International Journal of Fluids Engineering.
ISSN 0974-3138 Volume 6, Number 1 (2014), pp. 29-38

http://www.irphouse.com

Study of Flow over Car by Changing Different Parameters using Open Foam

L.D. Jathar[#1] and S.L. Borse[#2]

[#1]*Department of Mechanical Engineering, Rajarshi Shahu College of Engineering, Pune, India.*
[#3]*Department of Mechanical Engineering, Rajarshi Shahu College of Engineering, Pune, India.*
E-mail: [1] lax.jathar73@gmail.com, [2]sachinlb@yahoo.co.uk

Abstract

One of the major factor that determine fuel efficiency of a ground vehicle is its aerodynamic drag. Thus here interest was to study the aerodynamic performance of car geometry for Hatchback geometry and by changing their different parameters we can easily find out the Drag force and Lift force by using OpenFOAM software. This study involves different geometries with slightly different features, such as hood angle, wind screen angle, rear angle, and corner radius. In this project geometries are created in OpenFOAM itself by using blockMesh. Meshing is done by using snappyHexMesh. This study majorly contains 10 Hatchback geometries and their various contours are plotted at different speed range as well as their coefficient of drag and coefficient of lift is find out and one optimum geometry is find out from Hatchback.

Keywords: Open FOAM, Drag force, Lift force, blockMesh, snappyHexMesh

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 dynamics, with much theory shared with fluid dynamics. Aerodynamics is often used synonymously with gas dynamics, with the difference being that gas

Paper Code: 14430–IJFE

dynamics applies to all gases. Understanding the motion of air (often called a flow field) 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 solve for the properties. The use of aerodynamics through mathematical analysis, empirical approximation and wind tunnel experimentation form the scientific basis.

To save the energy and to protect the Global environment, fuel consumption reduction is a primary concern of the modern car manufacturers. Decreased resistance to forward motion allows higher speed for the same power output, or lower power output for the same speed. The shape is an important factor for drag reduction. To design an efficient shape of the car that will offer a low resistance to the forward motion, the most important functional requirement today is the low fuel consumption. The resistance, termed as the drag force (or the drag coefficient in non dimensional terms), is a strong function of the shape of the car. This suggests it is important how the fluid particles move about the car and how fast they move along their path.

This paper mainly concentrates on the external surface of the car geometries. Interest was to create the geometries by changing various parameter such as wind screen angle, front hood angle, rear hood angle, front corner radius etc. The evaluation of coefficient of drag, coefficient of lift, drag force, lift force can done by using OpenFOAM. software.

2. Methodology

The OpenFOAM (Open Field Operation and Manipulation) CFD toolbox is used to analyse the airflow around the the car geometries for both Hatchback and Sedan. OpenFOAM has extensive range of features to solve anything complex fluid flow involving chemical reactions, turbulence, and heat transfer to solid dynamics and electromagnetics.

The car geometries having different external parameters are created in OpenFOAM itself by using blockMesh. It is then converted into the stereolithogrphy (STL) file. STL file is then linked to snappyHexMesh for generation of mesh. potentialFoam was used to generate initial flow. By givinng initial boundary conditions calculations were done by using simpleFoam and finally postprocessing was done by using ParaFoam.

The velocity and pressure distribution over the geometries are visualized and discussed. Through the aerodynamic analysis a contour of pressure and velocity that impacts the car body is studied. Then accordingly coefficient of drag, coefficient of lift, drag and lift force are calculated. For the flow analysis the tyre portion is not taken into the consideration.

3. Mathematical Formulation

Drag Force (F_D)

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. The drag coefficient is dimensionless quantity that describes vehicle aerodynamic resistance and is useful tool for comparing different vehicle shapes.The Aerodynamic drag force is calculated by the following formula:

$$F_D = \frac{1}{2}\rho V^2 C_D A \quad (1)$$

where:
F_D=Drag Force
C_D=Drag Coefficient
A=Frontal Area Of The Vehicle
V=Wind Velocity
ρ=Air Density

Lift Force (F_L)

Lift force 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.

