ISSN XXXX XXXX © 2019 IJESC

IJESC

Research Article Volume 9 Issue No.3

Profile Drag Analysis of Sedan Car using Computational Fluid Dynamics

P.Chakravarthi[1], D.Adhishankaran[2], S.Ajith Kumar[3], S.Gaby Sachin[4], B.Giridharan[5]
Assistant Professor[1], UG Student[2, 3, 4, 5]
Department of Mechanical Engineering
KSR Institute for Engineering and Technology, Tiruchengode, Tamilnadu, India

Abstract:
One of the ongoing automotive technological developments is in the area of reduction of aerodynamic drag, which directly impacts the overall efficiency of the vehicle. This work mainly consists of determining the flow regime over vehicles and further evaluating the drag force and coefficient of drag using CFD analysis. SEDAN model passenger car is to be developed using Creo 3.0 and the analysis are carried out in CFD withvelocityof 40kmph. Further the analysis is extended to study the effect of adding a spoiler on the existing model.

Key words: SEDAN, Spoiler, Aerodynamic, Velocity, Drag Force.

1. INTRODUCTION

Aerodynamics is a branch of fluid dynamics concerned with studying the motion of air, particularly when it interacts with a moving object. Aerodynamics is also a subfield gas dynamic, with much theory shared with fluid dynamics. Aerodynamics is often used synonymously with gas dynamics, with the difference being that gas dynamics applies to all gases. Understanding the motion of air around an object enables the calculation of forces and moments acting on the object. Typical properties calculated for a flow field include velocity, pressure, density and temperature as a function of position and time. By defining a control volume around the flow field, equations for the conservation of mass, momentum, and energy can be defined and used to determining the properties. The use of aerodynamics through mathematical analysis, empirical approximation and wind tunnel experimentation form the scientific basis.

Figure.1.1Jelly Mold shape

Aerodynamics and its analysis are basically divided into two major sub-categories, namely the external and internal aerodynamics. External aerodynamics is the study of flow around solid objects of various shapes. Evaluating the lift and drag on an airplane, the shock waves that form in front of the nose of a rocket, or the flow of air over a wind turbine blade are examples of external aerodynamics. On the other hand, internal aerodynamics is the study of flow through passages in solid objects. For instance, internal aerodynamics encompasses the study of the airflow through a jet engine or through an air conditioning pipe and other internal flow conditions. The vehicle aerodynamic flow process is fall into three types (i) Flow of air around the vehicle. (ii) Flow of air through the vehicle body. (iii) Flow of air within the vehicle body.

1.1 AERODYNAMIC FLOW FIELD OF A CAR BODY

The streamline of an external flow around stationary vehicle. When the vehicle is moving at an undistributed velocity, the viscous effects in the fluid are restricted to a thin layer called boundary layer. Outside the boundary layer is the in viscid flow. This fluid flow imposes pressure force on the boundary layer. When the air reaches the rear part of the vehicle, the fluid gets detached. Within the boundary layer, the movement of the fluid is totally governed by the viscous effects of the fluid.

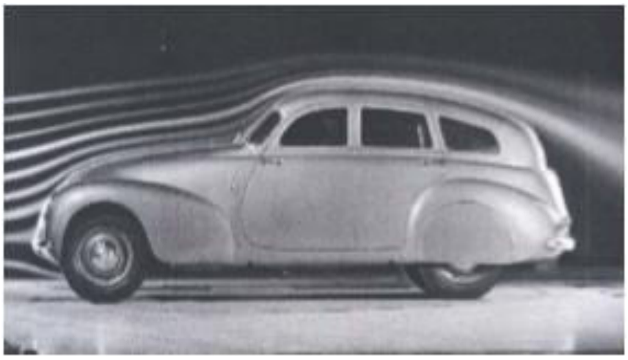


Figure.1.2 External Flow Field over the Car Body.

The boundary does not exist for the Reynolds Number which is lower than 104. The Reynolds number is dependent on the characteristic length of the vehicle, the kinematic viscosity and the speed of the vehicle. Apparently, the fluid moving around the vehicle is dependent on the shape of the vehicle and the Reynolds number. There is another important phenomenon which affects the flow of the car and the performance of the vehicle. This phenomenon is commonly known as 'Wake' of the vehicle. When the air moving over the vehicle is separated at the rear end, it leaves a large low pressure turbulent region behind the vehicle known as the wake. This wake contributes to the formation of pressure drag, which is eventually reduces the vehicle performance.

