© 2017 IJSRSET | Volume 3 | Issue 3 | Print ISSN: 2395-1990 | Online ISSN : 2394-4099
Themed Section: Engineering and Technology

Investigation of Drag Coefficient at Low Velocity for Front of Two Wheels Vehicle Based on CFD method

Indresh Kumar Jain[1], M. S. Khidiya[2], M. A. Saloda[3], Chitranjan Agarwal[2]

Department of Mechanical Engineering ,CTAE, Udaipur Rajasthan, India

ABSTRACT

Two wheels vehicles for city use are common in Asia and particularly in India. Two wheels vehicle is in large population because of affordability, efficiency and economic purposes. In the 10 to 20 years the market turned to bigger swept volumes vehicles such as 250 and 500 cc. This makes two wheels vehicle market richer and more adult, which needs a attractive design, comfort and stability at all speed. On the other hand, most of the vehicles for city uses at low velocities. For better performance of vehicles need more development of the engines and design makes two-wheeler aerodynamic characteristics analysis is much more necessary now. In this work, by using CFD method external flow field of a two wheels vehicle's front is analysed including the driver. Summarized data and the computational results are obtained, including velocity field, pressure field, and coefficient of drag. This work will provide specific reference significance to the numerical simulation on motorcycle and motorcycle design on the future.

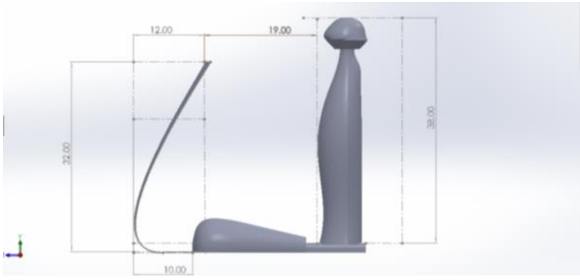
Keywords : Two-wheeler, Rider, CFD, Simulation, Coefficient of Drag.

I. INTRODUCTION

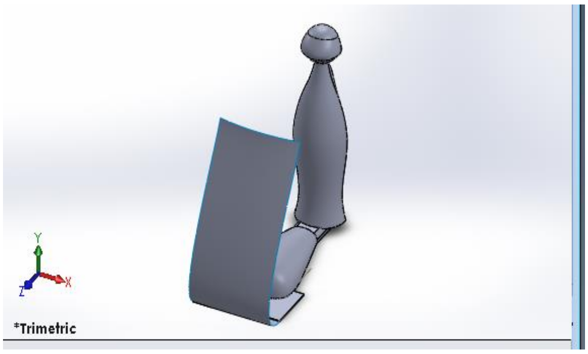
CFD numerical simulation method is a method of calculating the discrete numerical solution for the control equation of the flow field around the vehicle. The development of CFD methods and simulation enables the cheap accurate and fast development of vehicle for any environment conditions (Angeletti, Sclafani, Bella, & Ubertini, 2003). Due to the less availability, the power consumption comfortable in laminar flow results and inefficient in dynamic calculation the wind tunnel experiment has been loss its attention towards researches. CFD simulation and numerical unsteady methods has been adopted by the automobile sector to design and analysis purpose of automobile. In some aspect it has been quite successful. Due to the complexity of shape, control and analysis of two-wheeler it have not been analysed well which leads two wheels automobile research to out of margin (Chu, Chang, Hsu, Chien, & Liu, 2008; Gentilli, Zanforlin, & Frigo,2006).

There are an estimated 154.3 million Two-Wheelers registered in India according to the international organisation of motor vehicle manufacturers 2016 statics. The continuous popularity of two- wheeler among the high population of India due to economic and speed purpose the analysing of two-wheeler is must. The study can affect on improvement of performance and good aerodynamics of two-wheeler with rider, at the same time if CFD and numerical simulation adopted at the initial stage of design of two-wheeler can leads great overall quality improvement to manufacturing sector. CFD can provide detailed information on the characteristics of the flow field on around the vehicle. Especially in the two-wheeler research, CFD can play a shorter development cycle with less time and cost. CFD is capable to evaluate the data at the current stage of the system that cannot be possible in wind tunnel. Such as shorter distance, low velocity and turbulent flow the CFD gives better result, the commonly used software in ANSYS FLUENT, STAR- CD, CFX , POWER FLOW and so on.