The formula usually used to define this force is written as:

$$F_L = \frac{1}{2}\rho V^2 C_L A \quad (2)$$

where:
F_L=Lift Force
C_L=Lift Coefficient
A=Frontal Area Of The Vehicle
V=Wind Velocity
ρ=Air Density

For hatch back geometry

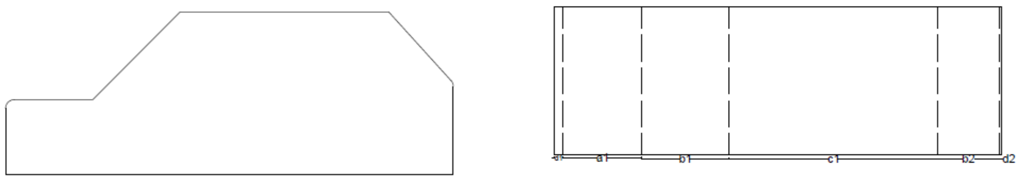


Fig. 1: Geometry of Hatchback.

Table 1: Parameters for Hatchback geometry

Case No.	d1	d2	Hood angle	Rear angle
1	0.05	0.001	0	0
2	0.1	0.001	5	0
3	0.15	0.001	10	0
4	0.2	0.001	15	0
5	0.25	0.001	20	0
6	0.05	0.001	0	5
7	0.1	0.001	5	10
8	0.15	0.001	10	15
9	0.2	0.001	15	20
10	0.25	0.001	20	25

Computational set up

Different cases were considered in order to achieve the given goals. However, the simulation setup is the same for all the cases and hence the procedure is same for all cases. The first step required to begin a new simulation is to input the actual mesh into the file structure of OpenFOAM .But in our project we are creating the geometry in OpenFOAM itself by using blockMesh. It is then converted into the stereolithogrphy (STL) file.

Domain specifications

The computational domain is designed to lead to a free flow with neglectable blockage, which essentially means a box that consist of a inlet, a outlet, two sides, a roof and ground surface. When the geometry was defined in the creation of the computational mesh, all faces of the domain were assigned names. The names of the inlet and outlet planes (at x= 0 and x= L) are front face and back face of domain as velocity inlet and pressure outlet respectively. The names of the planes at y=L, z=0, and z=L are outer wall as wall. The names of the model are car as a wall.

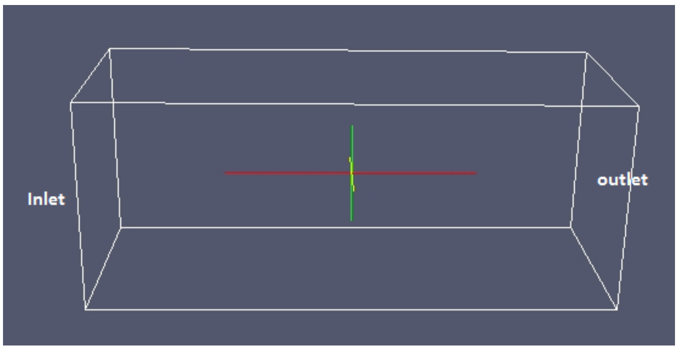


Fig. 2: Defining the Domain.

Mesh

Mesh is created using snappyhexmesh, OpenFOAM's native meshing tool. Utility snappyHexMesh is used to create high quality hexdominant meshes based on arbitrary geometry. It is controlled by parameters in the le snappyHexMeshDict. It can be executed in parallel. It preserves the feature edges, addition of wall layers. Details of mesh size is given in the following table.

Table 2: Details of mesh size.

No. of Points	No. of Cells	No. of Faces	No. of Internal Faces
1969583	1824778	5617183	5515085

