International Journal of Research in Engineering and Science (IJRES)
ISSN (Online): 2320-9364, ISSN (Print): 2320-9356
www.ijres.org Volume 9 Issue 4 ǁ 2021 ǁ PP. 30-40

Design and analysis of Aerodynamic component for reducing wake of automobile vehicles

Nirav vasava
Department of Mechanical Engineering, Parul university, Wadhodia, Vadodara
Corresponding Author: Nirav vasava

Abstract
Vehicle aerodynamic is main focus on the reducing drag force, preventing undesired lift force, safely achieve higher speed on road. In this paper we are discuss various aerodynamic force on the car and design component for vehicle, that component is reducing aerodynamic force like lift force, drag force, wake and vortex. This component is combination of the diffuser and spoiler. Diffuser decreases the velocity of a air and increase the pressure of air, this effect wake area (low pressure air area) and wake length is reduce. low speed air reduce the vortex length. Spoiler is create the downward force and reduce lift force. This component join on rear portion of the vehicle.
In This paper design the aerodynamic component and CFD analysis with Ahmed body model in Ansys software and compare wake structure with component and without component. We are try to reduce the lift and drag co-efficient. In this analysis larger eddy simulation (LES) model use for analysis and air velocity is 60 m/s.
Keywords: *Aerodynamic component, wake, aerodynamic wake, design of wake reduction component, air flow analysis of Ahmed body.*

Date of Submission: 28-03-2021 Date of acceptance: 11-04-2021

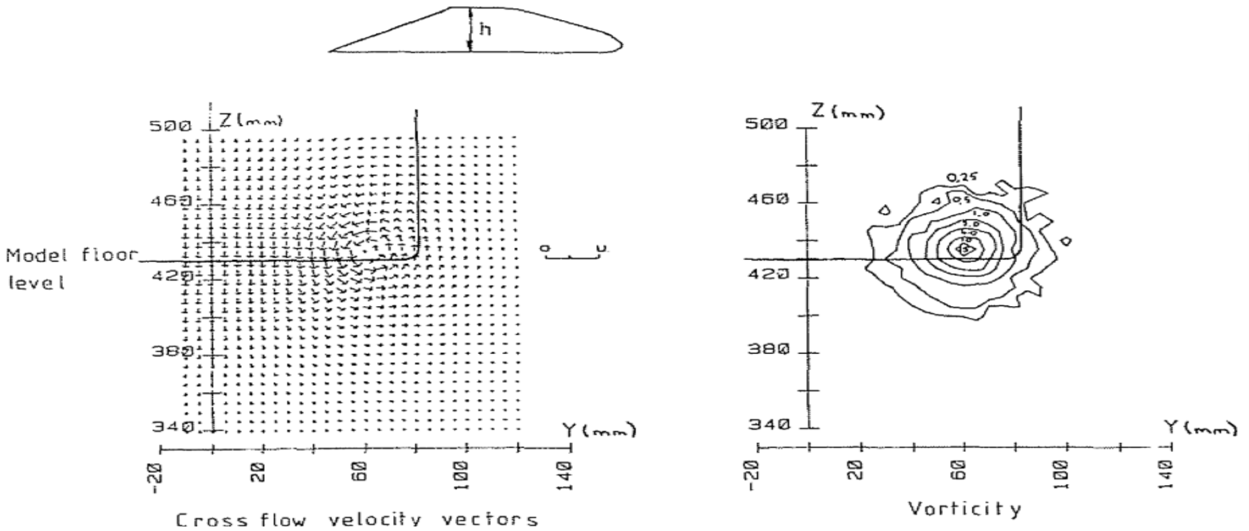
I. INTRODUCTION

The aerodynamics is the most important factor in the automobile industry. the art of aerodynamics allows far higher cornering speeds than would be possible without downforce, and so not only ensures a better performance but also even more safety. Automotive aerodynamic is main goals are reducing drag and wind noise, minimizing noise emission, and preventing undesired lift force and other causes of aerodynamic instability at high speed. It is the study of forces and the resulting motion of objects through the air . studying the motion of air around an object allows us to measure the force of lift, which allows an aircraft to overcome gravity, and drag, which is the resistance an aircraft feels as it move through the air.

In the aerodynamic main force are drag force, lift force, thrust force, wake structure, weight of car. Wake is very important factor for road vehicles. Longitudinal vortex are present in the wake of road vehicles. Wake and vortex produce by the two component of vehicle.

