ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

WIND TUNNEL EXPERIMENT AND FLOW PREDICTIONS FOR WEDGE AND VEHICLE MODELS AT LOW REYNOLDS NUMBER

Arif S. M. Sohaimi, M. S. Risby, Saiddi A. F. M. Ishak, Khalis S., N. Hafizi and A. B. M. Azhar
Protection and Survivability Research Unit, Faculty of Engineering, Universiti Pertahanan Nasional Malaysia , Kem Sg. Besi, Kuala Lumpur, Malaysia
E-Mail: risby@upnm.edu.my

ABSTRACT

Latest vehicle development demands a solid background in aerodynamics to reduce the unsteady flow. The complexity implied in the vehicle conception especially due to the shaped and accessories that form its geometry creates the validation projects costly. The low speed flow field for a wedge-shaped and armoured vehicle model is investigated both experimentally and numerically. The wedge-shaped dimension is 100 mm X 100 mm X 49 mm which 28° wedge angle. In order to measure the velocity and flow characteristics, a wind tunnel study was performed with wind speed 5.18 m/s and validated using Computational Fluid Dynamics (CFD) simulation software ANSYS FLUENT. Details about the flow field, including velocity and flow pattern, are shown. Further CFD simulation of an armored vehicle model using the similar approaches was performed at speed of 22.2 m/s to predict the flow pattern.

Keywords: boundary condition, ANSYS, turbulence, laminar.

INTRODUCTION

The development of aerodynamic concentrated on a vehicles using wind tunnel tests and Computational Fluid Dynamics (CFD) commands a good comparison between results and generated with both methods. CFD simulations ignored the influence of wind tunnel disturbance consequences, the comparability of the interference-free CFD results and experimental data will not perfect. In order to cut lead time, lower experimental work and lower the costs affiliated with vehicle conception work, aerodynamics specialists are constantly attempting fresh ideas and solutions capable of providing a fast and precise solution to the design targets. One way to obtain this goal is combining numerical simulation with experimental measurements in wind tunnel tests and key points in the development of computational codes is its validation with experimental results (Franck and El 2004). A study focused on wind tunnel tests is to determine the aerodynamic impact of various fuel saving devices used in a semi-trailer truck (Chowdhury *et al.*, 2013) and to determine the force and moment coefficients over a range of yaw angles and to investigate the flow mechanisms responsible for these forces that focused on truck (Coleman & Baker, 1990). While some researchers used CFD to solve aerodynamic problems. Mining dump trucks been developed to analyzing the velocity and pressure field with different shape of truck head (Wei, Wang, & Feng, 2008) but (Qi, Liu, & Du, 2011) conducted and developed both methods which is CFD and wind tunnel to studies the truck-induced airflow. However, no study been found that focused on armoured vehicle which is attempt to see the aerodynamic effect on land and its movement in water (for amphibious type).

The present work is based on analyzing the flow characteristics and velocity of the external flow field around an armoured vehicle. The work will start validating a simple wedge model subjected to a low velocity flow using a wind tunnel experimentation. A similar CFD model using similar boundary condition is developed and validated with the experimental results. Once the model is validated, the armored vehicle model is used and simulated at 22.2 m/s (equivalent to an armored vehicle top speed movement on land which is at 80 km/h).

EXPERIMENTAL METHOD

All of tests reported here were carried out in the 1 m X 10 m X 1.8 m National Defense University of Malaysia. The wind tunnel is an open-loop suction type wind tunnel structure, the outlet has to connect to outdoor environment. The maximum speed of the tunnel is approximately 100 m/s and the minimum speed of 0.5 m/sec can only be reached under a windless outdoor condition. The wind tunnel test section dimension is 300 mm X 300 mm X 1000 mm, 4 sides can be disassembled. The experimental wedge-shaped model dimension is 100 mm X 100 mm X 49 mm, 28° was connected through a mounting strut (Figure-1). The pitot static tube was used to measure hydrostatic pressure, which is need to convert into velocity (Equation. 1, 2 and 3).

$$P_0 = P_1 + \frac{1}{2}\rho v1^2 \tag{1}$$

$$V_1 = \sqrt{\frac{2(P_0 + P_1)}{\rho}} \tag{2}$$

$$P = h\rho g \tag{3}$$

Each data was recorded with 3 different position of pitot static tube in streamline and 3 different position in span wave (Figure-2). Tests were conducted at a frequency 7.23 Hz (equivalent to 5.18 m/s). Multiple data sets were collected and the results were averaged for minimizing the further possible errors in the raw experimental data.

