iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 5 Issue: XI Month of publication: November 2017

DOI:

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue XI November 2017- Available at www.ijraset.com

Computational Analysis of Drag Forces on Car Profiles

Anurag Goney[1]
[1]Student, Mechanical engineering, Kakatiya institute of technology & science, Warangal, Telangana, India

Abstract: drag is force acting opposite to the relative motion of any object moving with respect to a surrounding fluid. This is sometimes called as air resistance, or fluid resistance or fluid friction between solid surface and fluid. Drag forces depend on the following factors shape, size, velocity and inclination to flow. Drag forces indirectly affect on the fuel efficiency of an automobile .to improve the fuel efficiency of automobiles the design has to be optimized in terms of low drag forces .to achieve this the profile of the automobile has to be designed in such a way that the turbulence created around the vehicle has to be reduced .
In the present work i am going to study the effect of different car profiles on drag forces by modeling different car profiles using solid works, carryout computational fluid dynamic analysis using ansys fluent and analyze the net drag forces acting on the surface of the car.
Keywords: drag, automobiles, flow velocity, drag coefficient, car profile.

I. NTRODUCTION

In fluid dynamics, drag is a force acting opposite to the relative motion of any object moving with respect to a surrounding fluid. This exists between a fluid and a solid surface. Unlike other resistive forces which are nearly independent of velocity, drag forces depend on velocity.

$$F_d = 0.5 * C_D * ϸ * A_f * V^2 \quad (1)$$

F_d= Drag force (N)
Ƥ=density of air (kg/m^3)
A_f= frontal area (m^2)
V=velocity of fluid flowing (m/s)
C_D = coefficient of drag

The main reason for the development of drag force is the separation of flow. The separation of flow causes a pressure drop on the surface which produces the drag forces. The above equation (1) clearly shows that the drag force is directly proportional to the square of velocity, frontal area and mass of properties of fluid. Examples of drag include the net aerodynamic force on cars, aircrafts and boat hulls. Drag is a vector quantity and has both magnitude and direction. Drag acts in the direction that is opposite to the motion of the object. Lift acts in the direction perpendicular to the direction of motion. Drag is of greater consideration in automobile industry while lift is of greater consideration in aviation industry.

A. Efects of drag

1) Fuel and Speed efficiency: The fuel efficiency of automobile is inversely proportional to the net amount of drag force acting it. At a particular speed the engine can take a certain amount of load if the load is increased the rpm of the engines decreases resulting in the decrease of speed of the vehicle. Drag forces are the extra and unavoidable forces which act on the vehicle, due to these forces the load acting on the engine increases and thus reducing the speed. To accommodate this type of speed drop more amount of fuel is consumed to maintain the same speed. This way the fuel and speed efficiency are affected by the drag. Drag is proportional to the square of the velocity, so at increasing velocities the drag becomes a massive problem.

*2) Aviation :*For a flight to successfully take of the drag forces are to be minimum and it should have maximum lift.

B. FACTORS AFFECTING DRAG

1) Shape and size: b Geometry has a large effect on the amount of drag generated by an object. The drag depends linearly on the size of the object moving through the air. The cross-sectional shape of an object determines the form drag created by the pressure variation around the object. If we think of drag as aerodynamic friction, the amount of drag depends on the surface roughness of the object; a smooth, waxed surface produces less drag than a roughened surface. This effect is called skin friction and is usually included in the measured drag coefficient of the object.

2) Motion of the fluid: Drag is associated with the movement of the automobile through the air, so drag depends on the velocity of the air. Like lift, drag actually varies with the square of the relative velocity between the object and the air. The inclination of the object to the flow also affects the amount of drag generated by a given shaped object. If the object moves through the air at speeds near the speed of sound, shock waves are formed on the object which create an additional drag component called wave drag. The motion of the object through the air also causes boundary layers to form on the object. A boundary layer is a region of very low speed flow near the surface which contributes to the skin friction.