1. force on the side of car (side drag)
2. drag is generate by corner of the rear side of car (vortex component drag)

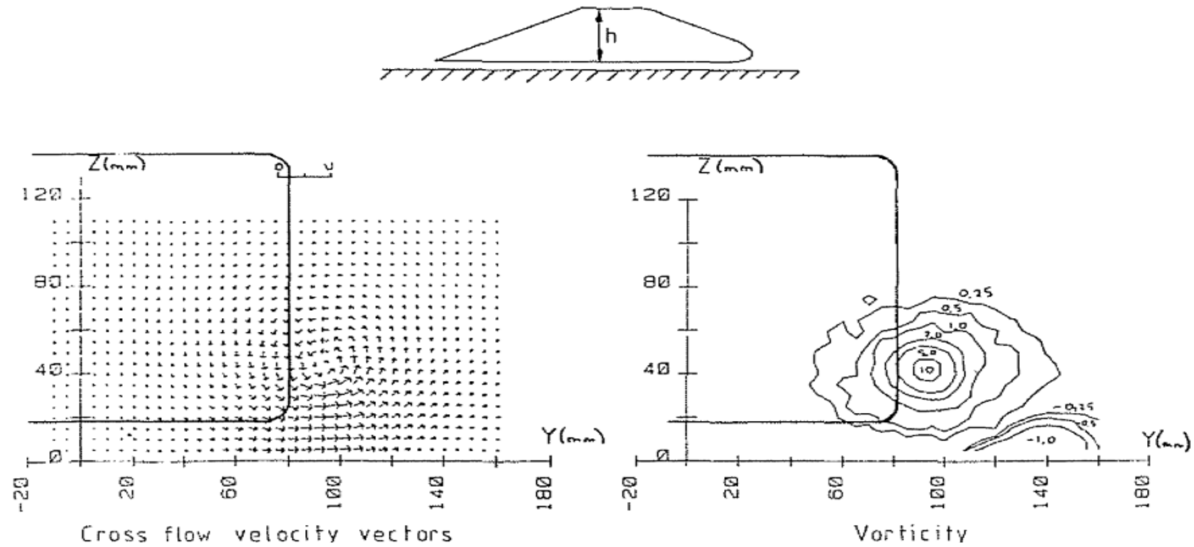
Wake structure is depended on the side, rear shape and height of car. wake is low pressure area and air speed higher. Vortex structure in drag force, lift force are available. These force are higher fluctuating on the driving time of vehicle. and some negative force is higher produce and car is lift from the floor. This drag and lift forces reduce by aerodynamic shape change of vehicle body but that is not right all time, because of this we are design the component for reducing wake vortex of the car. This component is reduce the wake and reduce length.



(a) (b)

Figure 1 wake survey of body around in air (a) air vectors around the body (b) vorticity of body in air(no floor)

In the picture above there is air flow around a body and there is no floor. figure (a) shows that air velocity vector around the body and vector is show the direction of air. Figure(b) show that the vorticity of the vortex behind the body. Vortex is generate behind the body and this vortex is generate positive lift force.



(c) (d)

Figure 2 wake survey of body with fix floor(c) air vector around the body (d) vorticity of body with fix floor

In the picture above there is floor is available of the bottom of the body and air is passing around the body. In the figure(c) is show that air velocity vector with height. Figure(d) shows that vortex behind the body and in this positive and negative force is produce. Also upper lift and downward lift are generate. this body is direct compere with the road vehicle and in the vehicle this type of vertex is generate.

1.1 Ahmed body model

In this paper we are design a component for the rear portion of car. this component combination of spoiler and diffuser. In this paper we will use Ahmed body and put our component on it and we will do CFD analysis in the Ansys software. With the help of this software, we will try to reduce the wake and vortex and lift force. Ahmed body used the body in the old days for aerodynamic analysis in automobile field. This body in seen better air wake and vortex and air flow around the body. Ahmed body analysis data are available on the net and many component use for testing on this body also valid research paper available for this reason we use the Ahmed body.

