Mohd. Rashaduddin et al. 2017, Volume 5 Issue 3
ISSN (Online): 2348-4098
ISSN (Print): 2395-4752

International Journal of Science, Engineering and Technology
An Open Access Journal

A Study on Airflow over a Car

[1]Mohd. Rashaduddin, [2]Ahmed Waheedullah

Abstract

The model of a car is made without spoiler and with roof box and its testing is done in wind tunnel and the results are verified. ANSYS software is used to get lift and drag forces at different velocities. In this paper an ideal roof box is also discussed. The objective is to simulate a car model to increase its efficiency in terms of speed, balancing and fuel consumption by managing drag and downforce.

Keywords - Car model; Spoiler; Roof Box; ANSYS; Wind tunnel; Aerodynamical parameters.

Introduction

In aerodynamics one discusses the flow of air inside and around the body. It can also be called as fluid dynamics as air is a thin fluid. Airflow impacts on speed, mileage and handling. Hence to build the best car it is necessary to study the airflow around and through the body [1- 4].

Car is one of the important means of transport. The space available in the boot is not enough for the luggage. Due to this problem normally people use the roof to put extra luggage. When a user installs a roof box on the top of the car, weight increases resulting increase of fuel consumption [5,6].

Fuel consumption is one of the most important factors to choose one vehicle than other. Hence installing a roof box should be done with such specifications that impacts should be decreased. In this paper an ideal roof box is also discussed.

In the present study, an attempt is made to develop a car model by minimizing the drag force and by increasing the down force (negative lift) which helps to stabilize the vehicle and which will lead to decrease in fuel consumption. The objective is to simulate a car model to increase its efficiency in terms of speed, balancing and fuel consumption by managing drag and downforce. Moreover how to install a roof box on the car?

Methodology

1. The physical boundaries are defined for the given problem.

2. The volume occupied with the fluid is divided by discrete cells and meshing is done.

3. Boundary condition is defined and involves specific fluid behavior and properties at boundaries of the problem.

4. The simulation is started and the equations are solved at steady state.

5. Finally a postprocessor is used for analysis and visualization of the results.

Aerodynamic Principles

Drag

A car requires energy to move through air which is used to overcome a force called Drag. Drag is comprised primarily of three forces:

1. Frontal Pressure occurs when tiny molecules of air hit the front of the car and is forced away to make room for other molecules to hit it. It's the effect created by a vehicle body pushing air out of the way.
2. Rear vacuum, or the effect created by air not being able to fill the hole left by the vehicle body.
3. Boundary layer or the effect of friction created by slow moving air at the surface of the vehicle body.

Between these three forces one can describe most of the interactions of the airflow with a vehicle body.

Frontal Pressure

Frontal pressure is caused by the air attempting to flow around the front of the vehicle. It is a form of drag. The molecules which approach the car compress the car due to which air pressure increases.

On the other hand the air molecules moving along the sides of the car are at atmospheric pressure which is lower than the pressure at the front side.

The vehicle tries to push air molecules. The compressed air molecules move from high pressure zone to low pressure zone.

Rear Vacuum

Rear suction occurs when an empty pocket of air is created in the back of the car resulting in a vacuum that tries to pull the car backward (Fig. 1.).

Rear vacuum is caused by the "hole" left in the air as a vehicle passes through it. At a certain speed the space behind the car's rear window and trunk is "empty" or like a vacuum. These empty areas are the result of the air molecules not being able to fill the hole as quickly as the car can make it. The air molecules attempt to fill in to this area, but the car is always one step ahead, and as a result, a continuous vacuum sucks in the opposite direction of the car. This inability to fill the hole left by the car is technically called Flow detachment.

Flow detachment applies only to the "rear vacuum" portion of the drag forces and has a greater and greater negative effect as vehicle speed increases. In fact, the drag increase with the square of the vehicle speed, so more and more horsepower is needed to push a vehicle through the air as its speed rises.

If the speed of the car increases it is essential to control areas of flow detachment. It will be good if the air molecules follow contours of a car's bodywork.