*3) .Properties of the fluid flowing :*Drag depends directly on the mass of the flow going past the aircraft. The drag also depends in a complex way on two other properties of the air: its viscosity and its compressibility. These factors affect the wave drag and skin friction which is described above.
We can gather all of this information on the factors that affect drag into a single mathematical equation called the Drag Equation (1).

C. Problem Statement

Drag forces the affect the performance of the car/automobile vehicle. These forces are generally considered when designing of a new automobile. These play a crucial role in determining the fuel and speed efficiency of the car. The drag force acts actually because there is an obstruction to the flow of the fluid. The car has to cut through the fluid with certain velocity and due to friction arising between the two surfaces a resistance is developed .this is called as drag. This present work is aimed at analyzing the effect of windshield inclination, hood effect and front bumper profile on the drag coefficient and the amount of drag forces produced at constant fluid velocity and Fluid properties.

II. METHODOLOGY

A. Cad model

To model the profiles of car I have used Solid works. The sedan model car which is mostly used by public is used for the design. To simplify the design process few intricate features of the car are neglected as they don't have significant impact on the drag forces.

TABLE I
Design conditions

Design number	Windshield inclination	Hood inclination	Bumper profile
1	90 degrees	0 degrees	Flat
2	75 degrees	0 degrees	Flat
3	60 degrees	0 degrees	Flat
4	60 degrees	5 degrees	Flat
5	60 degrees	5 degrees	Curved

Fig 1: Design 1

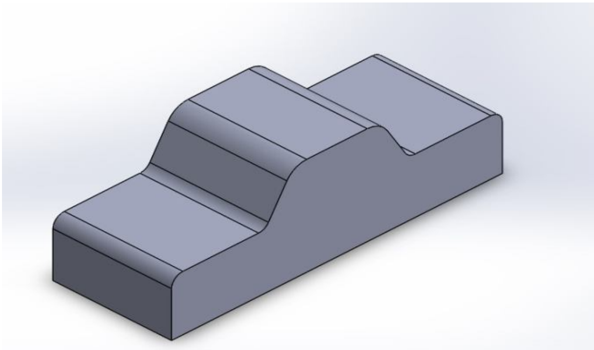


Fig 2: Design 2

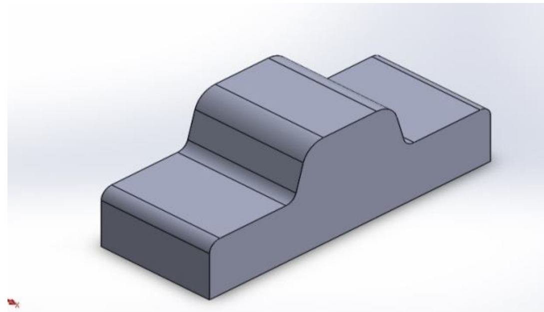


Fig 3: Design 3

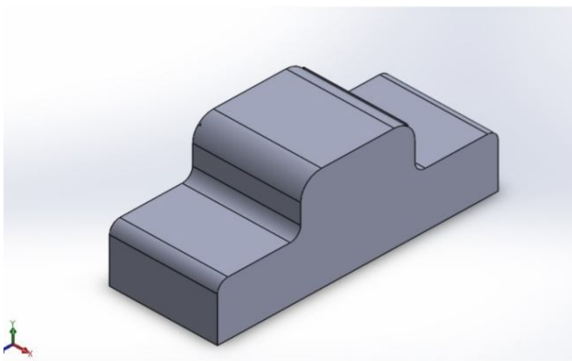


Fig 4: Design 4

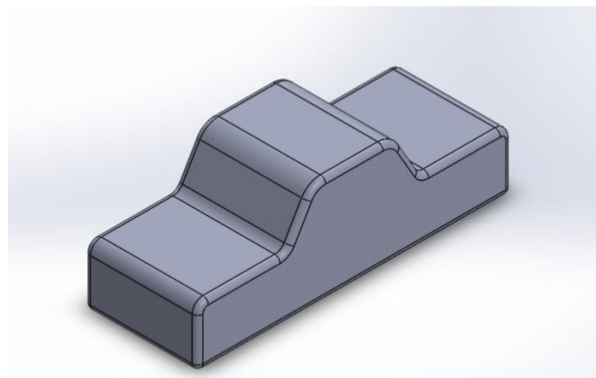


Fig 5: Design 5

The dimensions of the car are taken between: 2 *1*1 (in meters)