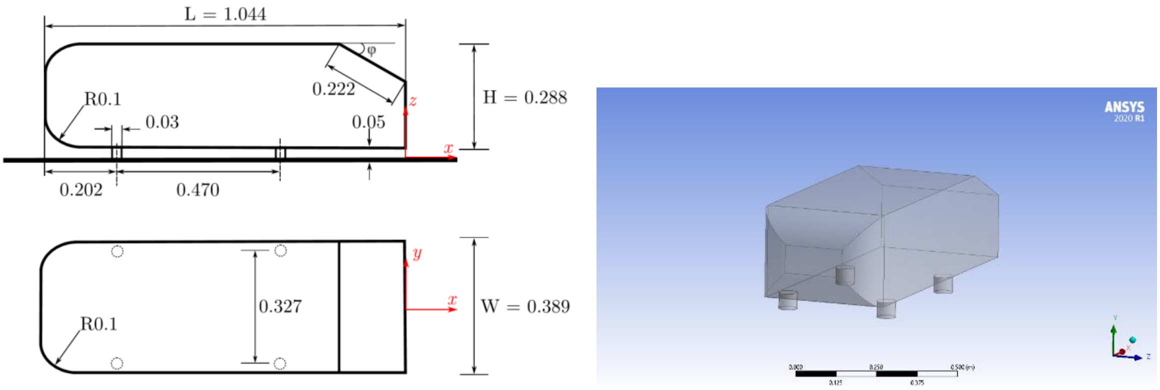
The co-efficient of drag(Cd) and co-efficient of lift(Cl) of the Ahmed body is compere with other research paper of Ahmed body and data are write in the below table.

- This data collect from the old research paper (air speed= 40 m/s)

Table 1 comparison of force co-efficient with previous studies for the Ahmed body (air speed is 40 m/s) *(A.rao, 2018)*

	Author name& year	Cd	Cl
1	Meile et al. (2011)	0.299	0.3450
2	Meile et al. (2016)	0.2964	0.358
3	thacker et al. (2012)	0.3840	0.4220
4	rossitto et al. (2016)	0.356	0.311
5	Guilmineau et al.(2017)	0.3802	0.3306
6	Guilmineau et al.(2017)	0.4371	0.3747
7	A.rao (2017)	0.3718	0.3815
8	Current study model	0.3616	0.2857

We have written all the value of co-efficient of drag (Cd) and co-efficient of lift (Cl) of the table above from the research paper and the Cd & Cl value of the our current models comes around their value, so this model use for the analysis for wake reduction component.

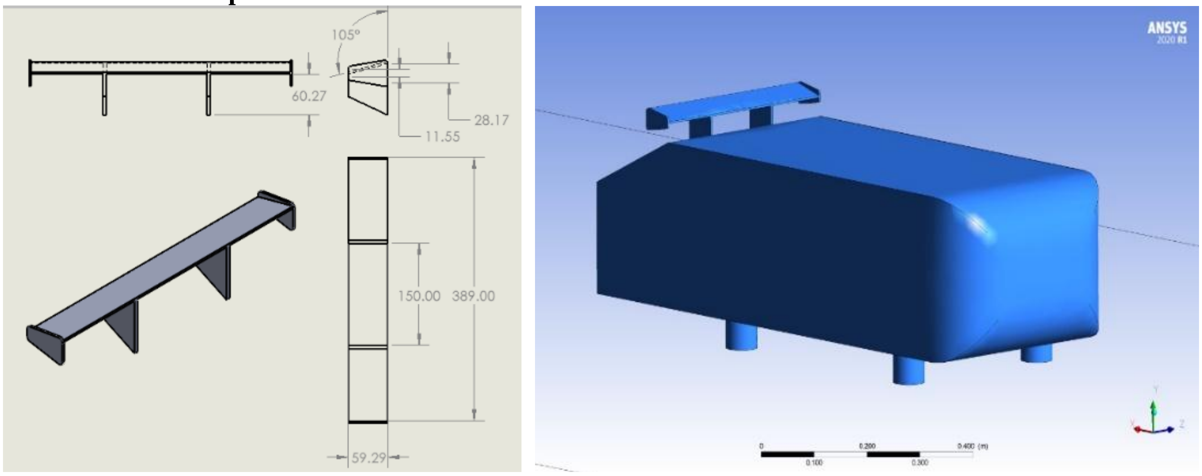


(1) **(2)**

Figure 3 (a) dimension of original Ahmed body (b) 3D model of Ahmed body

The Ahmed body shown in fig. () is an idealised car model, it is represent of the hatchback car of an automobile. The length (L=1.044 m), width (w=0.389 m), height (H=0.288 m) and back slant constant (0.222 m) at the angles (φ=25^{o}) of the Ahmed body used in this study were identical to the original model used in (S.R.Ahmed, 1983)