VOL. 11, NO. 10, MAY 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

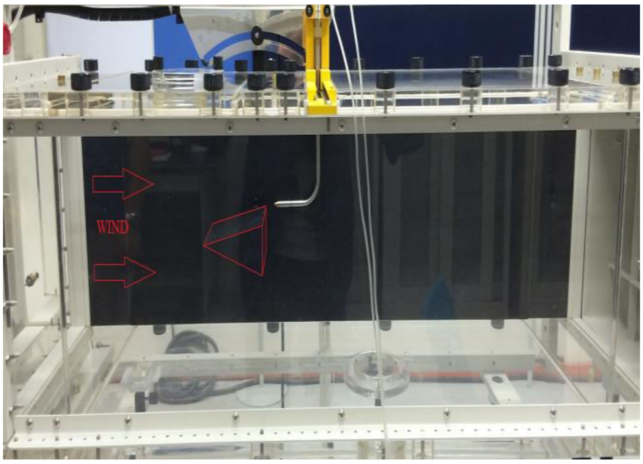

Figure-1. Schematic of the experimental setup.

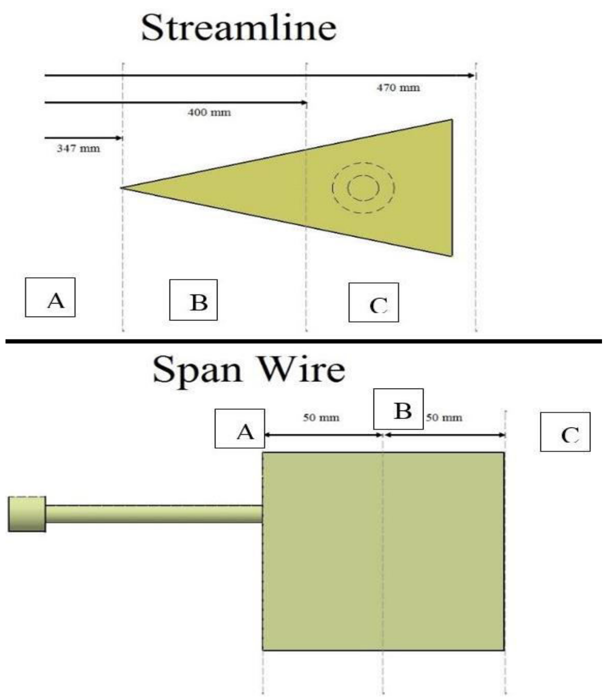

Figure-2. Different position pitot static tube in streamline and span wire.

For flow characteristics, the wedge-shaped model been install horizontally. The dropping frequency of liquid and wind speed is 10V which is a recommended value and to avoid from damaging the smoke wire. The wind speed for flow visualization is 5 m/s. Adjusting the dropping frequency of liquid and wind speed is to optimize the results.

EXPERIMENTAL RESULTS

External velocity

Hydrostatic pressure been measure using pitot static tube and convert into velocity. All the data will be validate on CFD method. Table-1 show the wind tunnel wedge-shaped velocity data (streamline and span wire).

Table-1. Streamline and span wire experiment data.

	Streamline	
347 mm (A)	400 mm (B)	470 mm (C)
5.18 m/s	5.55 m/s	6.15 m/s

	Span Wire	
A	B	C
5.87 m/s	6.15 m/s	5.98 m/s

Flow characteristics

A direct visual of the field of flow is shown in photographs taken in the wind tunnel (Figure-3). Despite the apparent edges shape of the wedge, visualization tests of flow indicate that the wedge is very streamlined and has minimal disturbance on surface. Flow visualization revealed the formation of a vortices at the end of the wedge, these vortices give rise to particularly low pressures especially on wedge end zone.

