

Research Article

## External Aerodynamic Analysis of TATA Nano using Numerical Tool

Manikandan. M<sup>A\*</sup>, Shiva Prasad U<sup>A</sup>, Ashish Ashok Suvarna<sup>B</sup>, Anuj Bhat B<sup>B</sup>, Vinayak Nair<sup>B</sup> and Gunda Shivakrishna<sup>C</sup>

<sup>A</sup>Department of Aeronautical Engineering, MIT- Manipal University, Manipal, Karnataka, ,INDIA.

<sup>B</sup>Department of Mechanical Engineering, MIT- Manipal University, Manipal, Karnataka, 576104,INDIA.

<sup>C</sup>Department of Aeronautical Engineering ,Vardhaman college of engineering ,Shamshabad ,501218, AP INDIA

Accepted 10 January 2014, Available online 01 February 2014, **Special Issue-2, (February 2014)**

### Abstract

*Fuel efficiency is the prime focus of the present work. Simulations are carried over external aerodynamic body to determine the drag and its coefficient by means of virtual environment with the help of commercial tool. The use of computer software makes it possible to be in time and cost savings. Numerical methods are employed on three-dimensional vehicle structure to resolved body forces. The present work is carried using the ANSYS-CFX and the equations governing fluid flow combined with standard model and solved by using the appropriate boundary conditions. The study results of the aerodynamic analysis of the air flow, pressure distribution and drag force on the vehicle. Few design modifications are made in the form of attachable accessories which have been resulted in reduction of the drag coefficient. These modifications have shown a drag reduction of 14.28% at a speed of 60 km/h with the drag coefficient reduced to 0.336 from 0.392, thereby reducing the fuel consumption allotted to external body by an amount of approximately 14.28%, which means, there is a saving of 2.14L of petrol for every full tank refill.*

**Keywords:** Automotive Aerodynamics, Computational Fluid Dynamics (CFD), Drag Coefficient, Drag Reduction, Fuel Consumption, Numerical Analysis, Tata Nano, ANSYS, visualization, environments, drag force, prototype

### 1. Introduction

Tata Nano was envisioned in 2003 to become a ‘people’s car, that is affordable by everybody, mainly the city dwellers of India. Since its debut in 2009, the sales of Tata Nano have gone down tremendously. In the first quarter of 2013, number of vehicles sold in the months of January, February and March were 1504, 1505 and 1507 units, respectively. When it was first conceived, a target price of 100,000 rupees was established but it crept up to about 142,000 rupees in very short time due to a steep increase in price of raw materials and input costs. Added to this, the soaring fuel prices, it’s no longer an easily affordable car. Studies have shown that 15 percentage of the fuel in a vehicle is used to overcome mechanical friction in the engine, 45 percentage of the fuel is used to overcome the rolling resistance and drag is responsible for the remaining 40percentage. A car with low drag force encounters lesser friction than a car with higher drag force. Hence, a smaller engine is required to drive such a car which eventually lowers the fuel consumption. In this paper we focussed on reducing the overall drag force on the vehicle, and thereby bringing down the fuel consumption, hence making the Tata Nano again a favourite among the Indian masses.

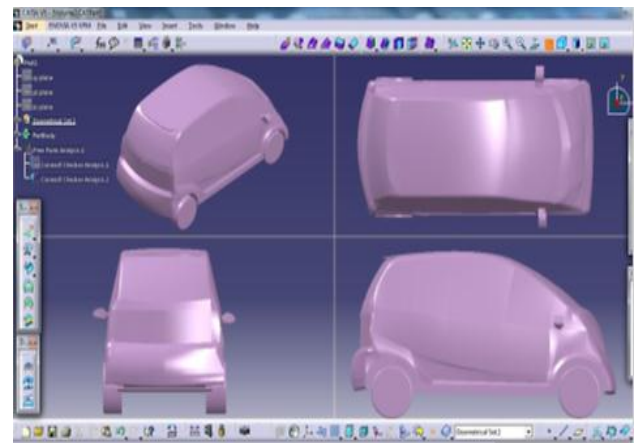
### 2. Objective

The key to aerodynamic research of TATA NANO car is

to perform external aerodynamic analysis using numerical tools in order to improve the vehicle aerodynamics and overall performance of the vehicle by reducing the Coefficient of drag ( $C_d$ ) and to improve the performance and efficiency of the vehicle.