1.2 Analysis model & wake reduce component
1.2.1 model 1 with spoiler

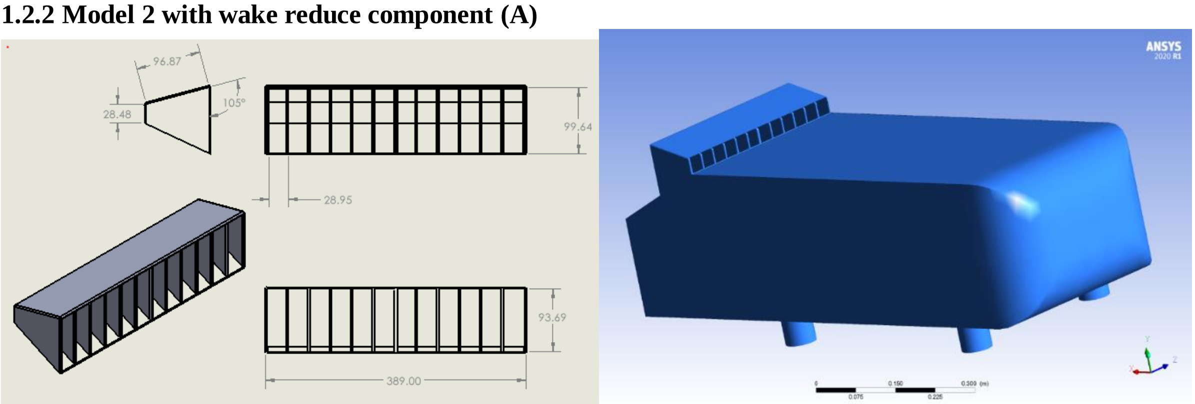


(a) (b)

Figure 4 (a) dimension of spoiler (b) spoiler mount on the Ahmed body (Model 1)

Spoiler all dimension is given in the figure. The spoiler is mounted on top of the Ahmed body because we have replace the spoiler to the wake component. and also spoiler is on the wake component. The angle of this spoiler plate is 15°.this angle is taken from the this paper (A.rao, 2018)

1.2.2 Model 2 with wake reduce component (A)

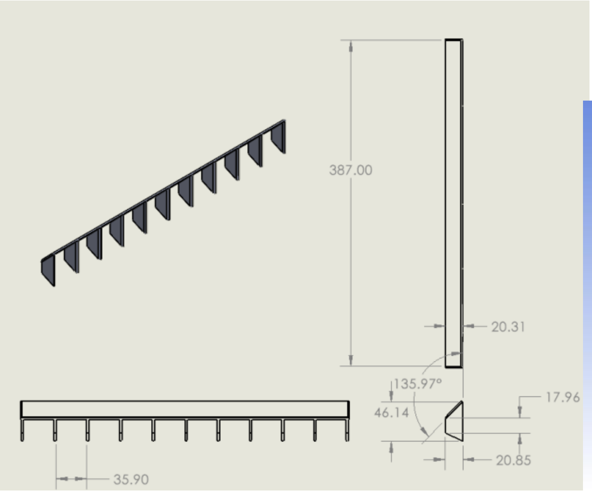


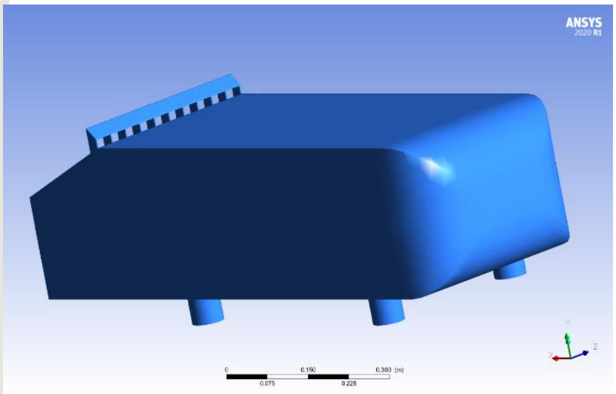
(c) (d)

Figure 5 (c)wake component A with dimension (d) wake component A is mount on the Ahmed body(Model 2)

Wake component (a) is connect on rear section of the Ahmed body same as the above figure. Component is connect back side because the component direct effect the wake and vortex area and air is direct act on the spoiler shape and upper air direct enter in the diffuser, and diffuser crate increase pressure and create low velocity of air. Component A on the subsection is provide for the diffuser, it is generate pressure. In this component A 12 section provide for create low velocity and increase the pressure of the air. The angle of spoiler plate is the 15°.