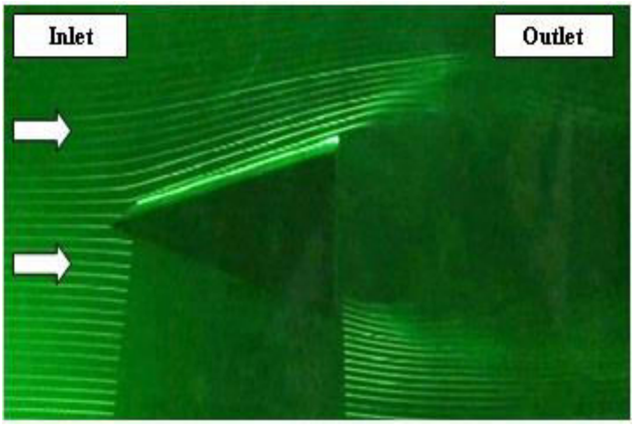

Figure-3. Flow visualization on wedge-shaped.

CFD SIMULATION APPROACH

CFD software package ANSYS FLUENT 15.02 was used to perform the simulations. Time-independent steady-state approach was chosen to simulate the wedge-shaped in n to simulate the wedge-shaped in turbulent flow. The solver was used to solve RANS (Reynolds-Averaged-Navier-Stokes) equations. The selected turbulence model was k-ε Realizable with Non-Equilibrium Wall Functions. Theoretical maximum flow velocity in the tunnel is approximately 100 m/s but due to obscurity of flow visualization, the velocity was constantly set to U_{∞}= 5.18 m/s (Ma = 0.02). The computational domain is created by taking 1000 mm for the length, 300 mm for the height, 300 mm for the width and 360 mm from inlet domain to the wedge-shaped mounting strut. The mesh and solver for CFD is generated by using ANSYS FLUENT 15.02. Grids in the computational domain, focused on separation flow pattern region should be smaller that will influenced the results. Computation domain should be large domain, although it will increase the computation times. The mesh created in this article is

ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

tetrahedral cells consisted 20 000 nodes (Figure-4). The air flowing around the wedge-shaped can be considered as a steady turbulent fluid flow. The coupled scheme are used in pressure-velocity coupling and also generated momentum, turbulent kinetic energy and turbulent dissipation rate using second order upwind.

The turbulent kinetic energy value was calculated with Equation. (4). The turbulence intensity I in the wind tunnel was about one percent and this value was also used to calculate turbulent kinetic energy k used in the simulations as an initial value. The turbulent dissipation rate ε was calculated with Equation. (5) using the value for the turbulent kinetic energy k.

$$k = \frac{3}{2}\left(U_{\infty} I\right)^2 \tag{4}$$

$$\varepsilon = C_{\mu} \frac{k^2}{\nu}\left(\frac{\mu_t}{\mu}\right)^{-1} \tag{5}$$

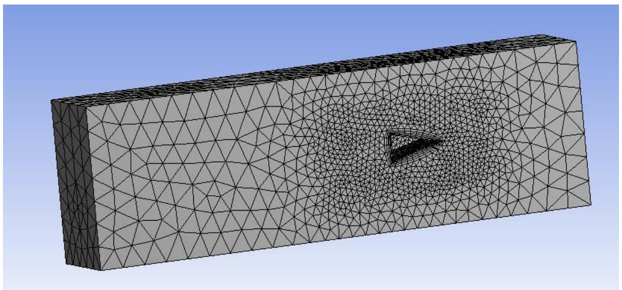

Figure-4. Meshing on the wedge-shaped.

CFD RESULTS ON WEDGE-SHAPED

In the case of the wedge-shaped, the vortex could be detected near the edges and wall. As was observed above, the flow alters the direction suddenly due to the edges and wall influence. The CFD solution shows this phenomenon very understandably (Figure-5a and 5b). See below Table-2 shows data due to coordinate (XYZ) which is equivalent to the pitot static tube positions in streamline and span wire.