### 3. Methodology

The fundamental basis for almost all CFD problems is the Navier–Stokes equations, which define any single-phase fluid flow.



**Fig. 1** Geometrical model of TATA NANO Car in CATIA

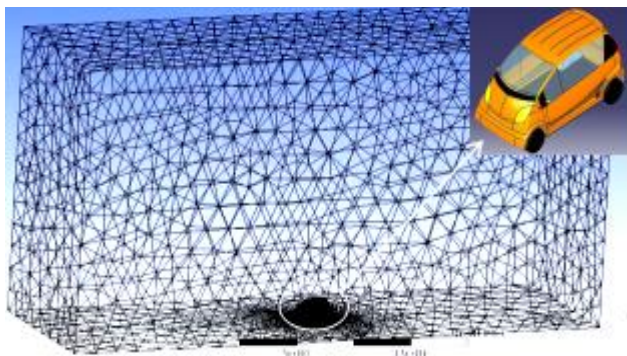
\*Corresponding author: **Manikandan. M**

The equations were derived independently by G.G. Stokes, in England, and M. Navier, in France, in the early 1800's. These equations describe how the velocity, pressure, temperature, and density of a moving fluid are related. The equations are extensions of the Euler Equations and include the effects of viscosity on the flow.

In practice, these equations are too difficult to solve analytically. In the past, engineers made further approximations and simplifications to the equation set until they had a group of equations that they could solve.

**4. Geometrical Details and Discretization**

The domain was discretized using ICFM CFD. The mesh type was tetrahedral with a total of 1149826 tetrahedral elements. The solution was calculated using fine mesh resulting in 208079 nodes as shown in fig 2. Analysis was made using ANSYS V13 subjected to similar boundary conditions as that of the experimental analysis of the prototype. Later the profile was modified from the current profile of Tata Nano using software to improve aerodynamic properties of the vehicle. Comparison of aerodynamic properties and performance of the vehicle was done by the addition of spoilers, air dam, rear screen and without these accessories. To know the performance the parameters of the vehicle.



**Fig. 2** Discretized domain with right corner geometrical

**5. Numerical Simulation**

In design of ground and aerial vehicles, an understanding of flow behaviour is critical. Armed with this knowledge, engineers can optimize their performance and streamline the bodies to increase efficiency. This information may be obtained experimentally, analytically or numerically. Though wind tunnel testing results are preferred but the visualization of flow over the model can only be predicted with high cost involved equipment's only comparatively numerical simulations, are only an approximation, but provide the best compromise between visualization, cost and accuracy. Analysis of the flow over automobiles using CFD was initially introduced in F1 car racing event which helped in modifying the design of the racing cars to get the maximum down force and least possible drag. Later this method of analysis was used in different segments of cars to optimize the design, so that, improvement in the fuel-efficiency and other performance parameters was achieved.

In a modern vehicle, the overall shape contributes to 45% of drag while the shares of wheels, their arrangement, the bottom, and its design details, are 30% and 25%, respectively.

**5.1 Boundary Conditions**

The working fluid was specified as air at 25 °C and 105 Pa. Steady state flow model was imposed. The flow of air around the car is taken to be as 60 km/h (16.67 m/s). The turbulence intensity on the inlets was chosen as medium turbulence case with 5% intensity.

**Table 1** Boundary conditions

Boundary	Boundary conditions
Car surface	Free Slip Wall
Inlet	Velocity inlet
Outlet	Pressure outlet
Side walls	Free Slip Wall
Top and bottom surfaces	Free Slip Wall

**6. Improvisation**

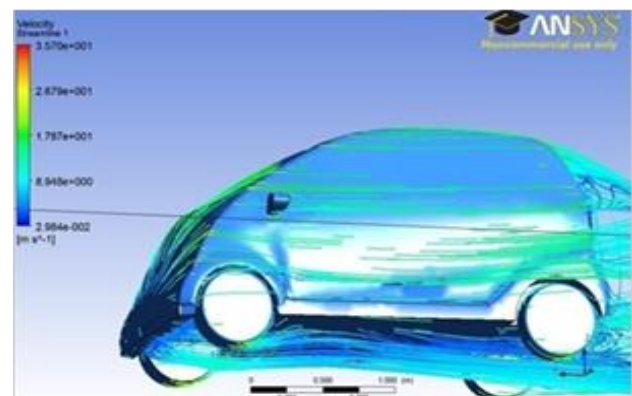
The various design modifications that were analyzed and are detailed in following sections with four approaches of design modifications at the rear end of the model.

**6.1 Air Dam**

An air dam reduces the air-flow below the vehicle while creating an additional down force at the top. A well-designed front air dam will keep the nose steady and pointed at the ground in high speed driving. This would give the driver more confidence to control the vehicle in sustained high speed driving environments. In the world of racing, an increment of a difference in aerodynamic tuning can spell the difference between victory and defeat.

The design shown in fig.3 resulted in a more streamlined flow of air underneath the vehicle, but it increased the drag force to 160.351 N. Another conjecture drawn was that the majority of the drag force on the vehicle would be due to the wake region behind the car and not mainly due to frontal air impact.

**6.2 Rear Spoiler**



**Fig. 3:** Airflow around the car with the air dam

The faster the vehicle moves, the more drag, which requires more power and fuel to maintain the speed. The next improvisation was made by introducing spoiler's as shown in fig. 4 purpose was to reduce drag. Any added surface in airflow adds to the drag. However, good spoiler designs can reduce the overall drag of your vehicle. Hence, the purpose of an automotive spoiler is to disrupt, or spoil, the airflow over the car to improve performance. This design reduced the drag force to 144.322N.

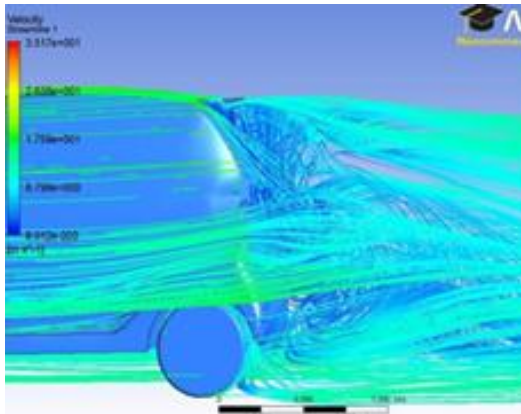


Fig. 4: Airflow with rear spoiler installed

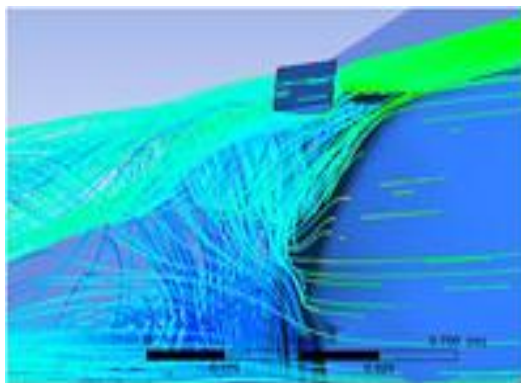


Fig 5. Zoomed in view of airflow with rear spoiler

### 6.3 Rear Protrusion

A rear protrusion was designed as shown in the fig. 6 is the third approach of improvisation.

### 6.4 Rear Screen

The best design modification came in the form of a rear screen which is the fourth approach of improvisation as shown in fig. 7. The screen was 5mm in thickness and at different distances from the car rear were analysed numerically. The zoomed view of velocity lines over the fourth approach is shown in fig 8.

The objective which is to reduce effect of aerodynamic drag is achieved with the fourth method of improvisation, which is for the sake of vortex that originates between vehicle model's rear wall and screen; the vortex enables no-separation flow of external air flux. This results in bottom trace narrowing and thus reducing of base drag for the sake of pressure increase.