B. Simulation

The design files were then imported into ANSYS FLUENT software which is widely used to solve the CFD problems. Meshing was done using the predefined settings available in the fluent solver. The material of the car was set to Aluminum and the fluid used for the analysis was air. The material properties were taken from the Ansys fluent material database. No slip conditions were given to simplify the analysis. The solution was carried out until the residuals reached a value of 10^{-3}.

TABLE II
Analysis conditions

Parameter	Value
Velocity	30 m/s or 108 kmph
Density of air	1.225 kg/m^3
Enclosure	1m cube with uniform properties

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue XI November 2017- Available at www.ijraset.com

III. RESULTS AND DISCUSSIONS

After the analysis has been carried out the drag forces and drag coefficients on different profiles are summarized into the table below

TABLE III
Results

Design number	Drag force (N)	Coefficient of discharge
1	841	0.762
2	713	0.646
3	506	0.458
4	351	0.318
5	311	0.282

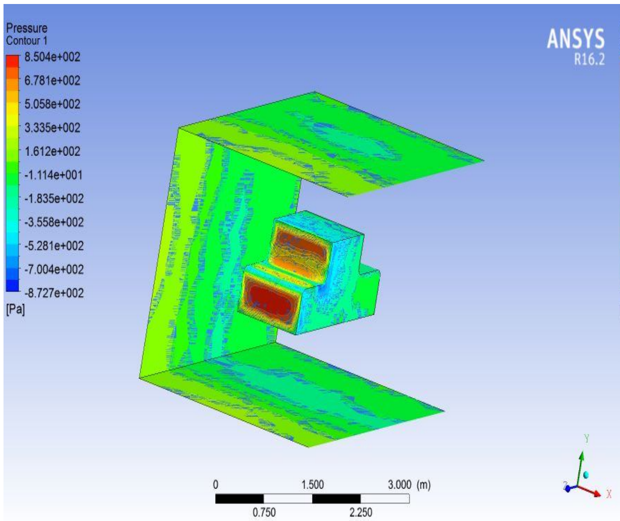


Fig 6: pressure contour design 1