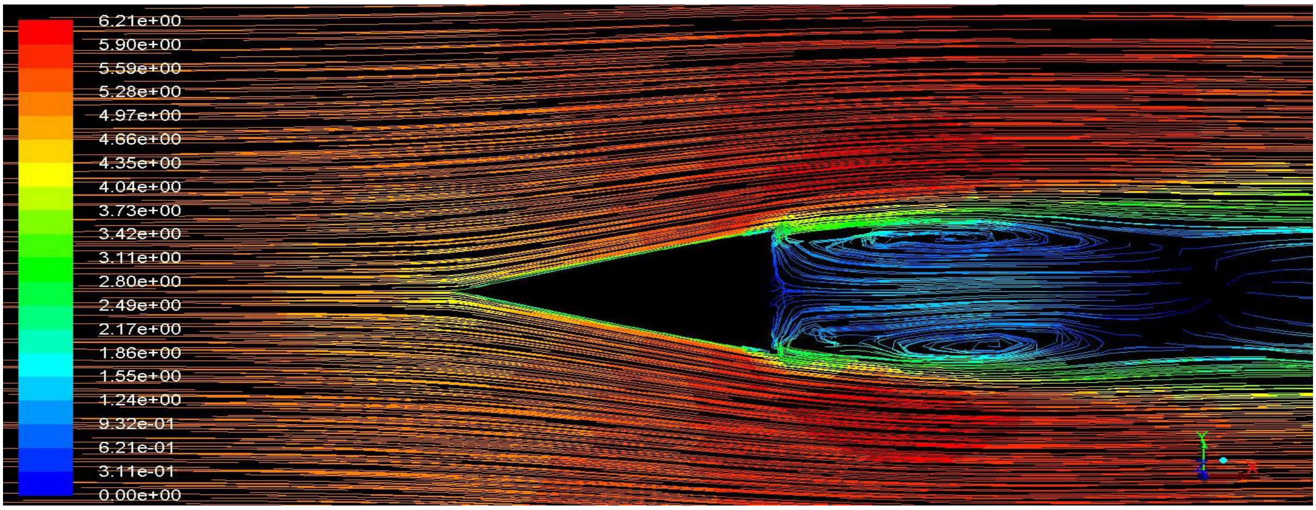

Figure-5a. Wedge-shape CFD results (Pathlines for velocity).

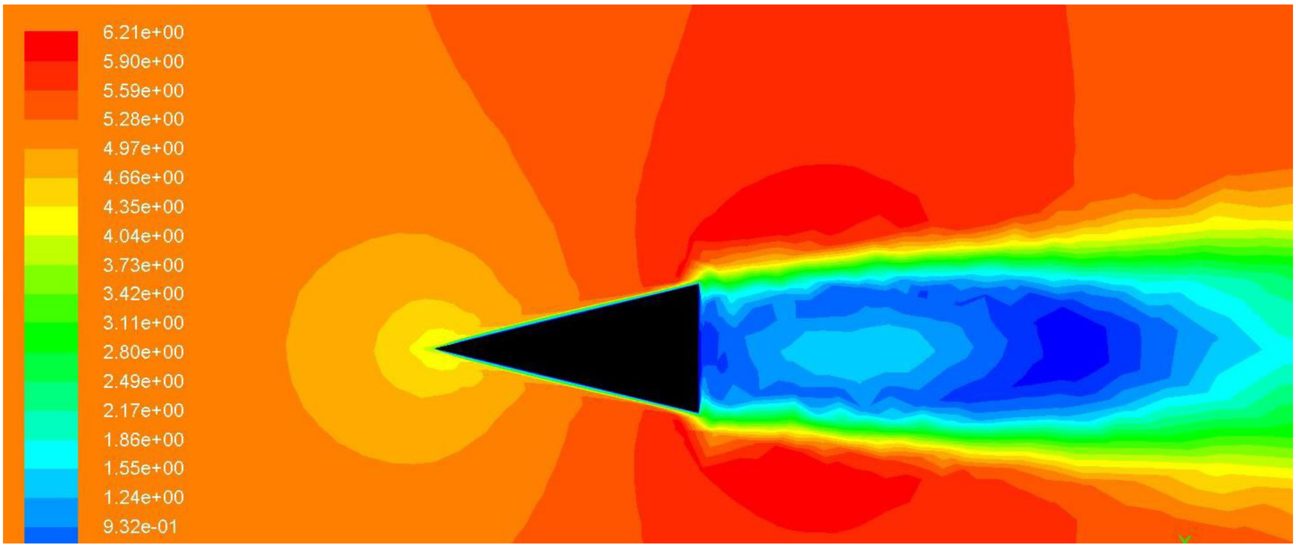

Figure-5b. Wedge-shape CFD results (Contours for velocity).