1.2.3 model 3 with wake reduce component (B)





(e) **(f)**

Figure 6 (e) wake component B with dimension (f) wake component B is mounted on the Ahmed body

Component B and component A is different from the size and dimension. In this component 10 section are given for diffuser. And upper plate is act as the spoiler, this spoiler plate is incline at the 45°.

1.3 Analysis setup

All the input data from this analysis is written in below In this three model is selected for the CFD simulation. model 1 spoiler has taken because the wake structure of the spoiler model 1 is compere with the model 2 and model 3. And model 1 spoiler is replace by the component (A) and (B) and compere wake structure.

1.3.1 computational domain

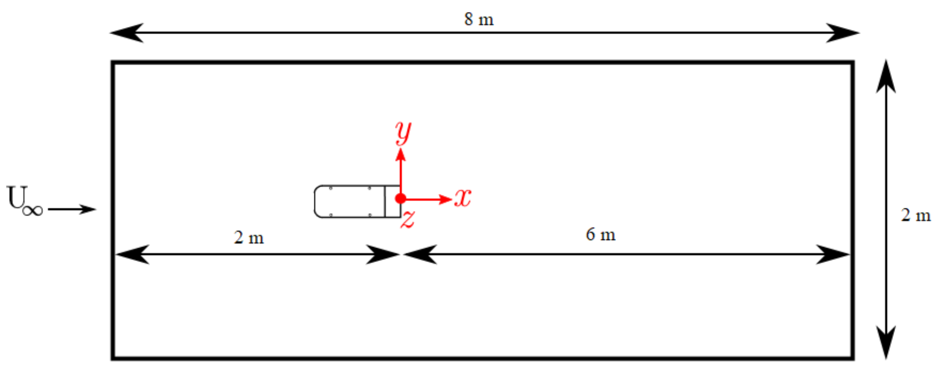


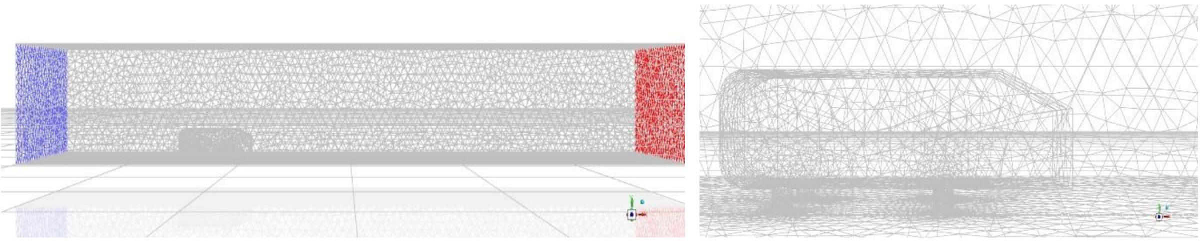
Figure 7 computational domain for Ahmed body

X=6m X= -2m
Y=1m Y= -1m
Z=1m Z= -0.01m

The size of the computational domain for analysis is given, and domain is crate for the air flow and the mesh generation.

1.3.2 Mesh generation

In the domain we will create input and output section plan and Ahmed body section, give name. next generate mesh – element size is 100 mm, Ahmed body surface is high smoothness and the mesh shape is triangle.



(a) (b)

Figure 8 (a) triangle mesh generate on the computational domain (b) triangular mesh generate on the Ahmed body

1.3.3

1.3.4 Turbulence model – larger eddy simulation

In LES, large eddies are resolved directly, while small eddies are model. Large eddy simulation (LES) therefore falls between DNS and RANS in terms of the fraction of the resolved scales. The rationale behind LES can be summarized as follows:

- Momentum, mass, energy, and other passive scalars are transported mostly by large eddies.
- Large eddies are more problem-dependent. They are dictated by the geometries and boundary conditions of the flow involved.
- Small eddies are less dependent on the geometry, tend to be more isotropic, and are consequently more universal.
- The chance of finding a universal turbulence model is much higher for small eddies.

Resolving only the large eddies allows one to use much coarser mesh and larger time-step sizes in LES than in DNS. However, LES still requires substantially finer meshes than those typically used for RANS calculations. In addition, LES has to be run for a sufficiently long flow-time to obtain stable statistics of the flow being model. As a result, the computational cost involved with LES is normally orders of magnitudes higher than that for steady RANS calculations in terms of memory (RAM) and CPU time. Therefore, high-performance computing (for example, parallel computing) is a necessity for LES, especially for industrial applications. The main shortcoming of LES lies in the high resolution requirements for wall boundary layers. Near the wall, even the 'large' eddies become relatively small and require a Reynolds number dependent resolution. This limits LES for wall bounded flows to very low Reynolds numbers ($Re = 10^4 = 10^5$) and limited computational domains.

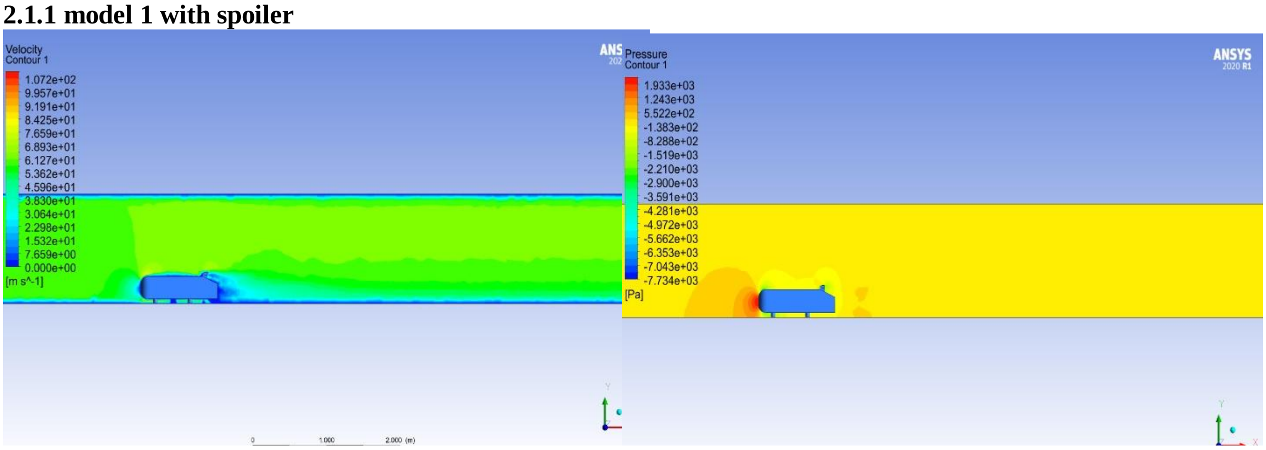
1.3.5 Air speed

In this simulation air velocity is 60 m/s. This air flow is the laminar flow and velocity method is the vortex method. In this analysis main focus on the turbulence. Solution method is the coupled, pressure is the solve by the secondary order, momentum is solve by the bounded central differencing and transient formulation is bounded second order implicit.

II. RESULT AND DISCUSSION

2.1 Analysis of models

2.1.1 model 1 with spoiler



(a) (b)

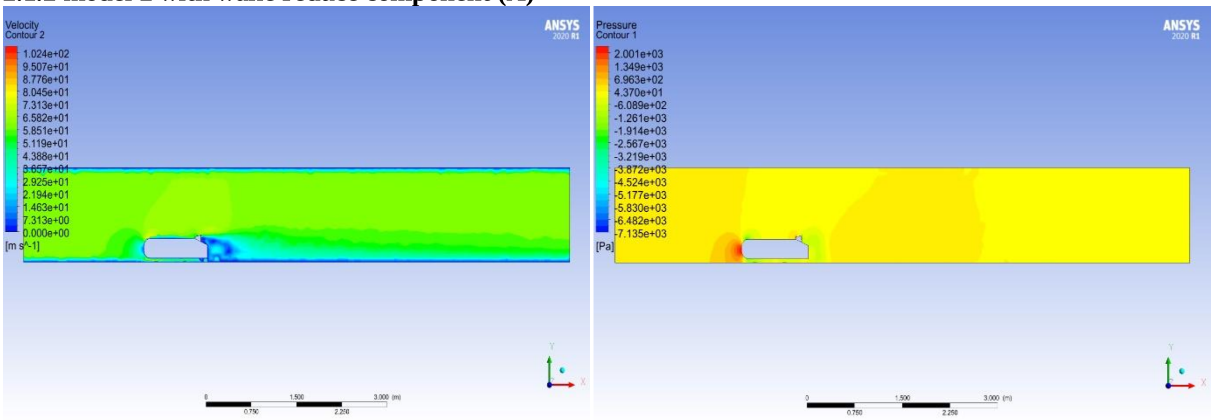
Figure 9 (a)air velocity around the model1 (b)air pressure around the model 1

Ahmed body on the join the spoiler and create driving original environment of the model outside. Air is flow from the left to right (forward) direction and air speed is 60 m/s. and the larger eddy simulation is use and subgrid scale model in we use wall adapting local eddy -viscosity model use.