The force created by the rear vacuum exceeds that created by frontal pressure, so there is very good reason to minimize the scale of the vacuum created at the rear of the vehicle.

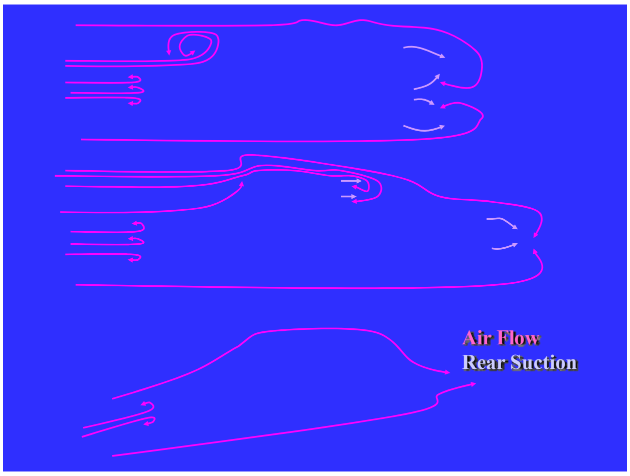


Fig.1. Air flow and Rear Section

Turbulence

The detachment of airflow from the vehicle creates turbulence. When the flow detaches it becomes very turbulent when compared to front smooth airflow. The air flow detaches from the flat side of the mirror impacts on the back of the car.

Lift/ Downforce

When air passes over a flat surface then it causes the car to lift. Every object travelling through air faces lifting or downforce situation. Downforce is due to high pressure in curved surfaces. It pushes the car downwards. Normal cars create lift because the shape of the car creates a low pressure area above itself.

According to Bernoulli's principle, for a given volume of air, the higher the velocity the air molecules are travelling, the lower the pressure becomes. Likewise, for a given volume of air, the lower the velocity of the air molecules, the higher the pressure becomes. This applies to air in motion across a still body, or to a vehicle in motion, moving through relatively still air.

Drag Coefficient

Drag coefficient is a way to get the idea how the car is slippery or stable. To compare the drag produced by different cars a value known as drag coefficient was created. The drag coefficient of any car can be measured by wind tunnel data.

One can get required drag coefficient if the car has following properties:

1. If the nose/grill is smaller in size then frontal pressure will be less.
2. If the ground clearance is less then air flow under the car will be less.
3. If the car has a steeply raked windshield then frontal pressure can be decreased (Fig. 2.)..

Fig.2. A car with steeply raked windshield

4. If the car have fastback style then air flow will be (stay) attached. A fastback is a car body style whose roofline slopes continuously down at the back (Fig. 3.)

Fig.3. A fastback style car

5. If the tail is converging then it keeps the air flow attached and minimizes the area against which flow detachment occurs.

The best car is one whose C_d is around 0.28.

Reynolds Number

Reynolds number is a dimensionless number that gives a measure of the ratio of inertial forces to viscous forces for given flow conditions. The Reynolds number is denoted by Re. It is an important parameter which describes the flow characteristics.

To calculate Reynolds number we can use

$$Re = \frac{\rho v l}{\mu} = \frac{v l}{\nu}$$

where V: velocity; l: characteristic length, the chord width of airfoil; ρ: Density of fluid; μ: Dynamic viscosity; v: kinematic viscosity; Kinematic viscosity is the ratio of dynamic viscosity to density.

Reynold number of a typical car is calculated and data is present in Table1.

Table 1 – Data on Reynold Number of a car

ρ	V (km/hr)	V (m/s)	l	μ	Re = ρvl/μ
1177	10	2.78	1.66	1.846	2942
1177	20	5.55	1.66	1.846	5874
1177	30	8.33	1.66	1.846	8816
1177	40	11.11	1.66	1.846	11758
1177	50	13.89	1.66	1.846	14701
1177	60	16.66	1.66	1.846	17633
1177	70	19.44	1.66	1.846	20575
1177	80	22.22	1.66	1.846	23517
1177	90	25	1.66	1.846	26460
1177	100	27.77	1.66	1.846	29391
1177	110	30.55	1.66	1.846	32334
1177	120	33.33	1.66	1.846	35276
1177	130	36.11	1.66	1.846	38219
1177	140	38.88	1.66	1.846	41150
1177	150	41.66	1.66	1.846	44093
1177	160	44.44	1.66	1.846	47035
1177	170	47.22	1.66	1.846	49978
1177	180	50	1.66	1.846	52920
1177	190	52.77	1.66	1.846	55852
1177	200	55.55	1.66	1.846	58794
1177	210	58.33	1.66	1.846	61736
1177	220	61.11	1.66	1.846	64679
1177	230	63.88	1.66	1.846	67611
1177	240	66.66	1.66	1.846	70553
1177	250	69.44	1.66	1.846	73495
1177	260	72.22	1.66	1.846	76438
1177	270	75	1.66	1.846	79380
1177	280	77.77	1.66	1.846	82312
1177	290	80.55	1.66	1.846	85254
1177	300	83.33	1.66	1.846	88197

Car in the absence of spoiler

A car model is created by using the following construction points and model is designed by using ANSYS software. Construction points used to construct the car model are as follows:

Table 2 – The x and y coordinates of Construction points

X - axis	Y - axis
0 .000	0.075
0 .000	0.113
0.025	0.063
0.025	0.119
0.025	0.176
0.189	0.057
0.189	0.214
0.302	0.050
0.302	0.220
0.440	0.050
0.440	0.289
0.566	0.050
0.566	0.302
0.755	0.057
0.755	0.277
0.818	0.214
0.956	0.069
0.975	0.082
0.975	0.126
0.975	0.182
1.000	0.082
1.000	0.113

The car model is constructed and is shown in Fig. 4.

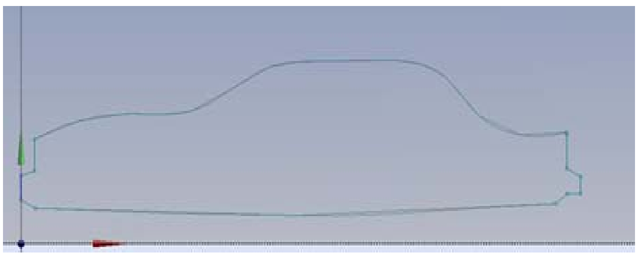


Fig.4. Constructed car model

Ideal Car

An ideal car is one which has following properties:

1. Drag coefficient should be less

2. Turbulence should be less.(Avoid flat upright surfaces in the front of the car to reduce turbulence)

3. Rear suction should be minimized.(Avoid flat upright surfaces in the back of the car to reduce rear suction)

4. Downforce should be high so as to balance lift.

5. Shield as much air at the bottom of car to maintain traction.

6. Keep the surface area exposed to oncoming air to a minimum.

7. Avoid unnecessary additions that add weight to the car.

8. The lighter is car the more power will be used.

9. Make sure that drill holes are straight and accurate to reduce extra forces acting.

10. Painting a car will reduce aerodynamic friction.

11. Axel design is important to reduce friction.

12. Lubrication of the axels can reduce friction.

13. If the contact of the wheels with the ground is less then friction will be less.

Creating computational domain

The computational domain means airflow in following sections which is created by using given boundary points.

Table 3 – Boundary points

X	Y
-8	0
-8	3
18	0
18	3

To create Car Model

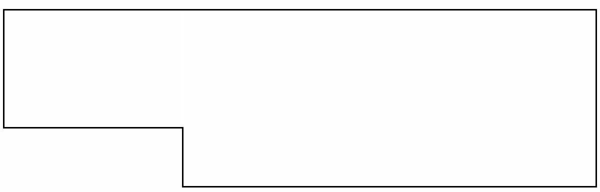
Step 1: Create 3D Geometry in Computational Domain

Generate the car geometry form the given construction point and then suppressed onto the 3D air domain in design model (Fig. 5.). It is to be assumed that the car is moving forward with a constant velocity 'U' through the air. But this is interpreted in simulation, if the air is moving towards the car with the same velocity but the opposite direction the car is stationary. The entire computational domain is done in the air domain. Therefore, the car is suppressed in the fluid domain and suitable boundary condition is assigned. The surface of the car body: inlet, sky, side plane, symmetry plane, ground, outlet and car body.

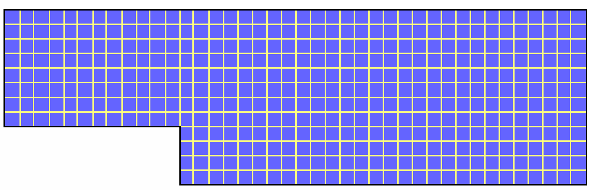
Step 2: Meshing the Model

The partial differential equations that govern fluid flow are not usually amendable to analytical solutions, except for very simple cases. Therefore, in order to analyze fluid flows, flow domains are split into smaller sub domains. The governing equations are then discretized and solved inside each of these sub domains. Typically, one of three methods is used to solve the approximate version of the system of equations: finite volumes, finite elements, or finite differences. Care must be taken to ensure proper continuity of solution across the common interfaces between two sub domains, so that the approximate solutions inside various portions can be put together to give a complete picture of fluid flow in the entire domain. The sub domains are often called elements or cells, and the collection of all elements or cells is called a mesh or grid. The origin of the term mesh (or grid) goes back to early days of CFD, when most analyses were 2D in nature. For 2D analyses, a domain split into elements resembles a wire mesh, hence the name.

Example of a 2D analyses domain and its mesh are shown in pictures below (Fig. 6.).



(a)



(b)

Fig. 6.Meshing Model (a) 2D analyses domain, (b) A domain is split into elements resembles a wire mesh

The process of obtaining an appropriate mesh (or grid) is termed mesh generation (or grid generation), and has long been considered a bottleneck in the analysis process due to the lack of a fully automatic mesh generation procedure. Specialized software programs have been developed for the purpose of mesh and grid generation, and access to a good software package and expertise in using this software are vital to the success of a modeling effort.

Meshing is one of the important steps for simulation in ANSYS. A suitable mesh will give exact and accurate solution whereas wrong meshing will give inaccurate or wrong answers. In order to know the exact turbulence the mesh should be grid independently and also it should be fine as much as possible especially at the boundary layer. On the other hand, greater number of mesh element in the car model which leads to finer meshing and this will increase the computational cost. Therefore, selecting mesh element size and applying suitable inflation on boundary layer will optimize in the computational cost and accurate results. Turbulence can be properly obtained when the mesh is done independent and fine including boundary layers.

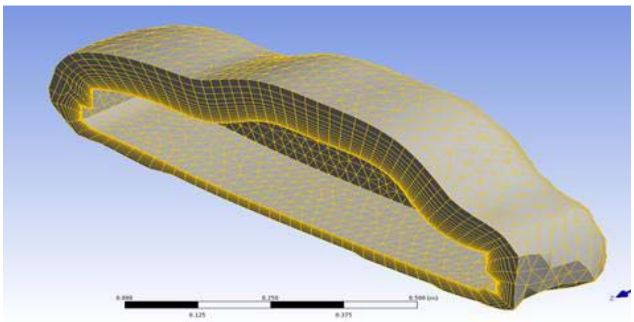


Fig. 6. Boundary layer Mesh

Step 3: Boundary layer mesh

In the ANSYS CFX, the method provided for turbulence models involving the boundary layer simulation (Fig. 7.). To know all the effect in the boundary layer of the car body, it is important to use inflation layer into the boundary layer with finer mesh. In the turbulence model, the first layer thickness setting is useful to save time in meshing because the value used by constant inflation layer.

The quality of the mesh is determined by the shape of the individual cells. The key factor that affects the quality of the cell is skewness, aspect ratio, angle between the adjacent element of the cells and determinants. Skewness is primary quality measure for a mesh. It determines how close to idea a face or cell is. In single car model the skewness average of k-epsilon model is 0.26648 and k-omega is 0.26611. The mesh element models have the shape to meet the requirements.