VOL. 11, NO. 10, MAY 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

Table-2. Coordinate streamline and span wire CFD results.

	Streamline			Span Wire	
(X= -0.096, Y= 0.057, Z= 0.05) A	(X= -0.048, Y= 0.057, Z= 0.05) B	(X= 0.008, Y= 0.057, Z= 0.05) C	(X= 0, Y= 0.057, Z= 0) A	(X= 0, Y= 0.057, Z= 0.05) B	(X= 0, Y= 0.057, Z= 0.1) C
4.91 m/s	5.34 m/s	5.28 m/s	5.37 m/s	5.72 m/s	5.54 m/s

Table-3. Comparison of streamline experimental and simulation results.

Streamline			
Experiment	5.18 m/s A	5.55 m/s B	6.15 m/s C
Simulation	4.91 m/s A	5.34 m/s B	5.28 m/s C
Error Percentage (%)	5.2 %	3.8 %	14 %

Table-4. Comparison of span wire experimental and simulation results

Span Wire			
Experiment	5.87 m/s A	6.15 m/s B	5.98 m/s C
Simulation	5.37 m/s A	5.72 m/s B	5.54 m/s C
Error Percentage (%)	8.5 %	7 %	7.4 %

From the results that been obtain from CFD simulation, there are slightly different (within 5-9 %) compare to wind tunnel test results. It is because CFD simulations ignored the influence of wind tunnel disturbance consequences, the comparability of the interference-free CFD results and experimental data will not perfect. Pitot static tube that been attached during wind tunnel test will affect the wind tunnel test results due to disturbance flow that against the pitot static tube and have a reverse flow.

CFD RESULTS ON ARMOURED VEHICLE MODEL

After validated the wedge-shaped model using both method consisted wind tunnel test and CFD simulation, the model changed into an armoured vehicle (Figure-6a). This armoured vehicle is focused on CFD simulation only due to the limited space of wind tunnel that university have. Armoured vehicle model will simulate using ANSYS FLUENT 15.02 same as the wedge-shaped CFD simulation parameters but the velocity changed to 22.22 m/s (80 km/h) due to a real speed of armoured vehicle on the road. Armoured vehicle geometry consisted main body only due to the computational time and low PC specifications. A complicated geometry will lead to high computational time and ANSYS FLUENT also cannot detect those complex geometry parts. Armoured vehicle geometry is 7 m X 1.2 m X 1.4 m (Figure-6b) equivalent to full scale of armoured vehicle. The mesh created in this article is tetrahedral cells consisted 580000 nodes and simulate using Realizable k-epsilon turbulence model. Figure-6c shows the results of CFD simulation targeted on armoured vehicle. There is low pressures at trail region, top and bottom region and also at the frontal edge. This is due to the sharp and curvy edges at the frontal of armoured vehicle which make the flow separated and became a low turbulence flow.

Figure-6a. Armoured vehicle.

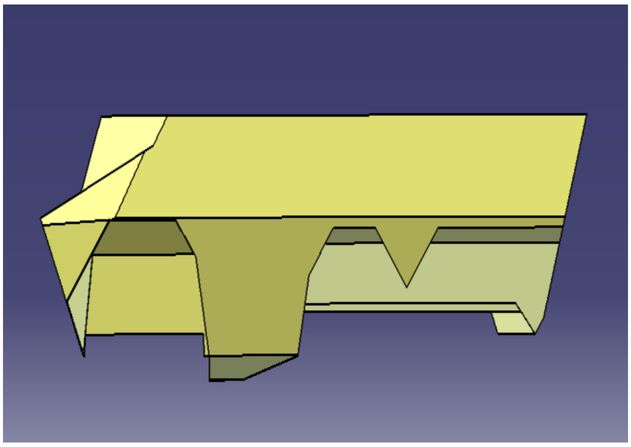

Figure-6b. Armoured vehicle CAD model.