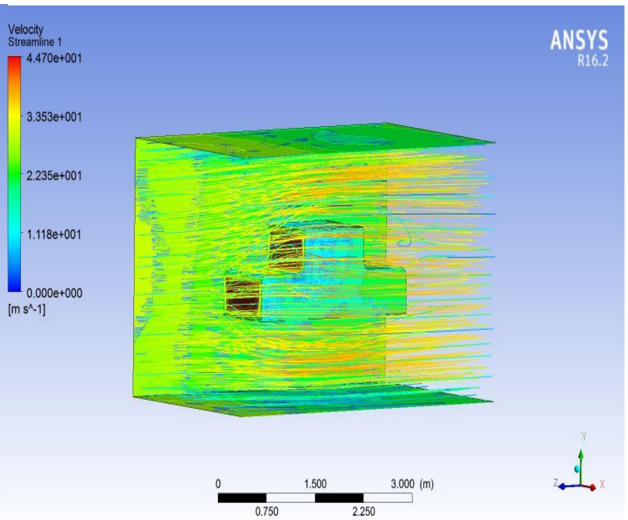


Fig 7: Streamline flow of design 1

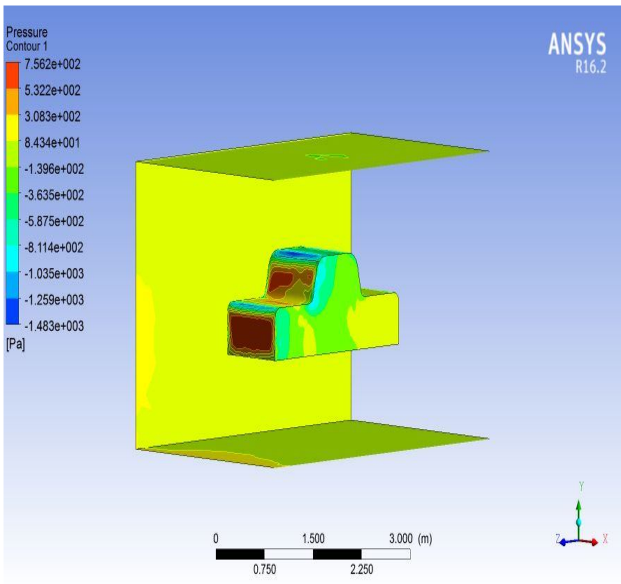


Fig 8: pressure contour design 2

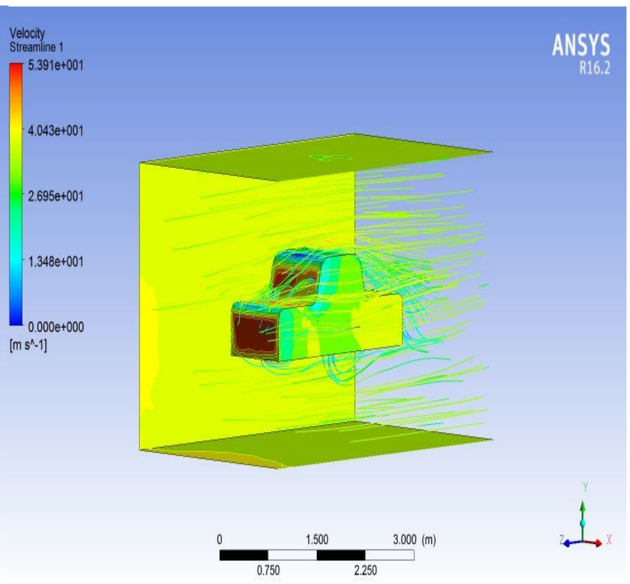


Fig 9: streamline flow of design 2

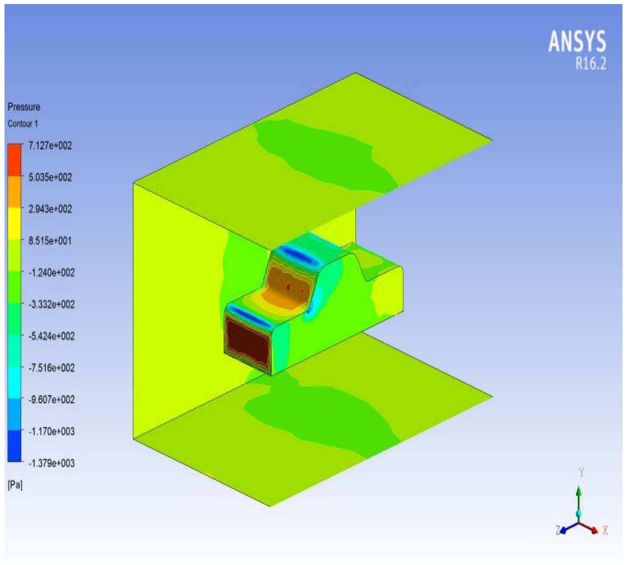


Fig 10: pressure contour design 3

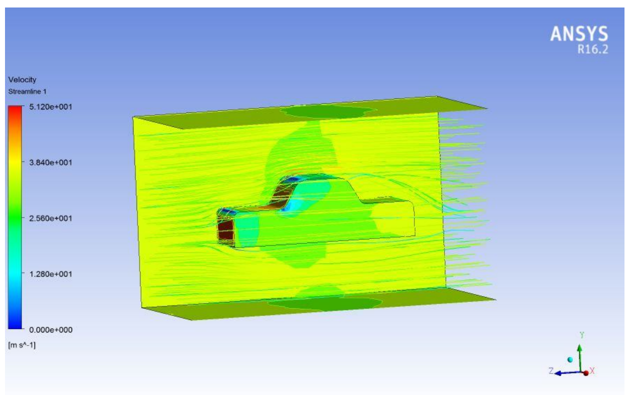


Fig 11: Streamline flow of design 3

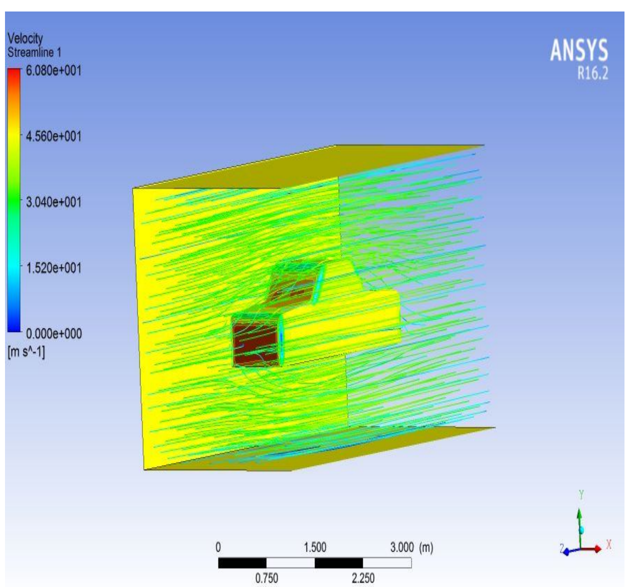


Fig 12:pressure contours design 4

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue XI November 2017- Available at www.ijraset.com