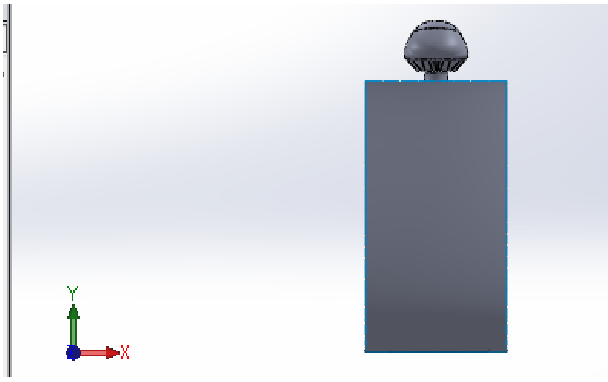
The aim of this study is to provide an improved evaluation of coefficient of drag and understanding of the instantaneous flow around a two-wheeler's front subjected to straight wind with laminar and steady state techniques. In the current work, the velocity is 22.22 m/s ground. The ANSYS FLUENT 15 was used to solve the flow equations.



(I)



(II)



(III)

Figure - 1

II. Two- wheeler front model

The current research was carried out on a two wheeler front with rider (see Figure 1) ,including the nomenclature adopted in this current work. The coordinate system used in this paper is also shown, where z in the direction of travel, y is in the vertical direction and z is in the lateral direction. The negative motorbike velocity (U_Z) is applied, which acts relative velocity to the vehicle direction of travel. The length, height and width of the two wheeler front with rider are 29 inch , 38 inch and 12 inch, respectively. The vehicle model maintains a high level of geometrical detail including the main two wheeler components, although small details such as cables and bolts are omitted.

III. Numerical Method

The finite volume CFD package ANSYS FLUENT 15 was used to solve the incompressible flow equations in all the simulations of this paper. The flow around a vehicle subjected to normal to the two wheeler is dominated by highly three-dimensional flow structures. In order to obtain information about these laminar flow structures in pressure – velocity coupling based on simple algorithm is used.

In the algorithm after Set the boundary conditions the gradients of velocity and pressure has been evaluated than the discretized momentum equation has to be solved to compute the intermediate velocity field .By Computing the uncorrected mass fluxes at faces pressure correction equation solved to produce cell values of the pressure correction and Updation of pressure field has been occurred by

$$p^{k+1} = p^{k} + urf \bullet p^{'}$$

where urf is the under-relaxation factor for pressure.

Than after Updating the boundary pressure corrections the correct face mass fluxes has been evaluated

$$\dot{m}_f^{k+1} = \dot{m}_f^{*} + \dot{m}_f^{'}$$

Correct the cell velocities:

$$\vec{v}^{k+1} = \vec{v}^{*} - \frac{Vol\nabla p^{'}}{\vec{a}_P^{v}} ;$$

where $\nabla p^{'}$ is the gradient of the pressure corrections,

$\vec{a}_P^{v}$ is the vector of central coefficients for the discretized linear system representing the velocity equation and Vol is the cell volume.

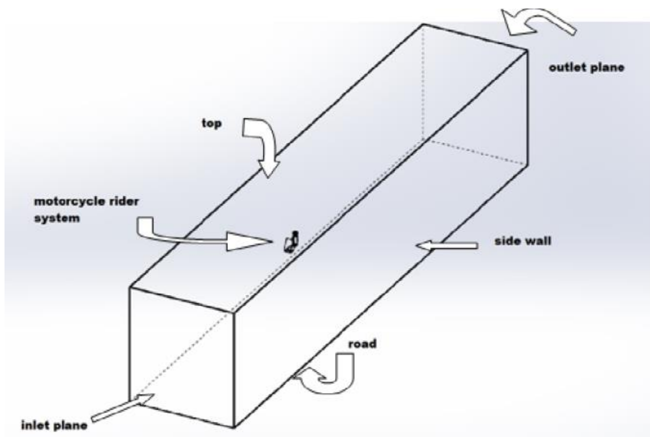
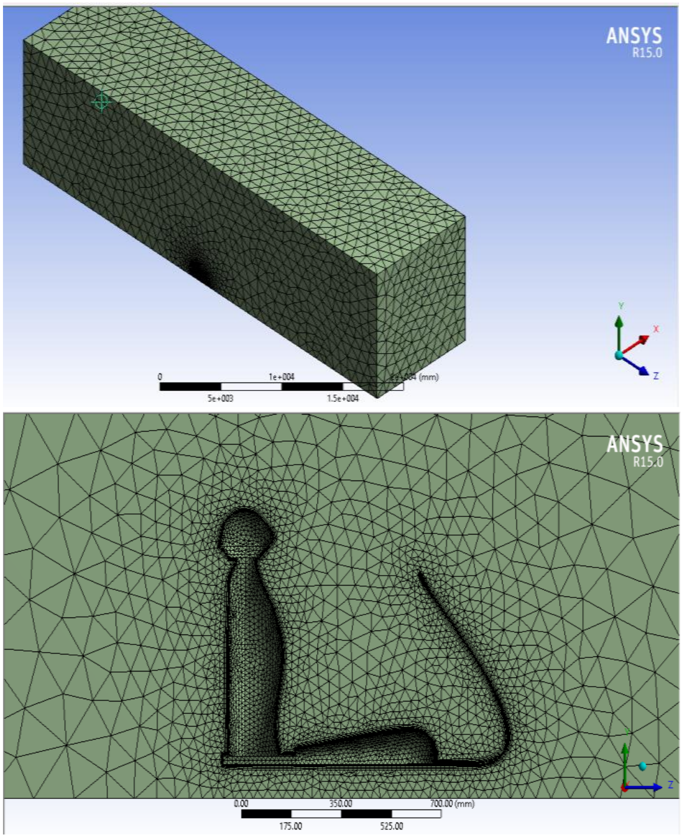


Figure 2. Computational Domain

IV. Computational domain and boundary conditions

A generalized computational domain was used in this investigation (see Figure 2). One inlet and one outlet boundaries are used to simulate the flow condition. At the inlet boundaries, the flow in the negative direction of travel, Z, with constant at 25 m/s in simulation and whole system is put in 1 Atmospheric condition. The total Z-dimension was set as 40 m. These distances from the motorbike surface to the exit plane were chosen to be large enough for the zero-pressure exit boundary condition to be applied without affecting the flow or pressure fields around the motorbike. No-slip boundaries were applied on the surface of the vehicle, rider and ground surface. The ground surface was simulated as asmooth wall with a velocity identical to the motorbike's speed (U_Z = 22.22m/s), representing correct relative movement between the motorbike and the ground. The surface mesh was created as the shown in figure 3(i) and 3 (ii). The most of the meshes are 88 % tetrahedral the minimum face size is 15 mm and maximum face size is 1000 mm. at the surface mashing. 129031 elements and 24391 nodes have been created in present model.the medium meshing is selected for better simulation results. Five inflation layers also has been created for better flow simulation.



V. Results and Discussion

It is can be assumed that the calculated values have been reached the physical convergence and it can used for analysis

A. Coefficient of drag analysis

From the analysis results the model has a coefficient of drag in the range of 0.6 to 0.7 range (Cheng et al., 2011). The results are similar to those of the motor cycle in the reference. At the same time results gives good two wheeler shape design respectively.