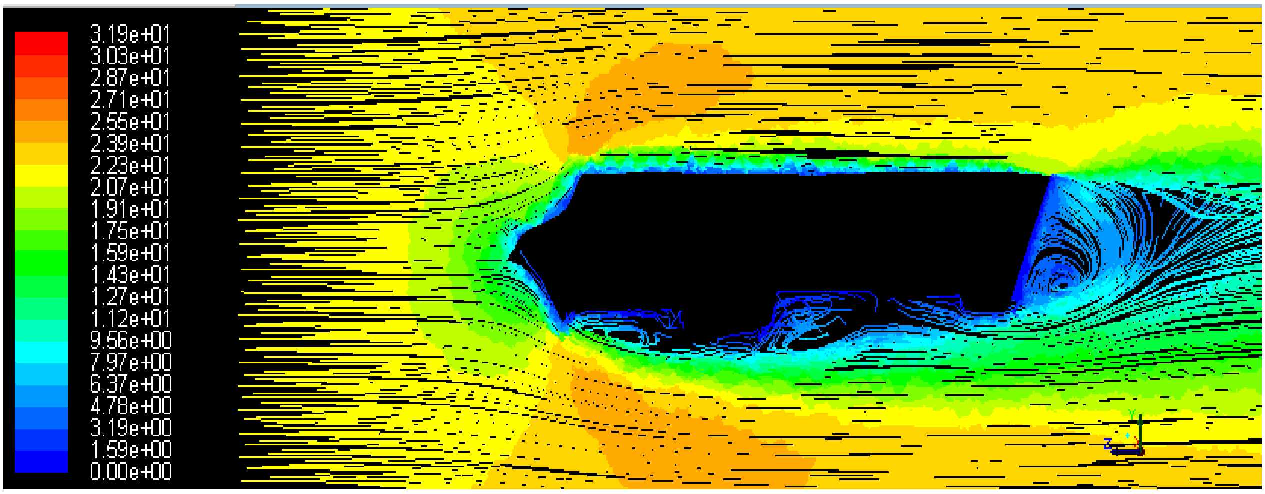

Figure-6c. CFD simulation results on armoured vehicle (Pathlines for velocity).

CONCLUSIONS

Wind tunnel tests have been carried out for the wedge-shaped model. The flow characteristics and hydrostatic pressure with different coordinate using low Reynolds number has been analyzed. On the basis of the experimental results of the wedge-shaped model tests, it can be concluded that the function of the wind speed and angle of attack, are significantly influenced the results.

Computed results obtained on the wedge-shaped model using ANSYS FLUENT are in agreement with the experimental results. An armoured vehicle model is presented in order to better understand the phenomenon and further tests are necessary. Further recommendation to study the flow in water medium (different boundary condition) when the armoured vehicle is deployed in amphibious mode.

ACKNOWLEDGEMENTS

The author wish to acknowledge the research grant provided by Long Term Research Grant Scheme (LRGS) LRGS/B-U/2013/UPNM/DEFENCE&SECURITY-P3 from the Malaysian Ministry of Education, that lead to the realization of this work.

REFERENCES

[1] Chowdhury, H., Moria, H., Ali, A., Khan, I., Alam, F., & Watkins, S. (2013). A Study on Aerodynamic Drag of a Semi-trailer Truck. Procedia Engineering, 56, 201-205. doi:10.1016/j.proeng.2013.03.108.

[2] Coleman, S. A., & Baker, C. J. (1990). High Sided Road Vehicles In Cross Winds. Journal of Wind Engineering and Industrial Aerodynamics, 36, 1383–1392.

[3] Franck, G., & J. D'Elia (2004). CFD modeling of the flow around the Ahmed vehicle model, Proceedings of 2nd conference on advances and applications of GiD 5–8.

[4] Qi, X., Liu, Y., & Du, G. (2011). Experimental and numerical studies of aerodynamic performance of trucks. Journal of Hydrodynamics, Ser. B, 23(6), 752–758. doi:10.1016/S1001-6058(10)60173-4.

[5] Wei, X., Wang, G., & Feng, S. (2008). Aerodynamic characteristics about mining dump truck and the improvement of head shape. Journal of Hydrodynamics, Ser. B, 20(6), 713–718. doi:10.1016/S1001-6058(09)60006-8.