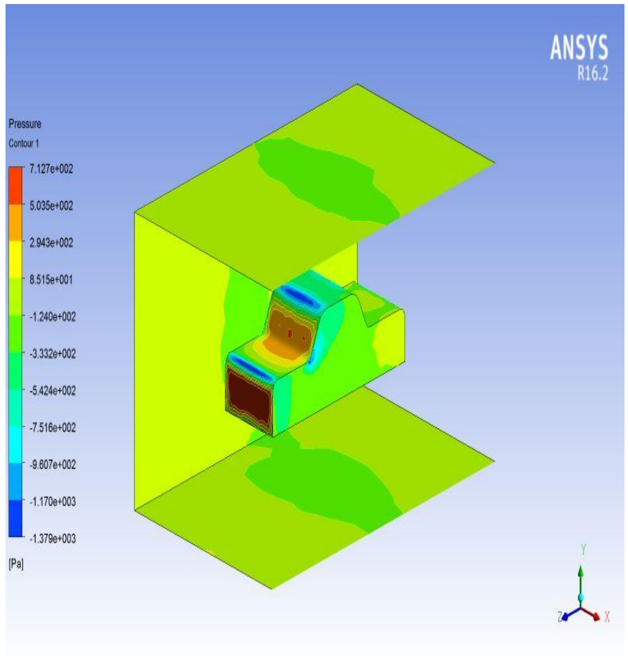


Fig 13: streamline flow of design 4

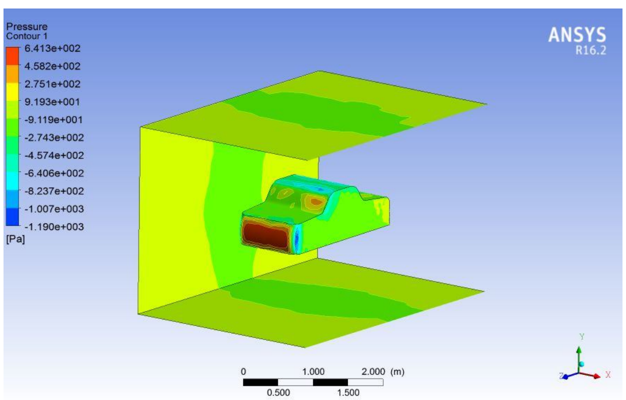


Fig 14: pressure contours design 5

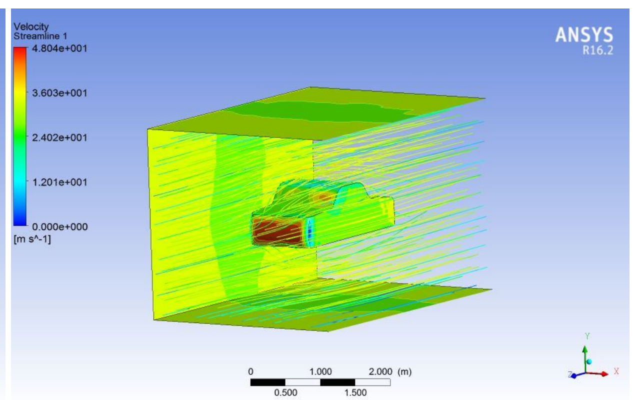


Fig 15: streamline flow of design 5

From the Table number 3 it can be clearly observed that the drag forces and coefficient of drag are decreasing. When the pressure contours are observed from Design 1 to design 4 the front bumper being flat completely obstructs the flow of air creating a stagnation point at the bumper but in case of design 5 the pressure is maximum at the bumper part even though it is curved but it is less compared to the pressure created in the other four designs. Drag is a force that is created when the flow of obstructed. Flat bumper completely hinders the flow of air while the curved bumper allows the air to flow on its surface creating the minimum amount of drag.

In the first 3 designs the hood of the car was horizontal and in the next two designs i.e., design 4 &5 the hood is inclined at 5 degrees. The importance of this inclination can be known from viewing the streamline flow of air over the car. In the first three cases the air flow is hindered at the junction of hood and windshield but in the other two cases this inclination facilitates the easy flow of air over the surface of air there by significantly reducing the drag forces .When the Streamline flows are analyzed there is lot of turbulence created in the first two designs compared to the other designs. This clearly depicts that more the hindrance to the air flow more is the drag. When the windshield is made to incline, as the angle of inclination decreases the drag forces decrease this is because of the ease of flow of air. If the pressure contours of all the designs are observed there is a great reduction in the amount of pressure that is built up on the windshield. In the design 1 there is stagnation point built up because of the complete hindrance to the flow of air at 90 degrees orientation but as the inclination decreases the pressure built decreases.

IV. CONCLUSIONS

The results obtained showed that by changing the orientation of hood and windshield there is significant reduction in the drag forces and this would ultimately help in increasing the fuel efficiency of the car. Though the numbers may be low when operated at lower velocities but this reduction in the drag coefficient will be help us quench the thirst for the hunt of high performance cars.