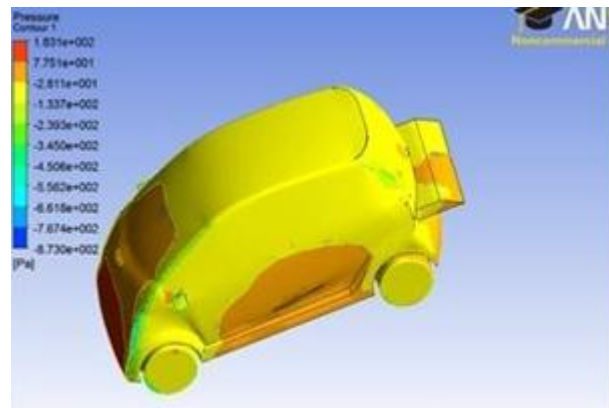


Fig. 6: Pressure contour for car with rear protrusion

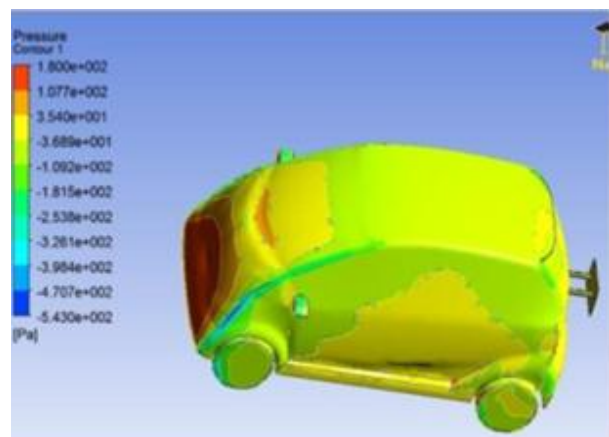


Fig. 7: Pressure distribution with rear screen

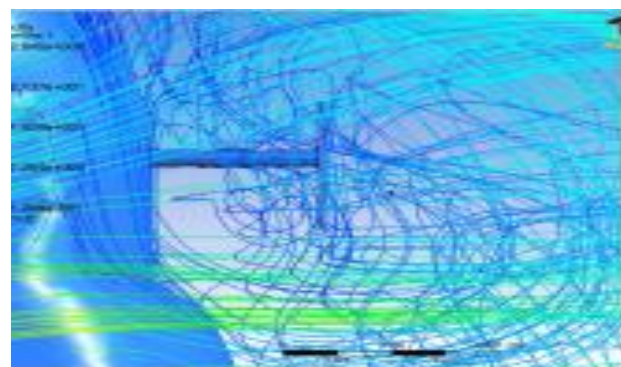


Fig 8: Zoomed in view of rear screen.

## 7. Results and Discussion

Reynolds Averaged Navier-stokes equations have been employed for numerical simulations to estimate the variation of drag data on different design of the vehicle. The numerical simulation reports the drag value of 147.38 N. Finally the drag co-efficient is calculated as 0.392.

In this paper the computational results for the two of the below mentioned cases are presented:

- Case I: results for the car model without the rear screen
- Case II: results for the car model with the rear screen.



Drag co-efficient for the above two cases and the reduction rate are presented in the table 2.

**Table 2** Reduction rate of drag co-efficient with improvisation

Case	Drag Co-efficient	Reduction Rate
Model I	0.392	
Model II	0.336	14.28%

Considering the fact that as much as 65 % of the power required for ground vehicles to travel on a highway at 60 km/hr is consumed to overcome the aerodynamic drag, the power consumption is given by

$$F_d \times \frac{V}{0.65} = \frac{C_d \times \rho \times A \times V^3}{1.3} \tag{1}$$

The relation between the power and fuel economy is

$$KML = \frac{1.3}{BSFC \times C_d \times \rho \times A \times V^2} \tag{2}$$

Fuel economy is the distance travelled per unit volume of fuel used and unit is kilometres per litre (km/L). BSFC is an index of fuel efficiency. The higher the value, the more economical the vehicle is. For both the cases as the car runs at constant speed (60 km/hr) and assuming steady flow, expression (4.2) can be approximated to

$$KML \propto \frac{1.3}{C_d} \tag{3}$$

As per the above approximation the following values as presented in table 3 were obtained for the fuel economy for the two cases, with a reduction of 14.28%.

**Table 3:** Power Consumption for different cases

Model	Power Consumption (kW)	KML
Case I	3.776	3.316
Case II	3.237	3.869

The model was analyzed using the same boundary conditions as before and at the same speed. The following table shows how the drag coefficient varies with the distance of the screen from the car.

