iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 8 Issue: V Month of publication: May 2020

DOI: http://doi.org/10.22214/ijraset.2020.5422

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue V May 2020- Available at www.ijraset.com

Investigation of Light Vehicle Stability on Overtaking Heavy Vehicle using CFD

A. Sabik Nainar
Assistant Professor, Department of Aeronautical Engineering, Kalasalingam Academic of Research and Education.

Abstract: The importance of overtaking investigation is to ensure that vehicle can remain stable under turbulent conditions. The flow distribution occurring on a light vehicle (bike) when it passes over a heavy vehicle (bus) is investigated using three dimensional (3D) Computational Fluid Dynamics. Previous tests have been researched and compared; it was found that, the overtaking vehicle causes the other vehicle to experience sideward forces, leading to roll followed by undergoing a yawing moments as well as reductions in speed caused by changing pressure distributions throughout the vehicles. An aerodynamic characteristic of a vehicle is a significant interest in reducing accidents due to wind loading and in reducing the fuel consumption. To avoid accidents we should reduce the wake region of the entire body of each vehicle. For that modulation in shape the boundary layer condition are changed and the flow separation are also reduced to small.
Keywords: Diffuser; CREO Parametric 2.0; ANSYS R14.5.

I. INTRODUCTION

In the past, the external shape of cars has evolved particularly for safety reasons, comfort improvement and also aesthetic considerations. Consequences of these guidelines on car aerodynamics were not of major concern for many years. However, this situation changed in the 70's with the emergence of the oil crisis. The first studies on the overtaking effects were investigated in response of the weight reduction of cars involved by the first oil crisis [1]. After this oil crisis, car manufacturers have made substantial efforts to reduce the fuel consumption.

This was achieved by improving the design of the cars by developing efficient engine or by decreasing the vehicle weight. With the third option, vehicles became more sensitive to unsteady aerodynamic effects such as those induced by an overtaking manoeuvre [2]. The overtaking manoeuvre between two vehicles yields additional aerodynamic forces acting on both vehicles. These additional forces lead to sudden lateral displacements and rotations around the yaw axis of each vehicle.

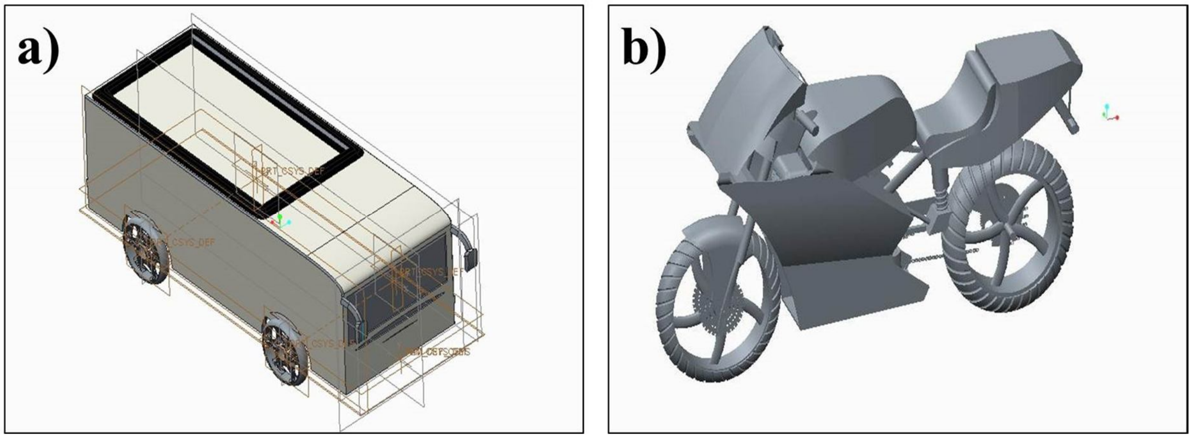
Such sudden change of the side force and of the yawing moment, complicates the steering corrections performed by the driver and can yield critical safety situations, in particular in adverse weather conditions, such as crosswinds or rain. when a vehicle moves in a certain velocity along an open road or through a tunnel, it will displace air around it causing an air flow profile creation along the vehicle. The air profile is always evaluated for the risk factor of causing accidents and also the potential of generating electricity [3]. Several wind and weather-related accidents of road vehicles occur every year, especially at exposed locations where topographical features magnify the wind effects as they researched about the parameters influencing wind-related accidents of road vehicles. The most notable ones involve high-sided vehicles. The theory presented is applied to a multitude of scenarios to explore the interrelation between the various basic variables and how they affect the probability of accident given in terms so-called accident index. [3] The analysis demonstrates that wind-related accidents are the consequence of a combination of several basic variables as represented by the accident index. Overtaking or passing is the act of one vehicle going past another slower moving vehicle, travelling in the same direction, on road.