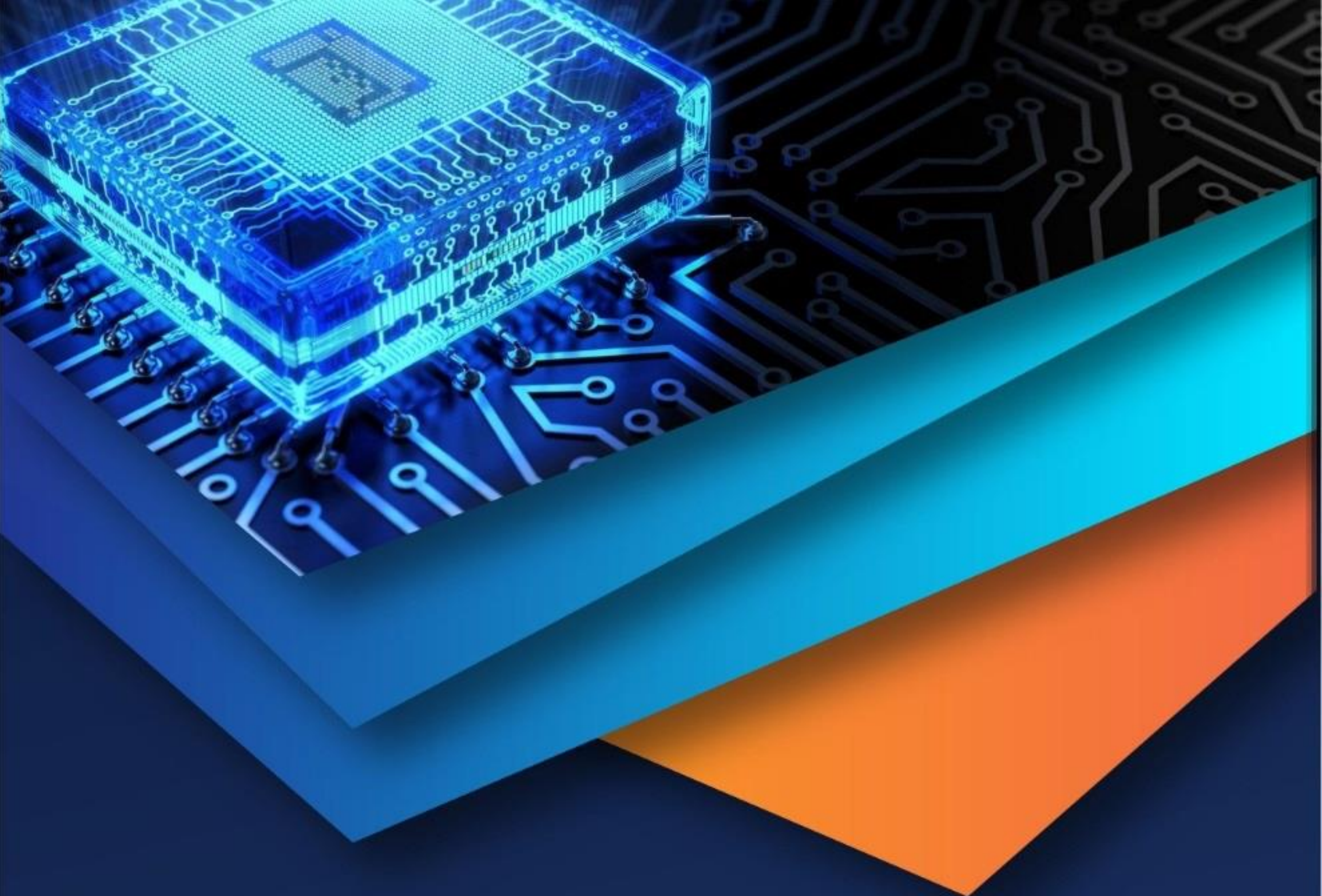
V. ACKNOWLEGEMENTS

I express my deep sense of gratitude to Late Dr. K. GURU RAJ, (ex-principal), KITS Warangal for igniting this spark of research interest in me. I would like to express my special thanks to Sri G. VINOD KUMAR, Assistant professor Mechanical department, KITS Warangal for being a constant support and guiding me through this process.

REFERENCES

[1] Gavin Dias , Nisha R. Tiwari ,Juju John Varghese and Graham Koyeerath , “Aerodynamic analysis of a car for reducing drag force” , IOSR journal of Mechanical and Civil engineering. E-ISSN: 2278-1684

[2] AshfaqueAnsari,RanaManojMourya, “Drag force analysis of car by using low speed wind tunnel”, International journal of Engineering Research and Reviews .ISSN 2348-697X.



10.22214/IJRASET

INDEX COPERNICUS
45.98

IMPACT FACTOR:
7.129

TOGETHER WE REACH THE GOAL

IMPACT FACTOR:
7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY