Numerical simulation of an Indian auto-rickshaw model

Neha Singh, Saransh Abbey, Pratyush Kumar Singh and Kannan B T
Department of Aerospace Engineering, SRM Institute of Science and Technology, Kattankulathur, Chennai, Tamil Nadu, 603203, India.

E-mail: nehasingh_jaishankar@srmuniv.edu.in, saranshabbey_harish@srmuniv.edu.in, pratyushkumarsingh_pramod@srmuniv.edu.in, skyinventorbt@gmail.com

Abstract. Most of the cities in India offer auto-rickshaw services for short and large distances at an economical price. Yet there is little importance given to its study provided that there is a lot of scope to improve its aerodynamic efficiency. The objective of this work is to carry out numerical analysis to predict the aerodynamic characteristics of the vehicle. This numerical study is carried out using CFD software ANSYS-Fluent by solving Reynolds Averaged Navier Stokes equations with a realizable K-ԑ turbulence model via first-order schemes. The results obtained indicate the flow field variation in terms of pressure and velocity around the bluff body. This would also give an understanding of flow physics for further improvement in aerodynamic forces.

1. Introduction
There has been an extensive study on the external aerodynamics of cars for optimization in order to make it efficient to reduce the fuel consumption and prevent any undesirable lift forces. Little importance has been given to the field of study of internal aerodynamics. India being a developing nation, has seen an exponential growth in the automobile industry. This calls for investing the research and development in the study of aerodynamics of the most commonly used vehicle- auto-rickshaw. In order to fulfil the demand, simulations have to be carried out to understand the flow field inside the vehicle compartment which can be followed up with improvements.

The traditional methods of finding aerodynamic characteristics using wind tunnel is tedious and expensive. Thus resorting to numerical study of the scaled model of the selected auto-rickshaw model will be a much more pragmatic approach despite CFD having certain limitations such as the RAM, CPU time.

2. Literature Survey
There hasn’t been much study on the auto-rickshaw model. But there has been quite some progress in the field of numerical simulation of SUVs, F1 cars and generic passenger cars. Ahmad N et al. [1], worked on the strategy to accurately find the value of drag (within 4% of the experimental data) of a vehicle by developing optimized mesh parameters. This meshing strategy can be extended to study external aerodynamics of various other ground vehicles with accuracy while consuming less time. There has been a study on the external aerodynamics of an SUV by Al-Garni A M et al. [2]. They gathered detailed experimental data for setting the benchmark towards validation of the CFD results for a pickup truck model. Altinisik A et al. [3]. also carried out a numerical investigation on the passenger car model in order to find out the pressure distribution as well as drag. The numerical results were validated with the help of a scaled-down model of the car (1/15). Bayraktar I et al. [4].
study was again concerning the external aerodynamics of the Ahmed body. The experimental data was backed up by carrying simulation of the model with varying back angles and calculate the lift and drag coefficients. Duncan B et al. [5], assessed the vehicle’s external flow. The drag was calculated by breaking down part by part component, surface drag was found out. Apart from this, aeroelastic analysis was also done to analyse the underbody noise. This was done for establishing the vehicle’s performance at the very early stages of the designing process of the vehicle. Jindal S et al. [6], carried out numerical simulation on an SUV and general pickup truck to an accuracy of 6% of the experimental data. Immersed body technique along with RANS CFD solver was used reducing the meshing time. Krajnović et al. [7], suggested an alternative to LES (Large-eddy simulation) to understand the external flow around the vehicle. The simulations carried out were on the cube and a bus.

The simulations agreed with the experimental result in predicting the horseshow vortex and the formation of the separation bubble. Lanfrit et al. [8], suggested various strategies to carry out CFD simulation over the vehicle to study external aerodynamics efficiently. Using this document, the meshing conditions and set up for steady-state simulations were made. Meile W et al. [9], Carried out an experimental and numerical study on the Ahmed body at two different slant angles i.e. $25^\circ$ and $30^\circ$. Simulations were carried out with the RSM turbulence model. The numerical data were found to be comparable with the experimental results of Lienhart H et al. [10].

3. Geometry
The auto-rickshaw model is created on CAD-system CATIA V5. The geometry is similar to the production shape with very little variation. While carrying out simulation in CFD software, it is necessary for the geometry to have the least simplifications [1], having a closed surface.

![Figure 1. Dimensions of the auto-rickshaw model.](image)

4. Computational Domain
In order to form an air volume around the auto-rickshaw model, it is important to create a volume of air around the model which will act as a virtual wind tunnel. For these purposes, a domain has to be constructed around the model using the Design Modeller. According to the research done previously on Ahmed body [1], the domain should extend to 3 vehicle lengths in the front of the vehicle and 5 vehicle lengths at the aft of the vehicle. In the case of geometries with holes, gaps in edges, repairs have to be done by setting minimum hole/edge limits before proceeding with the meshing. The auto-
rickshaw model is subtracted from the domain, and an internal box is made for the local refinement during the meshing of auto-rickshaw with cavity.

The computational domain for studying the external aerodynamics around the auto varies by not incorporating an internal box for local refinement as it seems unnecessary.

![Figure 2](image1.png)  ![Figure 3](image2.png)

**Figure 2.** Computational domain of auto-rickshaw to study external aerodynamics.  **Figure 3.** Computational domain of auto-rickshaw to study internal aerodynamics.

5. **Meshing**

5.1 *Specifying Global Mesh Settings*

To carry out the meshing of the geometry, ANSYS Meshing which is a component of ANSYS workbench is used. In the early phases, coarse mesh is chosen for preliminary analysis since it requires less time for computation. The advanced size function is chosen as “Proximity and Curvature” in order to have a more controlled grid with the ability to give minimum and maximum size controls. The relevance centre is chosen as a coarse and curvature normal angle is $18^\circ$ to capture the geometric details. The minimum size, maximum tetrahedron size and growth are chosen such that an overall mesh of medium coarseness is achieved for the far-field.

5.2 *Local Mesh Settings*

In order to achieve a much refined surface mesh at and around the target body, sizing and inflation are inserted. Face sizing is provided to all the faces of the auto with element size as 20mm and growth rate of 1.2. For local refinement around the vehicle body, body sizing is provided to the inner rectangular domain acting as the body of influence with element size as 80mm and growth rate of 1.2. This provides for a finer mesh around the body with more number of elements present around the auto as well in the wake region of it. The overall number of elements generated in the mesh is 1231407. Figure 4 shows the mesh generated in order to study the internal flow over an auto-rickshaw. For generating mesh around the auto-rickshaw with no cavity in order to study the external aerodynamics, the global mesh settings are the same. In local mesh settings, the inflation layer is provided for capturing the boundary effect of the flow. The inflation options given are as follows- Inflation option: first aspect ratio, first aspect ratio as 5, maximum layers as 5 and growth rate as 1.2.

The overall number of elements generated in the grid is 1230561. Figure 5 shows the mesh generated in order to study the external flow over an auto-rickshaw.
Figure 4. Mesh of auto-rickshaw to study internal aerodynamics with a close up view of the object of interest.

Figure 5. Mesh of auto-rickshaw to study external aerodynamics.
6. Grid Independent Study
In order to carry out grid independent study for the auto-rickshaw model without cavity, three mesh sequences are made as follows: mesh 1 which comprises of 8 lakh elements, mesh 2 which has 12 lakh elements and mesh 3 which has 20 lakh elements. The mesh varies from coarse, medium to fine respectively. This study is done in order to make sure that the results acquired are accurate and is not dependent on the grid size. On examining the graph plotted, it is observed that there’s very little difference between the coefficient of drag of mesh 2 type (Cd=0.2539) and coefficient of drag of mesh 3 type (Cd=0.25368). A similar trend is seen for the auto-rickshaw model with cavity.

![Figure 6. Grid independent study of auto-rickshaw without cavity.](image)

7. Steady Flow Numerical Simulation
Pressure based solver is used to simulate the low-speed incompressible flow. Steady simulation is opted. Realizable K Epsilon model (2eqn) with wall function (Non-equilibrium) is used because it performs better in adverse pressure gradient and provides more accurate results of the behaviour of viscous sublayer. Non-equilibrium wall function is ideal for simulating external simulations.

Inlet boundary conditions are defined with a magnitude of 12m/s since the city limitation of Bajaj auto-rickshaw is 40 kmph, turbulence intensity of 1% and turbulent viscosity ratio of 10%. For pressure outlet boundary, backflow turbulent intensity is 5% and backflow turbulent viscosity ratio is 10%. The auto body is provided with a no-slip under shear condition as it is stationary in the wind tunnel. The reference values are computed from inlet with area= 0.8191023 m^2 which is half of the frontal projected area of the auto-rickshaw, density=1.225 kg/m^2 and length= 2.63228m of the vehicle body.

Pressure based coupled solver is used for solution methods which solved both velocity and momentum equations as coupled. This helps to converge faster at the cost of using more resources. First-order upwind is used for momentum and turbulent quantities as it converges easily. For highly skewed meshes with tetrahedral elements, explicit relaxation factors of 0.5 or below is used.
8. Simulation results

8.1 External Aerodynamics of Auto-rickshaw
From figure 7 it can be seen that the stagnation point is achieved at the front end of the auto body. The coefficient of pressure develops into negative values wherever the airflow accelerates which is at the top rear and front ends of the auto body as seen in figure 8. The $C_d$ of the auto with the cavity is predicted to be 0.688.

![Figure 7. Static pressure contour over symmetrical plane with no cavity.](image1)

![Figure 8. Pressure coefficient contour over auto-rickshaw body with no cavity.](image2)
The contour in figure 9 helps in investigating the energy contained in fluctuations. Wherever there are more fluctuations, there’s more kinetic energy that is present at the trailing side of the vehicle body. From figure 10 it can be seen that there is a higher turbulent intensity where vortices are present compared to other regions of the auto body.
From analyzing the flow streamlines around the auto-rickshaw body, it is observed that a wake region is formed at the rear of the body which leads to drag. Closely spaced streamlined at the bottom means that there is a high velocity and thus less static pressure. The pressure gradient which is observed between the upper portion and the lower portion of the vehicle leads to a negative lift (downforce). The pressure difference present between the roof of the vehicle and the underside leads to trailing vortices which extends back to the aft of the vehicle. The region between the reattachment and separation point leads to the formation of the separation bubble at the aft end.

8.2 Internal Aerodynamics of Auto-rickshaw

The contour in figure 13 helps us understand the areas where there is more kinetic energy. These regions comprise the internal back cavity region where the passenger sits as well as the aft portion of the body. From figure 14 it can be analyzed that there’s vortex formation present in the internal compartment of the auto causing high turbulent intensity displayed in the yellow-colored region.

The flow field in figure 15 shows the velocity streamlines at the plane parallel to the symmetric plane. This is a circulatory flow pattern produced in the front portion of the interior cavity of the auto-rickshaw. The centre of the circulatory flow corresponds to the minimum pressure region. Reverse flow occurs past separation point forming eddies releasing a lot of energy known as the wake region. There’s also a formation of horseshow vortex.

**Figure 11.** Velocity streamline over auto-rickshaw.
Figure 12. Static pressure contour over symmetry plane.

Figure 13. Turbulent kinetic energy contour of auto-rickshaw along the symmetry plane with the internal cavity.
Figure 14. Turbulent intensity contour of auto-rickshaw along symmetry plane with internal cavity involved.

Figure 15. Velocity streamline over symmetry plane for understanding internal flow.
9. Conclusion
The external and internal aerodynamics was studied for an auto-rickshaw model at velocity=12m/s. The simulations results were carried out with the help of a realizable k-ε turbulence model combined with non-equilibrium wall for near-wall function which helped in prediction of the coefficient of lift and drag across both the auto-rickshaw with and without cavity. Pressure, turbulence kinetic energy and turbulence intensity contours along with velocity streamlines helped in visualizing the real-life physical scenario of how the flow field is around an auto-rickshaw. The wake region is perfectly visualized along with the internal flow structures. The more detailed work on this vehicle would lead to higher efficiency and lesser pollution.

Acknowledgments
The author wishes to acknowledge the usage of the Advanced Computing Lab for a part of documentation work and also for the resources provided by SRMIST.

10. Reference
[1] Ahmad N E, Abo Serie E and Gaylard A 2010 Mesh optimization for ground vehicle Aerodynamics CFD Lett. 2 54–65
[2] Al-Garni A M, Bernal L P and Khalighi B 2004 Experimental investigation of the flow around a generic SUV SAE Tech. Pap.
[3] Altinisik A, Kutukeeken E and Umur H 2015 Experimental and numerical aerodynamic analysis of a passenger car: Influence of the blockage ratio on drag coefficient J. Fluids Eng. Trans. ASME 137
[4] Bayraktar I, Landman D and Baysal O 2001 Experimental and computational investigation of Ahmed body for ground vehicle aerodynamics SAE Tech. Pap.
[5] Duncan B D, Senthooran S, Hendriana D, Sivakumar P, Freed D, Gleason M and Hall D C 2007 Multi-disciplinary aerodynamics analysis for vehicles: Application of external flow simulations to aerodynamics, aeroacoustics and thermal management of a pickup truck
[6] Jindal S, Khalighi B and Iaccarino G 2005 Numerical investigation of road vehicle aerodynamics using the immersed boundary RANS approach *SAE Tech. Pap.*

[7] Krajnović S and Davidson L 2002 Development of large-eddy simulation for vehicle aerodynamics *ASME Int. Mech. Eng. Congr. Expo. Proc.* 2 165–72

[8] Lanfrit M 2005 Best practice guidelines for handling Automotive External Aerodynamics with FLUENT *Fluent* 2 1–14

[9] Meile W, Brenn G, Reppenhagen A, Lechner B and Fuchs 2011 A Experiments and numerical simulations on the aerodynamics of the Ahmed body *CFD Lett.* 3 32–8

[10] Lienhart H and Becker S 2003 Flow and turbulence structure in the wake of a simplified car model *SAE Tech. Pap.* 323–30