As the bike passes the bus during an overtaking manoeuvre the flow around the two vehicles are interact generating pressure variation over the car. This pressure variation may lead to adverse effect on the car handling and stability. This case is somewhat similar to the study of cross wind problems on vehicles. **[4]** Sivanesh et al investigated on the stopping distance of a car by introduction rear dynamic spoiler to reduce the stopping distance.

Sudden change of wind velocity in the wake of the truck causes rapid change of the Aerodynamic forces acting on a passing by car. This sudden change causes difficulty in controlling the vehicle which may result in miss-steering and an accident. To avoid such accidents, the behavior of the light vehicle to such disturbances created by the heavy/large vehicle is difficult to evaluate and are still not clear. This paper is to investigate that how a light vehicle passing by a heavy/large vehicle is affected by the flow disturbances created by the heavy vehicle. The transient aerodynamic forces acting on a vehicle model overtaking another vehicle model.

II. MODEL CREATION

The Modeling is done using CREO Parametric 2.0 is shown in Fig.1, but when modeling for a CFD analysis it is to be considered developing a separate part model for the fluid, this can only done using Boolean operation.



.

Fig.1. Creo Parametric model of a) bus and b) bike with aerokit.

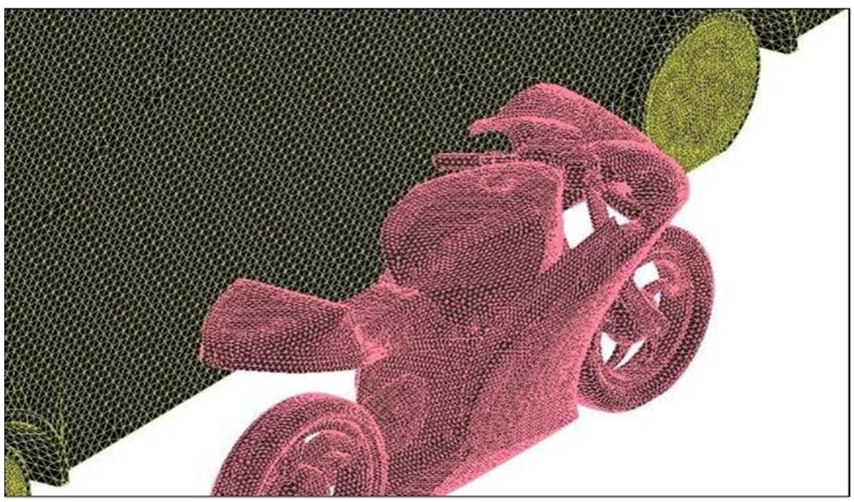


Fig.2. Surface mesh on fluid domain.

A. Transformation of Model

Then the model is converted in to the IGES and STP format which is most suitable and easy access for any other software"s. Using the STP format we can import the car wheel rim model from CREO PARAMETRIC to ANSYS. Now we can make CFD model for flow analysis, for the flow analysis we required air domain. Figure 3.8 shows the CAD model after fluid domain extraction.

B. Meshing

After cleaning up the geometry surface mesh (Fig.2) is generated in ANSA tool itself. All the surfaces are discredited using tri surface element .As the geometry has some complicated and skewed surfaces tri surface elements are used to capture the geometry. The meshing are 2 types. They are

1) Surface meshing
2) Volume meshing

C. Boundary Conditions

For flow analysis

1) Inlet is assumed to be velocity inlet at 16.6 m/s
2) The outlet is assumed to be pressure outlet at 0 Pascal.

III. RESULT AND DISCUSSIONS

Figure.3 depicts the static pressure and static velocity of base case without diffusers (aerokit). It is clear that the minimum static pressure of -5.51^{+02} and maximum static pressure of-$1.53e^{+02}$ are generated for base case without diffuser. Fig. 3b indicates that the velocity in overtaking process. The initial velocity in that process is 16.6 m/s. When the velocity is increased there is some suction medium created. The suction medium pulls over the particle, which is near of that. This is also called as “WAKE REGION”. Due to the wake region more accidents are occurring. Fig.3d shows the negative pressure region in overtaking process. This negative pressure has the capability to pull the nearer object. Due to this the uncontrollable driving and increased fuel consumption is created. Inorder to achieve the safety driving and large fuel efficiency negative pressure region (wake region) can be reduced by implementing aerokit. Fig.5 indicates that the static pressure contours of vehicle having the aerokit medium. It is observed that the minimum static pressure of $-7.37e^{+02}$ and the maximum static pressure of $1.78e^{+02}$ are produced in static medium. The static pressure indicates that the light moving vehicle passed over the heavy vehicle created some negative pressure. The maximum pressure indicates the reduction of the negative pressure region (wake region) in frontal area of heavy vehicle but airflow in outside attains minimum pressure region.