1.2 FORCES AND MOMENT ON VEHICLE

When the vehicle is moving at a considerable speed, the air passing over it imposes various forces and moment on the vehicle. The detailed sketch view of the various forces and moment acting on the vehicle body.

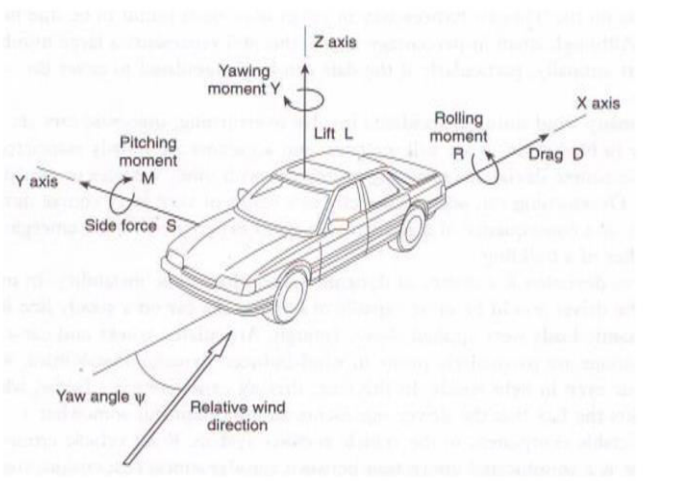


Figure.1.3 Aerodynamic Forces and Moments Acting On a Vehicle

The vertical force acting on the body causes the vehicle to get lifted in air as applied in the positive direction, whereas it can result in excessive wheel down force if it is applied in negative direction. Engineers try to keep this value to a required limit to avoid excess down force or lift. Aerodynamic drag force is the force acting on the vehicle body resisting its forward motion. This force is an important force to be considered while designing the external body of the vehicle, since it covers about 65% of the total force acting on the complete body. Crosswinds produce a side force on a vehicle that acts at the middle of the wheelbase, and when the crosswinds do not act at the middle of the wheelbase a yawing moment is produced.

1.3 FACTORS CONTRIBUTING TO FLOW FIELD AROUND VEHICLE

The major factors which affects flow field around vehicle are the boundary layers, separation of flow field, friction drag and lastly the pressure drag.

Boundary layer: The Aerodynamics boundary layer was first defined by the Aerodynamic engineer 'Ludwig Prandtl' in the conference at Germany. This allows aerodynamicists to simplify the equations of fluid flow by dividing the flow field into two areas: one inside the boundary layer and the one outside the boundary layer. In this boundary layer around the vehicle, the viscosity is dominant and it plays a major role in drag of the vehicle. The viscosity is neglected in the fluid regions outside this boundary layer since it does not have significant effect on the solution. In the design of the body shape, the boundary layer is given high attention to reduce drag. There are two reasons why designers consider the boundary layer as a major factor in aerodynamic drag. The first is that the boundary layer adds to the effective thickness of the body, through the displacement thickness, hence increasing the pressure drag. The second reason is that the shear forces at the surface of the vehicle causes skin friction drag.

Separation: During the flow over the surface of the vehicle, there is a point when the change in velocity comes to stall and the fluid starts flowing in reverse direction. This phenomenon is called 'Separation' of the fluid flow. This is usually occurred at the rear part of the vehicle. This separation is highly dependent on the pressure distribution which is imposed by the outer layer of the flow. The turbulent boundary layer can withstand much higher pressure without separating as compared to laminar flow. This separation causes the flow to change its behavior behind the vehicle and thereby affect the flow field around the vehicle. This phenomenon is the major factor to be considered while studying the wake of the vehicle.

Pressure drag: The blunt bodies like large size vehicle show different drag characteristics. On the rear part of such vehicles, there is an extremely steep pressure gradient which leads to the separation of the flow separation in viscous flow. The front part of the flow field shows high pressure value, whereas on the rear part flow separates leading to a high suction in the area. As we integrate the force component created by such high change in pressure, the resultant is called as 'Pressure Drag'. This factor is affected by the height of the vehicle as well as the separation of the flow field.

Friction drag: Each wall surface or material has a distinct friction which resists the flow of fluids. Due to molecular friction, a stress acts on every surface of the vehicle. The integration of the corresponding force component in the free stream direction leads to a friction drag. If the separation does not occur, then friction drag is one of the main reasons to cause overall drag.

2.LITERATURE SURVEY

Dr.K.M. Parammasivam, Shankar Dr.G. Devaradjane Aerodynamic force plays an important role in vehicle performance and its stability when vehicle reaches higher speed. Nowadays the maximum speed of car has been increased above 180 km/hr. but at this speed the car has been greatly influenced by drag and lift forces. So the researchers are mainly focused in reduction of co-efficient of drag and lift in car model at higher speed. Even though the various techniques are found by researchers for improving vehicle performance and its stability still we are in need of further improvement So we are implementing vortex generator as a aerodynamic add on device at rear portion of vehicle. The various yaw angles and location of vortex generator are analyzed to obtain the efficient one to reduce the aerodynamic forces. An approximate outer profile of the typical sedan car body (Hyundai Elantra) which has a Co-efficient of drag value CD (0.35) has been generated in two configurations of with and without vortex generator by using solid modeling software and it has been analyzed using computational fluid dynamics (CFD) tool to reduce the aerodynamic drag and lift forces. Results show good improvement in reduction of above two forces by implementation of vortex generators on the car body.

Bhavini Bijlani has investigated both experimentally and analytically the aerodynamics of sedan and square-back cars, measuring drag coefficient and analyzed flow of air over the car body. 1:20 aluminum scale model of popular sedan and square-back cars where used in the experiments carried on a subsonic wind tunnel having test section of (30cm x 30cm x 100cm). The computational analysis was carried out in ANSYS CFX-13. A comparison of the values that were predicted by both experimental and computational methods confirms their reliability and recommends for further experimentation. The drag co-efficient which is evaluated for exterior profile of the Sedan and the Square-back is in the order of 0.38 and 0.66 that is acceptable. It was concluded that the sedan model is more aerodynamic than square-back model.

Praveen Padagannavar and Manohara Bheemanna In this paper, ANSYS CFX method is used to simulate a single car model with and without a spoiler and by using two types of mesh. The solution of the Reynolds average Naiver Stokes equations (RANS equations) has been achieved by using two models such as K-Epsilon and K –Omega Turbulence model will be analyzed. In this report, mesh quality, boundary layer and turbulent y+ value simulation has been thoroughly analyzed and solution for both the models has also been compared and discussed the results. We use the ANSYS software to determine the drag and lift forces at different turbulence kinetic energy variables k-Epsilon and K-Omega for the given vehicle domain. Further, the effects of aerodynamic are verified with and without the spoiler. The calculated drag coefficient for the baseline model of Land Rover Discovery agrees very well with the experimental data. It is clear that the use of a ventilation duct has a significant effect in reducing the aerodynamic drag coefficient.

3.DEVELOPMENT OF CAD MODEL

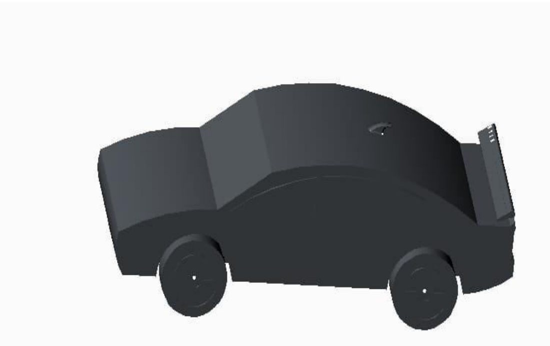


Figure. 3.1 CAD model development

4.DRAFTED MODEL

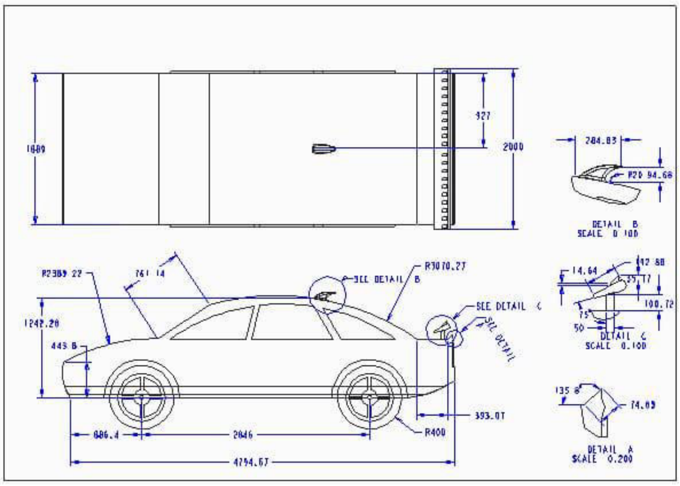


Figure.4.1 Model with dimensions

5.EXPERIEMENTAL ANALYSIS

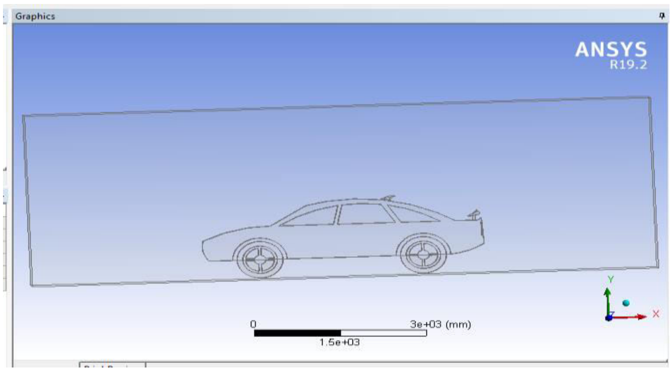


Figure.5.1 Boundary Condition

The boundary condition provided for the analysis are x=3m, y=2m, z=1m.A domain is then created to analyze the flow regime of the designed model

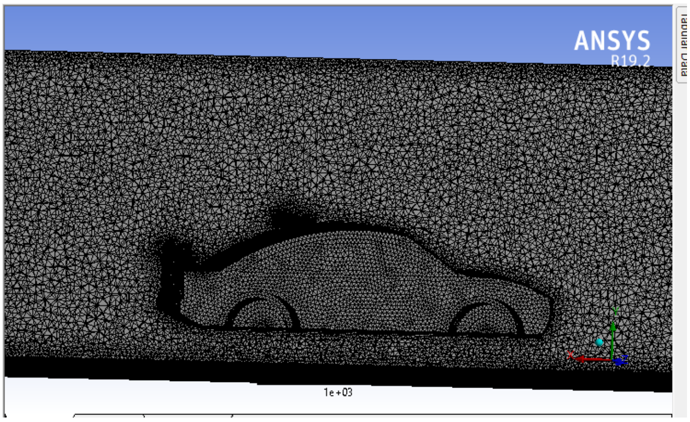


Figure. 5.2 Meshing

After providing the boundary conditions meshing of the model is done where the tetrahedral nodes generated are meshed together.

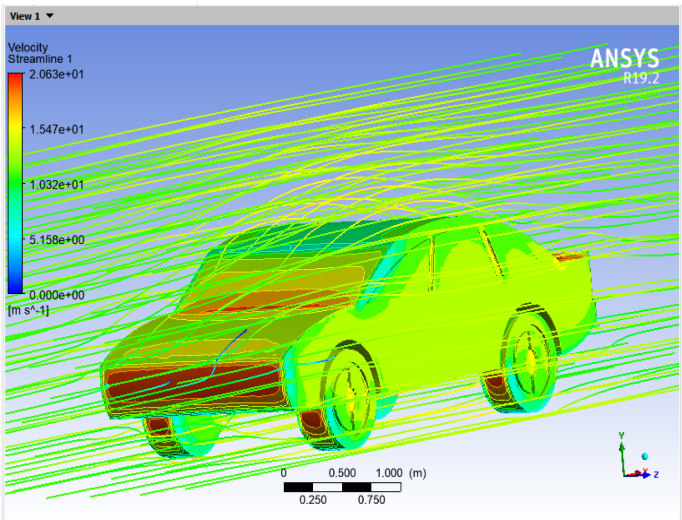


Figure.5.3 Velocity Flow Regime at 40km/h (without spoiler)

- The maximum velocity of air is 20.63 m/s (shown in red) and it decreases until it reaches 0 m/s(shown in blue)

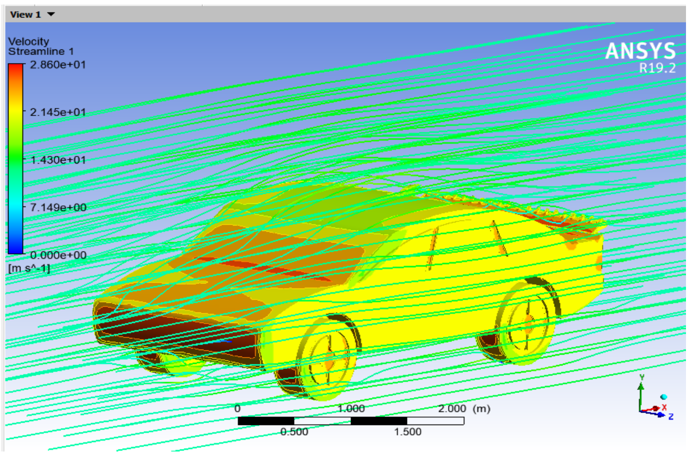


Figure. 5.4 Velocity flow regime at 40km/h(with spoiler)

- The maximum velocity of air is 28.60 m/s (shown in red) and it decreases until it reaches 0 m/s(shown in blue)

6. PRESSURE REPORT

1.Without spoiler

```
Forces - Direction Vector (1 0 0)
                          Forces (n)
Zone                      Pressure        Viscous         Total
car                       167.57405       9.5202803       177.09433
------------------------- --------------- --------------- -------------
Net                       167.57405       9.5202803       177.09433
```

2.With spoiler

```
Forces - Direction Vector (1 0 0)
                          Forces (n)
Zone                      Pressure        Viscous         Total
car                       159.19092       9.9177946       169.10871
------------------------- --------------- --------------- -------------
Net                       159.19092       9.9177946       169.10871
```

7.CALCULATIONS

To calculate the Drag CoefficientCd:

$$Cd = \frac{D}{\rho \frac{V^2}{2} A}$$

Where,

Drag D (N)
Density of air ρ(kg/m^3)
Velocity of air V(m/s)
Area of the domain A(m^2)
x=3m, y=2m, z=1m

1.without spoiler

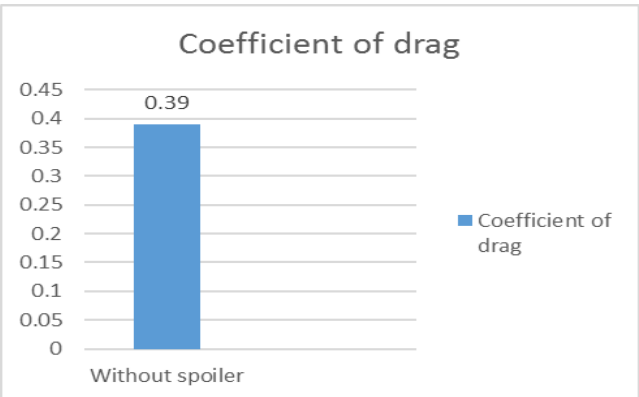
$$Cd = \frac{177.09}{1.22 \frac{11.11^2}{2} 6} = 0.39$$

2.with spoiler

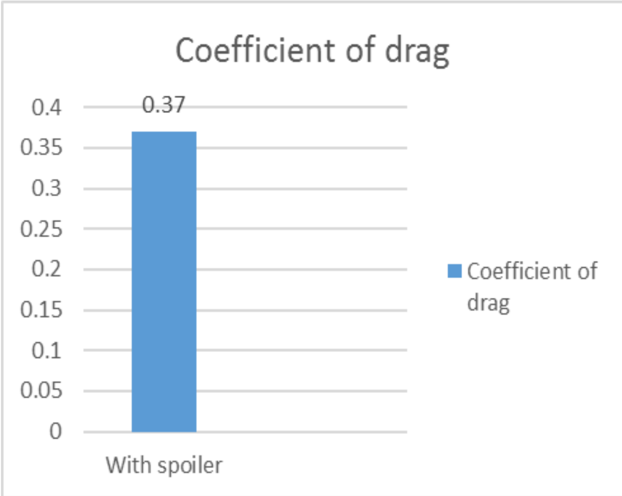
$$Cd = \frac{169.10}{1.22 \frac{11.11^2}{2} 6} = 0.37$$

8.RESULTS AND DISCUSSION

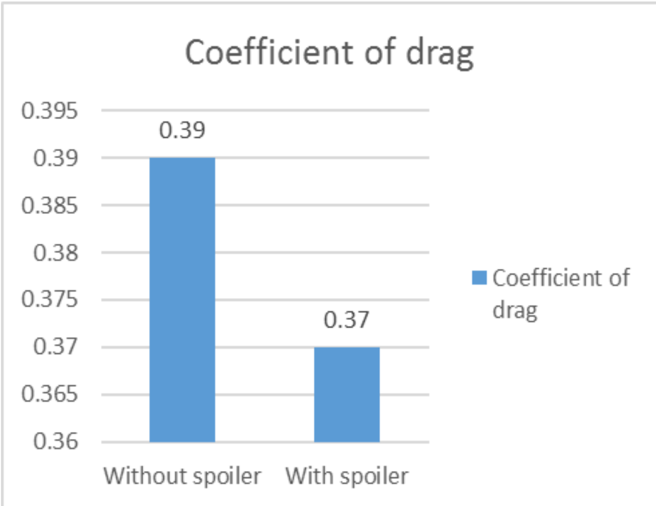
This section focuses on presenting the observations and findings gathered during the course of experiments. The data analysis provides the basis and justification for conclusion drawn in this work. Comparison between two types of models were carried out, one with spoiler and another without spoiler and their respective coefficient of drags are compared.



The graph shows the maximum value of coefficient of drag is 0.39 when vehicle is without spoiler



The graph shows the minimum value of coefficient of drag is 0.37 when vehicle is with spoiler



The graph shows the variation of coefficient of drag when vehicle is without spoiler and with spoiler

9.CONCLUSION

Thus the flow regime of SEDAN model passenger car is determined. And, the coefficient of drag and drag force for the model is analyzed for the velocity of 40kmph.Further, adding spoiler reduces drag force by 5% compared with the model without spoiler.

10. REFERENCES

[1]. Toukir Islam, S.M. Rakibul Hassan and Dr.M.Ali "Aerodynamic Drag of Racing Cars". Proceedings of the Global Engineering, Science and Technology Conference 2012 28-29 December 2012, Dhaka, Bangladesh.

[2]. Abdul Ghani "Design Optimization Of Aerodynamic Drag At The Rear Of Generic Passenger Cars Using Nurbs Representation". A Thesis Submitted in Partial Fulfilment of the Requirements for the Degree of Master of Applied Science in Mechanical Engineering Faculty of Engineering and Applied.

[3]. Alaman Altaf, Ashraf A. Omar and WaqarAsrar "Review Of Passive Drag Reduction Techniques For Bluff Road Vehicles". IIUM Engineering Journal, Vol. 15, No.1, 2014 Department Mechanical Engineering, Faculty of Engineering, International Islamic University Malaysia.

[4]. Bhavini Bijlani “Experimental And Computational Drag Analysis Of Sedan And Square-Back Car”. International Journal of Advanced Engineering Technology, Associate Professor Department of Mechanical Engineering, Government Engineering College.

[5]. C.N. Patil, Dr.K.M. Shashishekar,. KBalasubramanian and Dr.S.V. Subbaramaiah, “Aerodynamic Study and drag coefficient optimization of passenger vehicles”.

[6]. LankaTataRao, Katakam Satyanarayana, M.S.S. Srinivas Rao, T.V. Hanumanta Rao, S.V. UmamaheswaraRao, “stress analysis of 4stroke diesel engine piston “IJCESR ISSN (print): 2393-8374, (online): 2394-0697, volume-2, issue-2,2015.

[7]. K. Satyanarayana et al., Condition Monitoring and Detection of Unbalance Using Vibration Signature Analysis. International Journal of Recent Scientific Research Vol. 6, Issue, 8, pp.57965800, August, 2015.

[8]. D.Simhachalam, N. Indraja, M. Raja Roy “Experimental Evaluation of Mechanical Properties of Stainless Steel by TIG Welding at Weld Zone", International Journal of Engineering Trends and Technology (IJETT), V26(3),120-127 August 2015. ISSN:2231-5381.

[9].K.Satyanarayana,VinodhKumar Padala, T.V.Hanumantha Rao, S.V. Umamaheswararao "Variable Compression Ratio Diesel Engine Performance Analysis", International Journal of Engineering Trends and Technology (IJETT), V28(1),6-12 October 2015. ISSN:2231-5381.