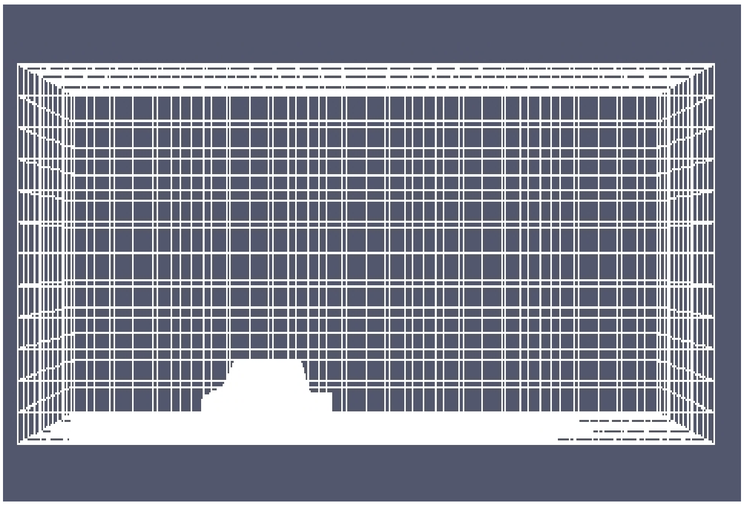


Fig. 3: Side view after Meshing.

Boundary conditions

The tests were taken at different operating speed range such as 60 km/hr, 70 km/hr, 80 km/hr, 90 km/hr,100 km/hr, 110 km/hr, 120 km/hr,130 km/hr. That means each geometry of hatchback can pass through these speed range. For example consider case 1 when it pass through 60km/hr, velocity of the inlet boundary condition is set with value 60km/hr (16.16m/s) with temperature of 300K (26.85^{0}C). The outlet boundary condition is set to pressure outlet with gauge pressure of 0 pa. The density of the air is set as 1.125kg/m^3 and the viscosity of the air is 1.7894×10^{-5} kg/(ms).

Turbulence Modeling

All of the simulations took turbulence into account with the k- ω-SST turbulence model. This model was used for its ω proven reliability in separation zones and its ability to blend a good freestream model to a good boundary layer model.

Turbulence Intensity

The turbulence intensity, *I*, is defined as the ratio of the root-mean-square of the velocity fluctuations, u`, to the mean free stream velocity, u.

$$I = \frac{u'}{u}. \quad (3)$$

For internal flows the value of turbulence intensity can be fairly high with values ranging from 1%–10% being appropriate at the inlet. The turbulence intensity at the core of a fully developed duct flow can be estimated as:

$$I = 0.16Re^{-1/8} \quad (4)$$

For external flows the value of turbulent intensity at the freestream can be as low as 0.05% depending on the flow characteristics. We are considering turbulence intensity as 0.02%.

About Turbulence Length Scale The turbulence length scale, *l*, is a physical quantity which represents the size of the large eddies in turbulent flows. Empirical relationship between the physical size of the obstruction (or characteristic length), *L*, and the size of the eddy, *l*, can be used to get an approximate length scale.

$$l = 0.07L \quad (5)$$

Turbulent Kinetic Energy(K) and Specific Dissipation Rate (ω) Required Turbulent Kinetic Energy(K) and Specific Dissipation Rate (ω) can br find out from the following equations and table no.5.2 gives the (K) and (ω) for our selected operating speed range.

$$\text{Turbulent Kinetic Energy (K)} = \frac{3}{2}(UI)^2 \quad (6)$$

$$\text{Specific Dissipation Rate } (\omega) = \frac{K^{1/2}}{C\mu^{1/4} \times l} \quad (7)$$

Table 2: Values of K and ω

Sr. No.	Speed in km/hr	Speed in m/sec	K	ω
1	60	16.16	0.1665	1.061
2	70	19.44	0.2268	1.2421
3	80	22.22	0.2962	1.4197
4	90	25	0.375	1.5972
5	100	27.27	0.4629	1.7747
6	110	30.55	0.56	1.9522
7	120	33.33	0.666	2.1296
8	130	36.11	0.7824	2.3071

4. Result and Discussion

Comparison of all Hatchback geometries at 100km/hr–In this comparison of all the 10 geometries of hatchback at speed of 100km/hr and there pressure contour, velocity contour, vector plot , front view, side view, and top view are pressure variation was observed.