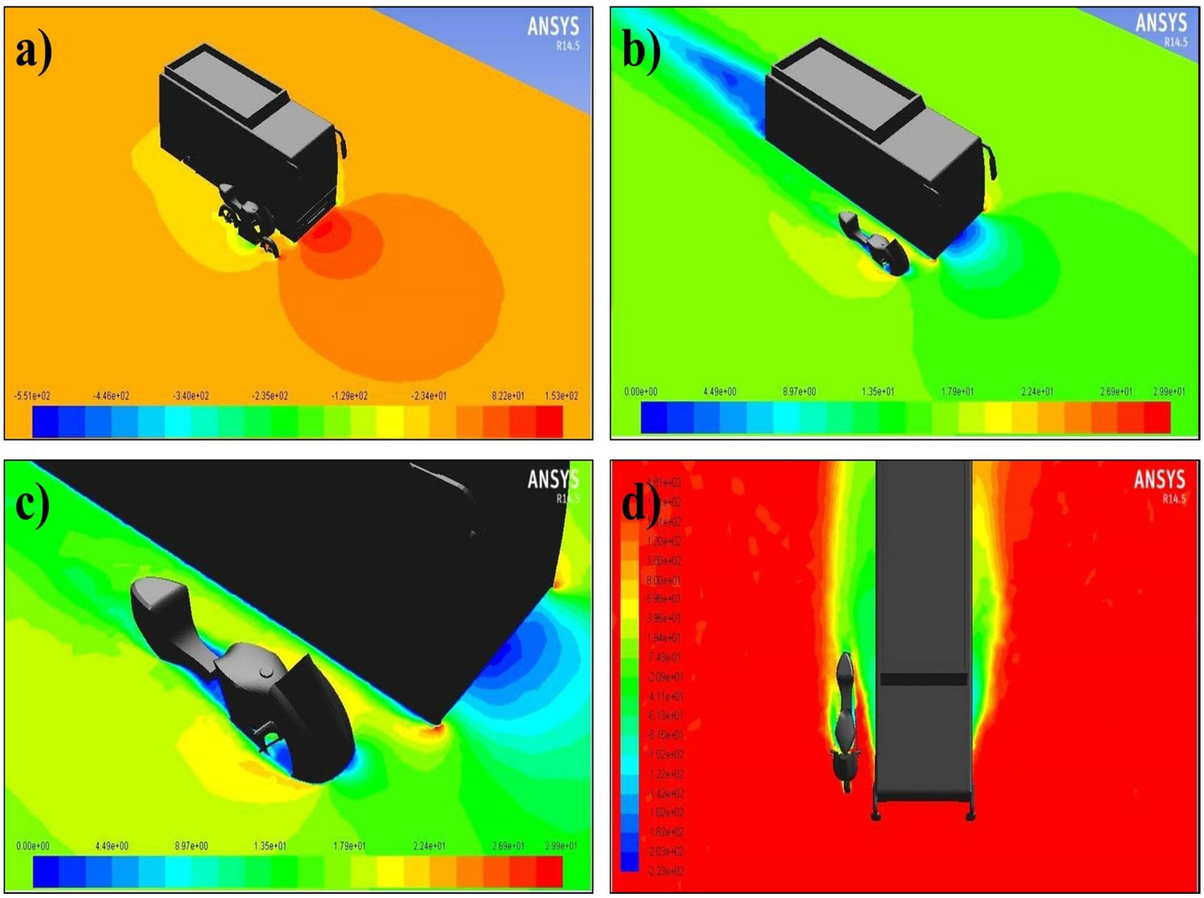


Fig.3. Base case without diffusers (aerokit) a) Static pressure contours, b) Velocity Magnitude, c) Increased velocity pattern between vehicles and d) Negative pressure region.

Fig.4 depicts the base of vehicle with aerokit. From Fig.3a it is observed that the minimum velocity magnitude of 1.01e+01m/s and the maximum velocity magnitude of 1.8e+02 m/s. The velocity level of 16.6m/s is applied. A circulation indicates the velocity speed. Fig.4c shows the velocity vector diagram of reduction of wake region while over passing. This speed creates some vortexes. This image explain that the re- circulation of velocity in both vehicles. The recirculation of vortex is creating some drag. Fig.4d shows the velocity vector recirculation zone of bus and bike. Using the diffuser (aero kit) reduction in the drag and recirculation velocity is attained.

