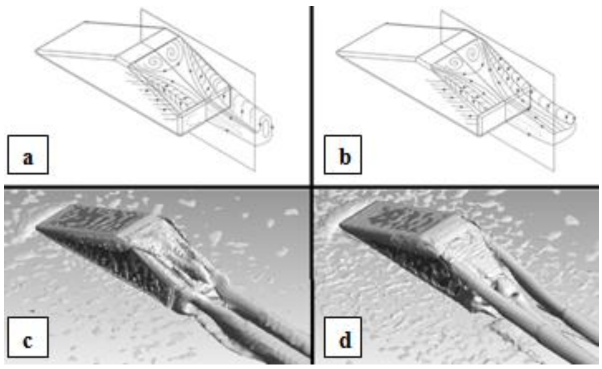

Fig. 8. a)&b): Sketch of the flow field of the round edge and square edge model based on experiment results; c)&d): Isosurfaces of the instantaneous second invariant of the velocity gradient (Q=3500 s−2) of round edge and square edge model.

Summary

CFD modeling and simulation strategy were proposed in this paper to serve as a practice guideline to simulate external aerodynamics of simplified automotive model accurately within short time frame. In particular within small yaw angle range, the RANS-based CFD modeling and simulation methods applied in this study are able to predict surface flow visualisations and pressure distributions accurately, estimate forces and moments correctly even having geometry modifications (round edge and square edge rear pillar), and also simulate vortex structure and vortices structure behind the simplified automotive model realistically. Obvious deviations on results occur when yaw angle simulation more than the range specified due to complicated flow field. All the CFD Simulation results have been well validated with the experimental results.

References

[1] Takashi Takiguchi, Kenta Ogawa, Hiroyuki Tateyama and Tatsuya Oda, The Automatic Aerodynamic CFD Framework Employing Vehicle Specifications at the Concept Stage of Development, SAE Technical Paper 2013-01-0604, 2013.

[2] Sinisa Krajnovic, Exploration and Improvement of Road Vehicle Aerodynamics using LES, SAE Technical Paper, 2011-01-0176, 2011.

[3] Natalia Castro, Omar D. Lopez and Luis Munoz, Computational Prediction of a Vehicle Aerodynamics Using Detached Eddy Simulation, SAE Technical Paper, 2013-01-1254, 2013.

[4] Joshua Thomas Baden Fuller, The Unsteady Aerodynamics of Static and Oscillating Simple Automotive Bodies, Doctoral Thesis, Loughborough University, August 2012.

[5] Shuhaimi Mansor, Estimation of Bluff Body Transient Aerodynamic Loads Using an Oscillating Model, Doctoral Thesis, Loughborough University, April 2006.

[6] John. P. Davis, Wind Tunnel Investigation of Road Vehicle Wakes, Doctoral Thesis, London University, 1982.

[7] William H.Rae, and Alan Pope, Low Speed Wind Tunnel Testing, Second ed, John-Wiley & Sons, 1984, pp.507.

[8] Nurul M. Murad and Jamal Naser , Firoz Alam and Simon Watkins ,Simulation of Vehicle A-Pillar Aerodynamics using Various Turbulence Models, SAE Technical Paper 2004-01-0231, 2004.

[9] ANSYS Meshing User’s Guide, Ansys. Inc, November 2013.

[10] Introduction to ANSYS Fluent, Chapter 6: Turbulence Modeling, Ansys. Inc, 2010.

[11] Sinisa Krajnovic, Exploration and Improvement of Road Vehicle Aerodynamics using LES, 2011-01-0176, 2011.

[12] ANSYS FLUENT User’s Guide, Ansys. Inc, November 2011.

[13] Guilmineau, E. and Chometon, F., Effect of Side Wind On A Simplified Car Model: Experimental and Numerical Analysis, Journal of Fluid Engineering, February 2009, Vol.131/ 021104-1.