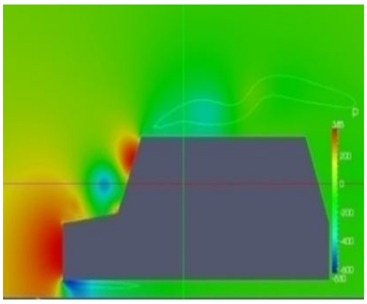


Fig. 4: Case 1 Pressure contour

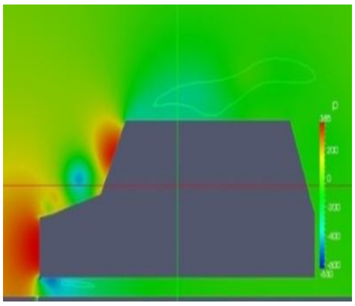


Fig. 5: Case 1 Pressure contour

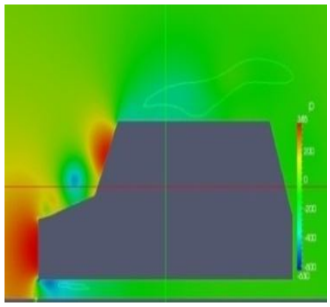


Fig. 6: Case 3 Pressure contour

Pressure contour–It is obvious from all the pressure contours that there wss a higher pressure concentration on the front of the car. Particularly, the air slows down when it approaches the front of the car and results in that more air molecules are accumulated into a smaller space. Once the air stagnates in front of the car, it seeks a lower pressure area, such as the sides and top of the car. as the air flows over the car hood, pressure is decreasing, but when reaches the front windshield, it increases briefly.

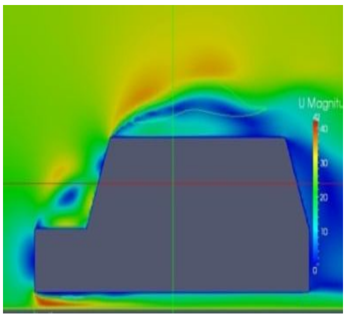


Fig. 7: Case 1 Velocity contour

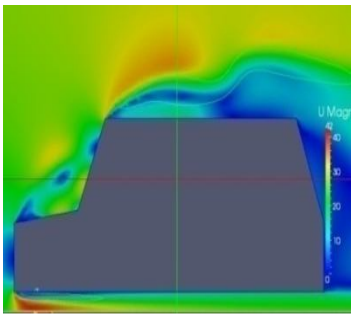


Fig. 8: Case 2 Velocity contour

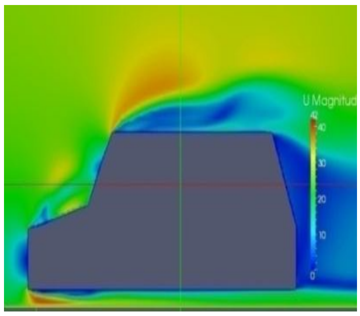


Fig. 9: Case 3 Velocity contour

Velocity contour- It is concluded from all the velocity contours that air velocity is decreasing as it is approaching the front of the car. Then air velocity increases away from the car front.