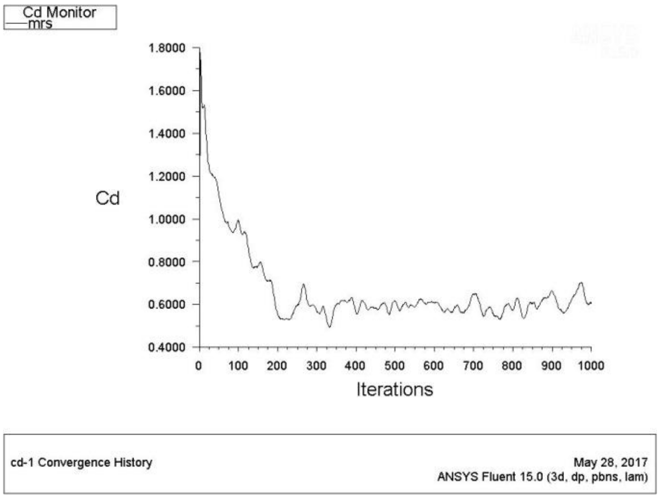


Figure 4

B. Pressure gradient analysis

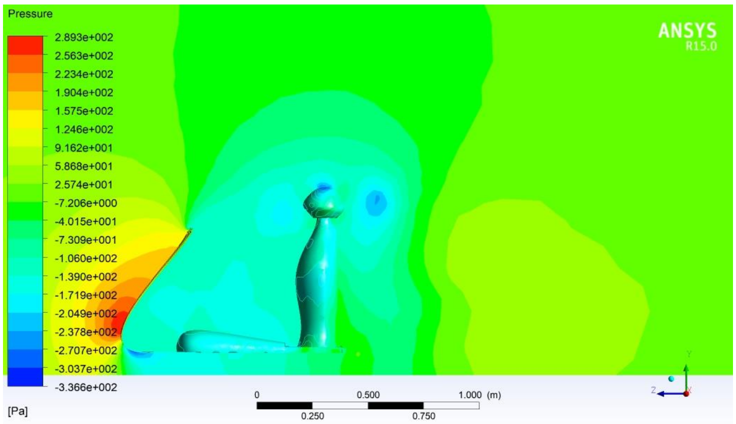


Figure 5

It shows the pressure distribution on two wheeler and human surface body the envelope which is created in the range of 0.51 pascal to 0.86 pascal (relative dynamic pressure)

VI. Conclusion

It has been seen from the results of numerical simulation test of two wheeler in this paper that it is convenient and effective to carried out CFD analysis of vehicle in the case of wind tunnel test. In the design stage the designer can be evaluate the parameter related to aerodynamics in the basic level of design at the same time CFD simulation analysis overcome the previous traditional wind tunnel testing limitation. By the use of CFD numerical simulation the product cycle is short and advantage can be seen in time and cost.

VII. REFERENCES

[1] Angeletti, M., Sclafani, L., Bella, G., & Ubertini, S. (2003).The role of cfd on the aerodynamic investigation of motorcycles. SAE Technical Paper 2003-01-0997.doi:10.4271/2003-01-0997

[2] Araki, Y., & Gotou, K. (2001). Development of aerodynamic characteristics for motorcycles using scale model wind tunnel. SAE Technical Paper 2001-01-1851. doi:10.4271/2001-01-1851

[3] Ribaldone, E., Puri D., Cogoti, I., Pizzoni, R., 2011. Optimizing the external shape of vehicle at the concept stage: integration of aerodynamics and ergonomics, SAE International Journal of engines.(4)2:2622-2628.

[4] Sivan, S. K., 2014. Fairing flap drag reduction mechanism (FFDRM), 5(9): 435-439.

[5] Saurabh, B., 2015. CFD Simulation of Flow around External Vehicle: Ahmed Body, IOSR Journal of Mechanical and Civil Engineering, 12(4): 87-94.

[6] Selvaraju, P.N., Parammasivam, K.M., and Devaradjane, G., 2015. Journal of chemical and pharmaceutical technology.(7).

[7] Scappaticci, L., and Risitano, G., Battistoni, M., and Grimaldi, C., 2012. Drag Optimization of a Sport Motorbike, SAE International Journal of Commercial vehicle.

[8] Paolo, C., ARGENTO, M., 2014. Parametric modelling and CFD analysis of maxi scooters: a design of Experiment approach.

[9] Cheng, P.K., Ting-chao, Z., and Jie, L. 2011. Civil motorcycle aerodynamics charactristic based on CFD method, International Conferance on Consumer Elecrtronics And Networks.

[10] Debraux, P., Greppe, F., Manolova, A. V., and Bretucci, W., 2011. Aerodynamics drag in cycling : method of assessment, Sports Biomechanics 10(3): 197-218

[11] Enrico, R., 2011, Optimizing the External Shape of Vehicles at the Concept Stage: Integration of Aerodynamics and Ergonomics , SAE Int. J. Engines, 4(2).

[12] Fintelmana, D., 2015. A numerical investigation of the flow around a motorbike when subjected to crosswinds, Engineering Applications of Computational Fluid Mechanics, 9(1): 528–542.

[13] Toshio Koba yashi,Kozo Kiton,A review of CFD methods and their application to automobile aerodynamics [J].SAE Transaction, pp. 377 – 388,1992.

[14] Fu LiMing,The Numerical Calculation Of Automobile Air [M].Beijing: Beijing University of Technology Press,2000.

[15] Zhang YinChao,Yang Bo,The Aerodynamic Characteristics Of Motorcycle[J]. Small Internal Combustion Engine and Motorcycle, February 2007