Mesh element requirements

Triangular: 45< θ< 135; Rectangular: 1< ; Ratio of adjacent side: < 5. The shape of the mesh element is qualified for the requirements shown in the above figures. In computational Fluid Dynamics k-omega (k-ω) turbulence model is a two equation model which is used in place of Reynolds-averaged Navier-Stokes equation. The model tries to say turbulence by 2 partial Differential equations for two variables k and ω.

The eddy viscosity ν_T, as needed in the RANS equations, is given by: $\nu_T = k/\omega$, while the evolution of k and ω is modeled as:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_j k)}{\partial x_j} = \rho P - \beta^* \rho w k + \frac{\partial}{\partial x_j}\left[\left(\mu + \sigma_k \frac{\rho k}{w}\right) \frac{\partial}{\partial x_j}\right]$$ where $p = \tau_{ij}\frac{\partial u_i}{\partial x_j}$

$$\frac{\partial(\rho\omega)}{\partial t} + \frac{\partial(\rho u_j \omega)}{\partial x_j} = \frac{\gamma\omega}{k}P - \beta\rho\omega^2 + \frac{\partial}{\partial x_j}\left[\left(\mu + \sigma_\omega \frac{\rho k}{\omega}\right)\frac{\partial_\omega}{\partial x_j}\right] + \frac{\rho\sigma_d}{\omega}\cdot\frac{\partial k}{\partial x_j}\cdot\frac{\partial_\omega}{\partial x_j}$$

Step 4: Boundary Conditions

1. The car is simulated to the different velocities.
2. The car is assumed to be fixed.
3. The outlet boundary condition is set to pressure outlet with the gauge pressure of 0pa.
4. The car contour, the top and the bottom of the wind tunnel are set as walls.
5. The ground moves in the opposite direction at the same speed. Ground is set as wall
6. The density of air is set as 1.185 kg/m^{-3}
7. The dynamic viscosity of air is 1.831 x 10^{-5} kg/ms.
8. Relative pressure of sky and side plane is 0; average static pressure of outlet is 0.
9. Car body is set as wall.

Results Analysis and Discussion

In the car model, the high pressure at the front of the vehicle is due to the flat and projected front portion. This can be reduced by making the design more streamlined. The pressure level varies behind the rear windshield. The vehicle turbulence air is considered as a fluid medium through which our current model moves at constant speed U. There are many types of turbulence produced on roadways.

Lift and Drag Coefficient of Car Model without Spoiler:

Drag coefficient = $C_d = \frac{2F_d}{\rho V^2 A}$ (1)

Where $F_d = \frac{1}{2}\rho v^2 C_d A$, the Drag force, ρ: air density, V: free stream velocity, v: Viscosity

A: the frontal area of the vehicle.

Lift can be manipulated to enhance the performance of a race car and decrease lap times. Lift is the force that acts on a vehicle normal to the road surface that the vehicle rides on.

Lift coefficient $C_L = \frac{2L}{\rho V^2 S}$ (2)

Where

ρ: air density; V: velocity; A: area of the top surface of the vehicle;

The forces and coefficient of the car model are presented in Table 4.

Table 4 - Data on Aerodynamic parameters and, forcs and thheir coefficients

Air velocity (km/hr)	160	120	80
Frontal area (km/hr)	1774	1774	1774 m^2
Lift force F_l (N)	64442	35842	15885
Drag force $F_d = \frac{1}{2}\rho v^2 C_d A$ (N)	79173	44589	20008
Lift coefficient $C_L = \frac{2L}{\rho V^2 S}$	0.298	0.295	0.294
Drag coefficient C_d	0.367	0.367	0.370

Roof box on the car

Roof box is one of the external attachments which are supported over roof. Various types of roof boxes are designed by different companies as per customer demands and requirements.

The main types of boxes are short wide, long wide, medium, wide and narrow types of boxes. According to the size, the box can be used to carry objects and equipments.

The safety of the passenger is considered a major factor. Placing excess luggage in the rear side of a car without any safety will lead to dangerous situation. To eliminate this situation, a roof box is essential. The position of the roof box should be arranged such that it will not contact the deck lid of the car in its open position. Even if the vehicle consumes 5-10% excess fuel, the journey will be safer and more comfortable.