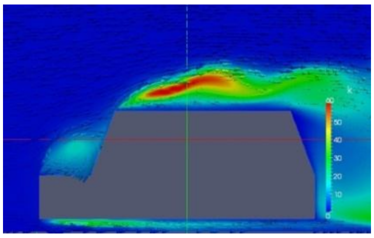


Fig. 10: Case 1 Vector plot

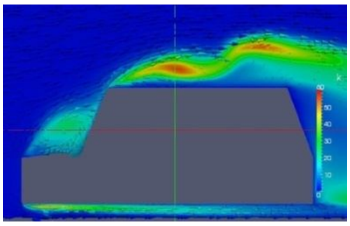


Fig. 11: Case 2 vector plot

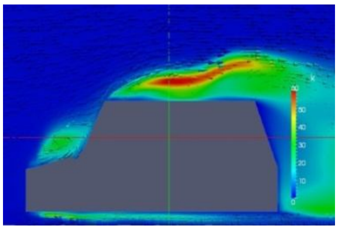


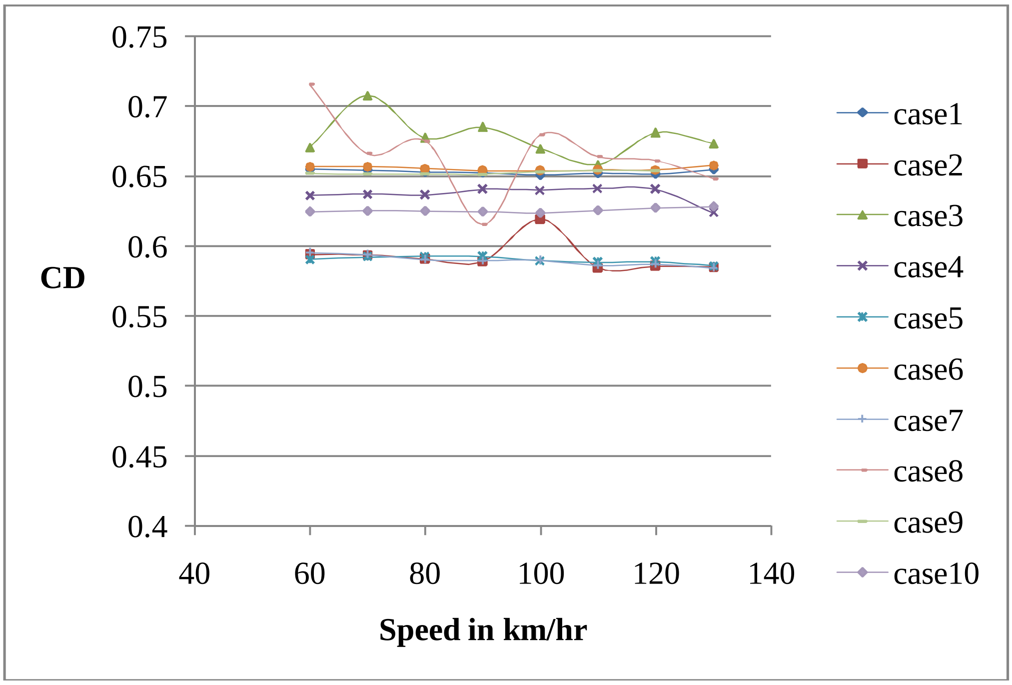
Fig. 12: Case 3 Vector plot

Vector plot for K.E.–From these plots it is concluded that the geometries having flat hood , large amount of turbulence is created at the front of the windscreen. On the other hand geometries having certain amount of hood angle the intensity of turbulence is minimum. The red vectors has maximum kinetic energy while blue vectors has minimum.

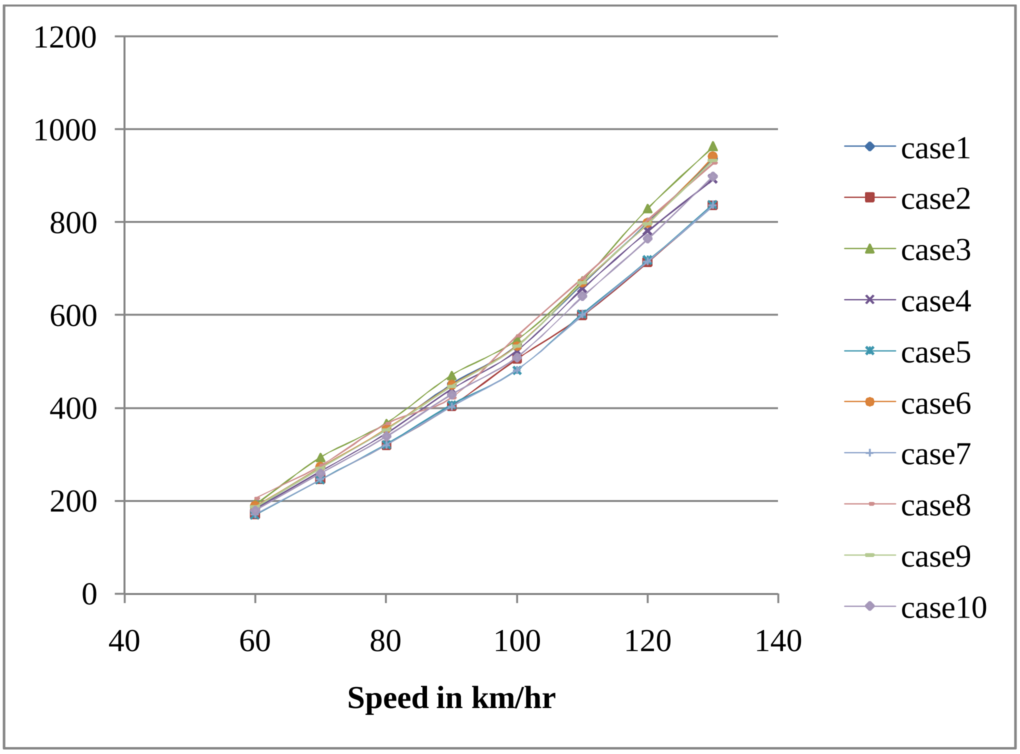
Coefficient of drag and drag force for Hatchback geometries.