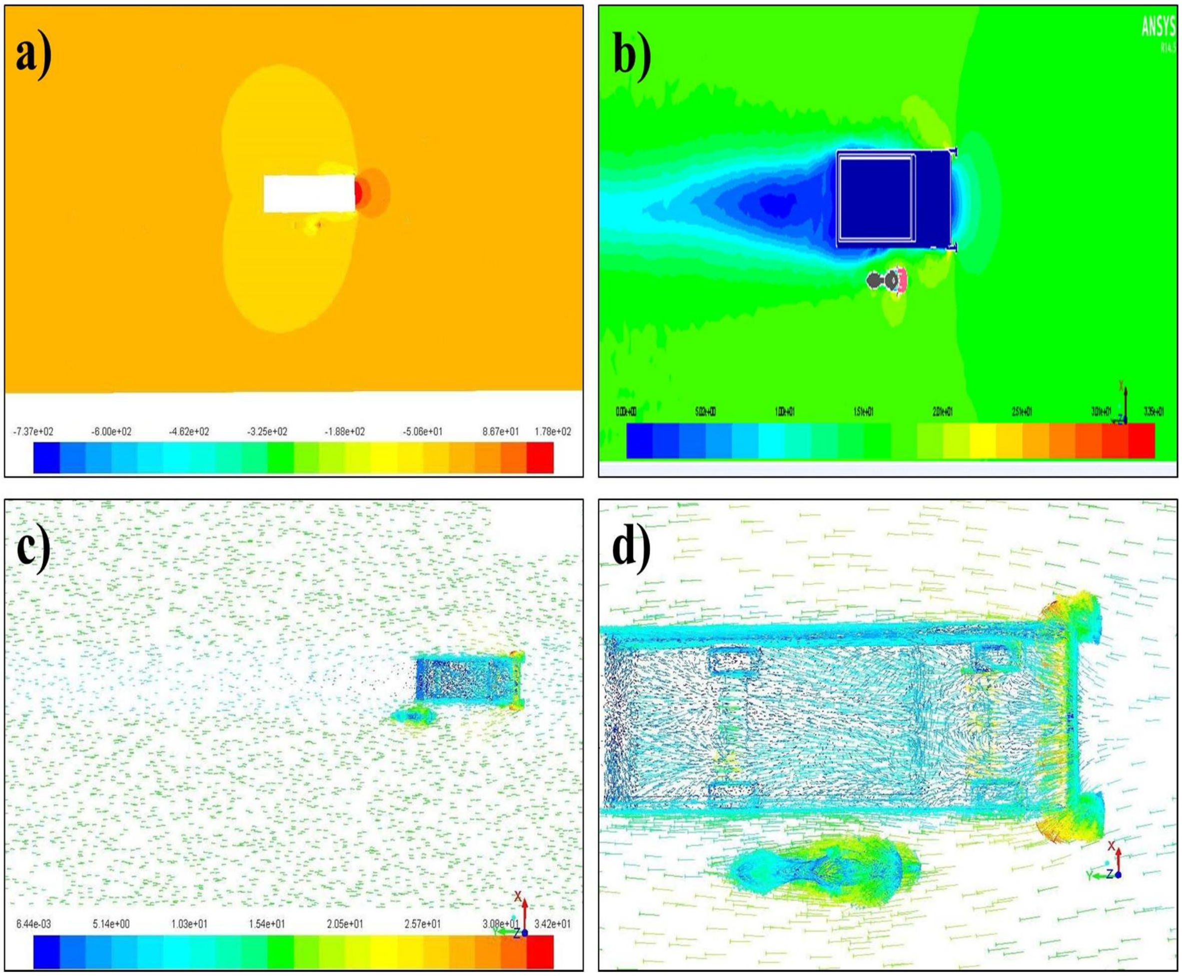


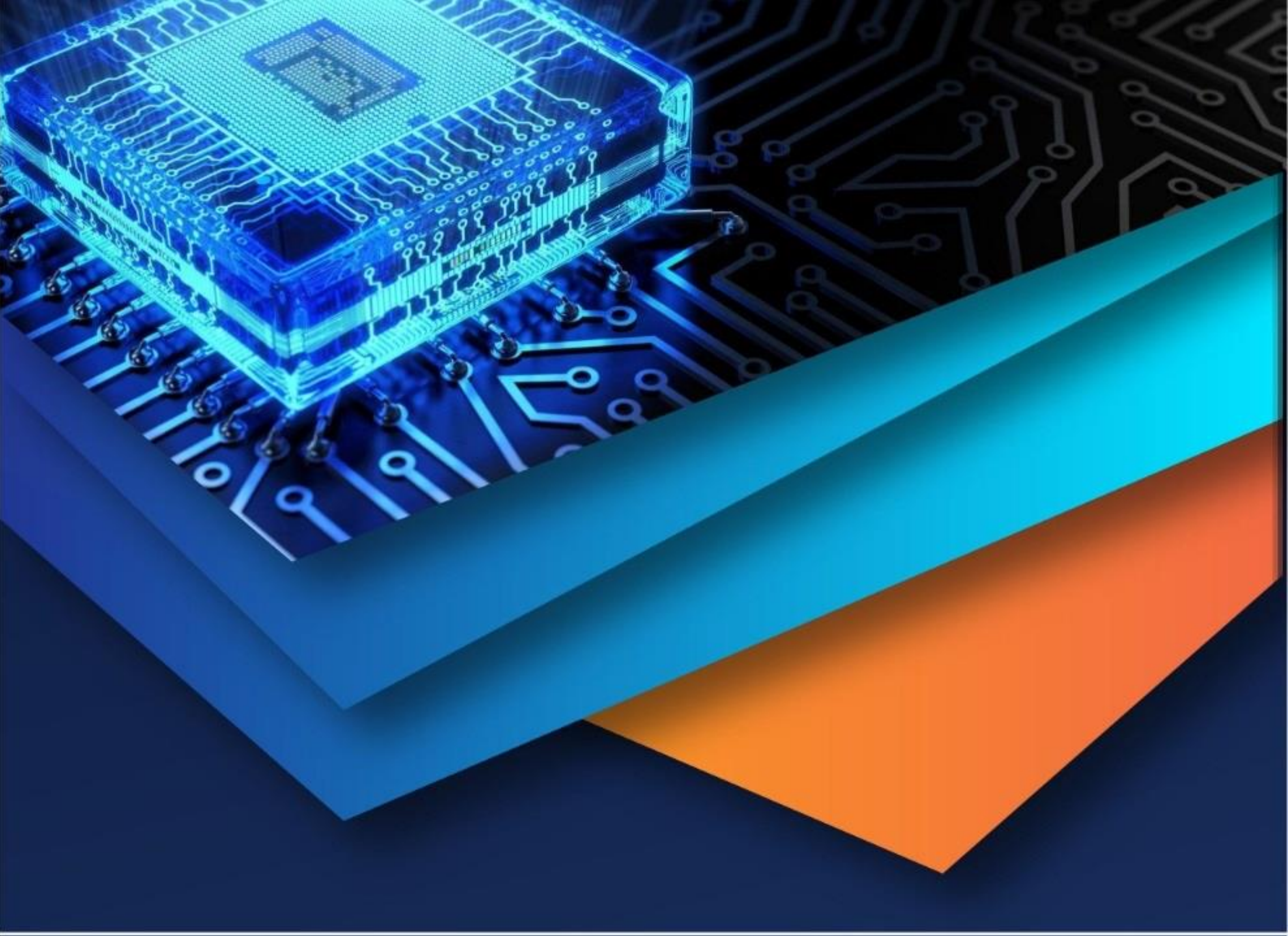
Fig.4. Base case without diffusers a) static pressure contour, b) Velocity magnitude in reduction of wake region, c) and d) Velocity vector recirculation zone of bus and bike

IV. CONCLUSION

In this study with and without diffuser in the vehicle has been investigated. The use of diffuser, which is placed at the side way of heavy vehicle resulted deviation in the air flow automatically. This gives the effect of reduction in wake region. When the wake region is reduced automatically the drag gets reduced. This information can be very much used to avoid or delay the point of separation which can be used to increase the stability of driving. Implementing diffuser in the vehicle may reduce the possibility of accidents, at the same time reduces the fuel consumption.

REFERENCES

[1] Heffrey R..K (1973), Aerodynamics of passenger vehicles in close proximity to trucks and buses, SAE paper 901 – 914.
[2] Telionis D. P, Fahrner C. J, Jones G. S (1984), An experimental study of highway aerodynamic imterferences, Journal of Wind Engineering and Industrial Aerodynamics 17 267 – 293.
[3] Snabjornson J, Baker C and Sigbjornson R (2007), Probabilistic assessment of road vehicle safety in windy environments, Journal of Wind Engineering and Industrial Aerodynamics, Volume 95, Issues 9–11, pp. 1445-1462.
[4] Sivanesh Prabhu.M, Arulvel.S,Mayakkannan.S (2017), CFD Analysis of Automobile Rear Dynamic Spoiler, International Journal of Scientific & Engineering Research Volume 8, Issue 8.



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