The table 4 shows that there is an optimum gap for the minimum drag coefficient. But, in all cases the drag value is reducing. It is worth mentioning that for this vehicle configuration it was not possible to form stable oval vortex in the clearance between screen and model's rear wall, which could provide more aerodynamic drag reducing. For the above shown design a drag force of 126.49 N was obtained from ANSYS CFX which gives a Cd value of 0.336.

**Table 4** Cd v/s Distance of rear screen from car body

Gap(mm)	Cd
150	0.351
240	0.336
300	0.346
Without Rear Screen	0.392

### 8. Computational Validation

Hence, we carried out the following methods for effective

validation of our software results. Refining Mesh (ANSYS Software Evaluation) In this method, we are comparing drag force values obtained in ANSYS - CFX workbench for different mesh qualities (4, 00,000 and 12, 00,000 elements)

**Table 5** Comparison between Drag Force values of Coarse & Fine Mesh.

Inlet Velocity (m/sec)	Force (4,00,000)	Force (12,00,000)	Percentage error
0.412	0.0881122	0.0910487	3.22
0.748	0.282836	0.293997	3.7
1.014	0.521487	0.540721	3.5
1.269	0.82214	0.826000	0.4

### 9. Conclusion

Computation of drag force over the modified model showing drag reduction of 14.28% at a speed of 60 km/h with the drag coefficient reduced to 0.336 from 0.392, thereby reducing the fuel consumption by about approximately 14.28%, which means, there is a saving of 2.14L of petrol for every full tank refill and observations are validated using wind tunnel testing on a 1:25 scaled model using momentum deficit method.

### References

Xu-xia Hu, Eric T.T. Wong.(2011), A Numerical Study On Rear-spoiler Of Passenger Vehicle, World Academy of Science, Engineering and Technology, 57, 576-582 .  
 Shengbo Eben Li and Hueli Peng.(2011) Strategies to Minimize Fuel Consumption of Passenger Cars during Car-Following Scenarios, American Control Conference on O'Farrell Street, San Francisco, CA, USA pp 2107-2112.  
 Rizal E. M. Nasir.(2012), Aerodynamics of ARTeC's PEC 2011 EMO-C CarScience Direct Procedia Engineering 41, 1775 – 1780.  
 Masaru KOIKE.(2004), Research on Aerodynamic Drag Reduction by Vortex Generators, Mitsubishi motors Technical Review, No.16.  
 Justin Morden(2012), Aerodynamic analysis and testing of a Formula Renault race vehicle using analytical and computational methods validated by real world test data, Swansea Metropolitan University.  
 M. M. Islam and M. Mamun.(2010),Computational drag analysis over a car body”, Proceedings of MARTEC 2010, The International Conference on Marine Technology, BUET, Dhaka, Bangladesh, 155-158.  
 Aniket A Kulkarni, S V Satish and V Saravanan.(2012), Ansys of flow of convertible” Computer-Aided Design & Applications, PACE,2, 69-75.  
 Wolf-Heinrich Hucho.(1998), Aerodynamics of road vehicles, IV edition, Publisher- SAE International.

### Contributors



**Mr. Manikandan M** Pursuing M. Tech - Astronomy & Space Engineering from Manipal Institute of Technology, Manipal, Karnataka. His research interests are: Aerodynamics, UAV, LTV and Propulsion. He has 2 conference publications and One Journal Publication to his credit



**Mr. Shiva Prasad U**, obtained his M. Tech (Aerospace Engineering) from Institute of Aeronautical Engineering, Hyderabad, A.P. Presently, he is working as Asst. Prof. in the Dept. of Aeronautical Engineering, Manipal Institute of Technology, Manipal University (MU-MIT), Manipal and also reviewer for some international Journal. His research interests include: CFD, Hypersonic Aerodynamics, Propulsion and Combustion. He has One Journal & Six International conference publications to his credit.



**Mr Gunda Shivakrishna** studying M.Tech in the aerospace engineering from MLR institution technology ,he was presently working as Asst. Professor in Vardhaman college of engineering (Autonomous),Shamshabad, A.P, his research interests in ANSYS,CFD Aerospace structures Aerospace Propulsion.