The lower side of the box was designed such a way that it can be mounted on the roof of the car. The first and foremost factor which leads to the selection of roof box is its external appearance. Style, shape and color are the leading factors which control the external appearance. Quality of roof box is another factor to be considered which includes durability and reliability. It should be strong enough to withstand aerodynamic drag and shaking while travelling.

The roof boxes are totally scratch resistant and should not lose their quality for a long time. The box should be stable, even if the speed exceeds motorway limits. The overall height of the vehicle, manufacturability, easiness in mounting and removing, safety, directional stability and wind noise are some factors to be considered. Short wide roof boxes are designed mainly for carrying luggage and camping items. Long wide box can carry many types of baggage. Weight of roof box should be directly proportional to the thickness. The surface should be water repellent in nature. Overall cost should not be too high.

Carbon Fibre shows excellent characteristics in manufacturing roof box. But, the final product is too expensive. ABS is another material which is less expensive and it has moderate performance when compared to carbon fibre. Most of the manufacturers and customers prefer this material.

The following objectives should be achieved for the successful fulfillment of this project.

1. A 3D roof box should be modeled for CFD analysis.

2. CFD analysis of roof box is conducted over cars.

3. It should be investigated how the position can be varied and how to design roof box over the car.

All these influences the aerodynamic performance and affect the overall power consumption.

1. The existing design should be modified to achieve minimum drag.

2. The influence of roof box should be investigated by analyzing the pressure variation around the car and box.

In the present study a commercial roof box and its effects are analyzed when the user installs the roof box. The increase of drag coefficient is notable and adding an increase of weight makes an increase of fuel consumption and power needed to beat the resistance forces in first attempt. To obtain these values, two types of analysis are made: One in 2 dimensions and the other one in 3 dimensions. The first is only to get an approximation of the values that we can obtain and if we were going in the correct way. The second analyze is a complex calculation and we get an accurate results of the values that we are interested (drag coefficient, pressure, speed, turbulence and Reynolds).

To solve this problem, a possible redesign is created and a modification of the position of the roof box is done. There are three possibilities:

1. The position of the roof box is modified to improve the fluid flow and reduce the air resistance.

2. A roof box is created such that it totally fits the vehicle that we analyze with the problem of having a non-universal roof box.

3. The last possibility is redesigning of the roof box and obtaining a universal roof box trying to get better values comparing with the commercial roof box.

In the 3 dimensional analyses, the values obtained are more reliable as the model is more realistic and the mesh has a high density number of elements. Although the increase of fuel consumption and power needed is considerable.

References

[1]. Anderson Jr, Computational Fluid Dynamics – The basics with Applications, McGraw-Hill Series in Mechanical Engineering Series, McGraw Hill, Inc, 1995.

[2]. Praveen Padagannavar and Manohara Bheemanna, Automotive Computational Fluid Dynamics Simulation of a car using ANSYS, Int. J. Mech. Engg.Tech., Vol. 7, No. 2(2016), pp. 91–104,

[3]. Aniruddha Patil, Siddharth Kshirsagar and Tejas parge, Study of front wing of formula 1 car using CFD, wwwijmerr.com., Vol. 3, No. 4(2014).

[4]. Mohammed Masood Karim, Hiroshi Matsui and Randall Guensler, A Mathematical model of wind flow, vehicle wake and pollution concentration in urban road, Trans. Res. D, Vol. 3, No. 3(1998), Elsevier Science Ltd., U. K.

[5]. Ivan Gonzalez Casal, Aerodynamic analysis and improvement of a roof box car, Master Thesis, Cracow University of Technology, Kraków, 2014.

[6]. Design and analysis of car body using CFD software, http://www.researchgate.net/ publication/277541041.

Author's details

[1]Department of Mathematics, Dravidian University, Kuppam – 517426 (A.P.), India, Email: rashadtelangana@gmail.com

[2]Lords Institute of Engineering and Technology, Hyderabad – 500008, India