Table 3: Coefficient of Drag for Hatchback Geometry.

Sr. No.	Speed in km/hr	C_d				
		Case 1	**Case 2**	**Case 3**	**Case 4**	**Case 5**
1	60	0.654951	0.594011	0.670589	0.636342	0.590656
2	70	0.654393	0.593766	0.707524	0.637281	0.592152
3	80	0.65299	0.590715	0.677222	0.636532	0.592919
4	90	0.652445	0.589211	0.685012	0.640736	0.592731
5	100	0.651024	0.619261	0.66963	0.640041	0.589595
6	110	0.652086	0.5846395	0.658186	0.641367	0.588576
7	120	0.651617	0.585798	0.680911	0.640709	0.5889
8	130	0.654798	0.585073	0.673265	0.623777	0.586087
Sr. No.	**Speed in km/hr**	C_d				
		Case 6	**Case 7**	**Case 8**	**Case 9**	**Case 10**
1	60	0.656767	0.595456	0.715567	0.651984	0.624737
2	70	0.656787	0.594008	0.666141	0.651568	0.625417
3	80	0.655578	0.590271	0.675141	0.651533	0.625077
4	90	0.653927	0.589647	0.615444	0.651276	0.624649
5	100	0.653934	0.589988	0.679625	0.653126	0.623819
6	110	0.654424	0.586203	0.663983	0.654069	0.625451
7	120	0.654499	0.587196	0.66095	0.6538655	0.627333
8	130	0.65789	0.584413	0.648114	0.652157	0.628304



Graph 1: Comparison of all C_D for various cases V/S Speed



Graph 2: Comparison of all F_D for various cases V/S Speed.

5. Conclusion

The guidelines pointed out in the text are of a general nature that can be implemented in most modern road going vehicles; Smooth vehicle shape, rounded corners, high angle for the windscreen, tapered rear end, minimized body seams. Use of rear screen angle allows a reduction in aerodynamic drag of the vehicle model. Drag reducing equally depends on configuration, dimensions and arrangement of screens as well as on model's rear part configuration.

6. Acknowledgment

The authors would like to present their sincere gratitude towards the Faculty of Mechanical Engineering in Rajarshi Shahu College of Engineering, Pune.

References

[1] Dr. Marc J. Richard , "Parallel CFD of a prototype car with OpenFOAM" June 21-24,2010

[2] Darko Damjanović et.al. " CFD Analysis Of Concept Car In Order To Improve Aerodynamics" 2011

[3] Kunal Kumar Tirkey, Suresh Kumar " CFD Analysis Of Car Body"

[4] W. Meile et.al, "Experimental And Numerical Simulation On The Aerodynamics Of Ahmed Body" vol.3(1)-March 2011

[5] Chainani. A, Perera. N, "CFD Investigation of Airflow on a Model Radio Control Race Car"July 2–4, 2008, London, U.K.

[6] JUSTIN MORDEN, "Aerodynamic Analysis And Testing Of A Formula Renault Race Vehicle Using Analytical And Computational Methods Validated By Real World Test Data.

[7] Shailesh Jindal , "Numerical Investigation of Road Vehicle Aerodynamics Using Immersed Boundary RANS Approach",2005 SAE International

[8] Mahmoud Khaled et.al. "Some innovative concepts for car drag reduction: A parametric analysis of aerodynamic forces on a simplified body" J.Wind Eng.Ind.Aerodyn.107-108(2012)

[9] Prof. G Sivaraj, Mr. Gokul Raj "Optimum way to increase the fuel efficiency of the car using base bleed." June 2012 Vol. 2 IJMER