Above figure show the air velocity around the model 1(Ahmed body with spoiler). the air velocity in the back of the body is very low, for the wake reduce we have to increase the low velocity area behind the model 1.

Figure (9) show the moving air pressure around the model 1. wake and vortex produce behind the model and this is reduce by the pressure increase behind model. Above figure in seen low pressure area behind the model and for the wake reduction we need high pressure create behind the model 1.
Cd= 0.4140774
Cl= -0.005066

2.1.2 model 2 with wake reduce component (A)



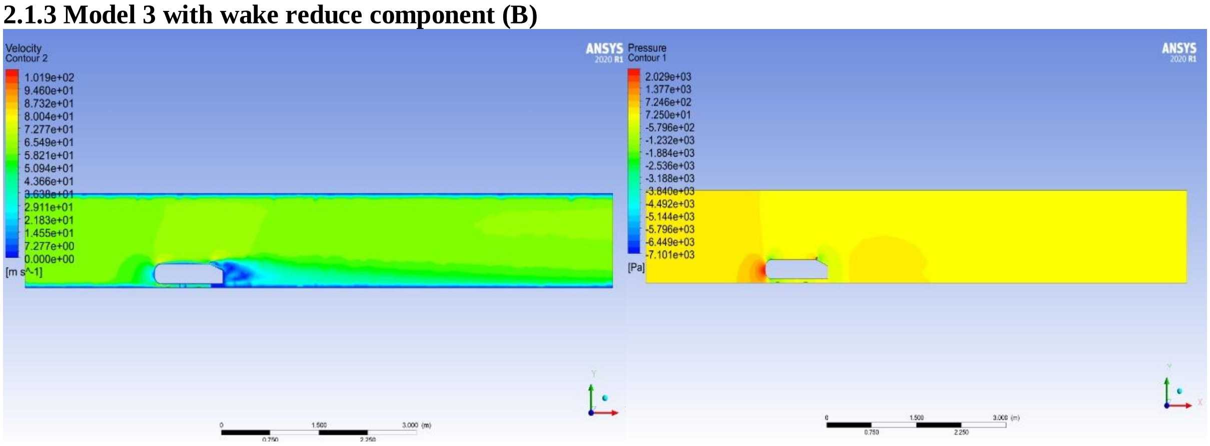
(c) **(d)**

Figure 10 (a) air velocity around the model 2 (b) air pressure on the model 2

Wake component (a) is connect on rear section of the Ahmed body same as the above figure (10). Component is connect back side because the component direct effect the wake and vortex area and air is direct act on the spoiler shape and upper air direct enter in the diffuser, and diffuser crate increase pressure and create low velocity of air.

Component (A) is generate pressure behind the Ahmed body and it is show in figure (10). this component low pressure area increase but rear side corner small pressure increase and this effect the vortex. Above figure (10) show the air velocity around the model 2. This component is generate low velocity of air behind the model and this shown in blue color.
Cd=0.348879
Cl=-0.207177

2.1.3 Model 3 with wake reduce component (B)



(e) (f)

Figure 11 *(e) air velocity around the model 3 (f) air pressure around the model3*

In this model, spoiler width is decrease and spoiler plate angle is change with the create 45° and diffuser width is also decrease. all the measurement is give figure (6). Above in this figure (11) seen component is create low velocity of air and is denoted with blue colour. low velocity air is direct effect the vortex and reduce the air velocity in the vortex and because this automatic vortex length is decrease. Above figure (11)show pressure around the model 3, In figure component is not generate increases pressure .
Cd=0.413369
Cl=-0.086142

2.2 comparison of wale structure of models

This analysis is comparison of the wake structure, and our model wake structure are given below. Figure (12) is show that the wake structure of the model 1 and also shows eddy viscosity in the x direction. In this model maximum eddy viscosity is 1.549e-02. Wake length of the model 1 is larger compere to the other models (model 2 and model3).

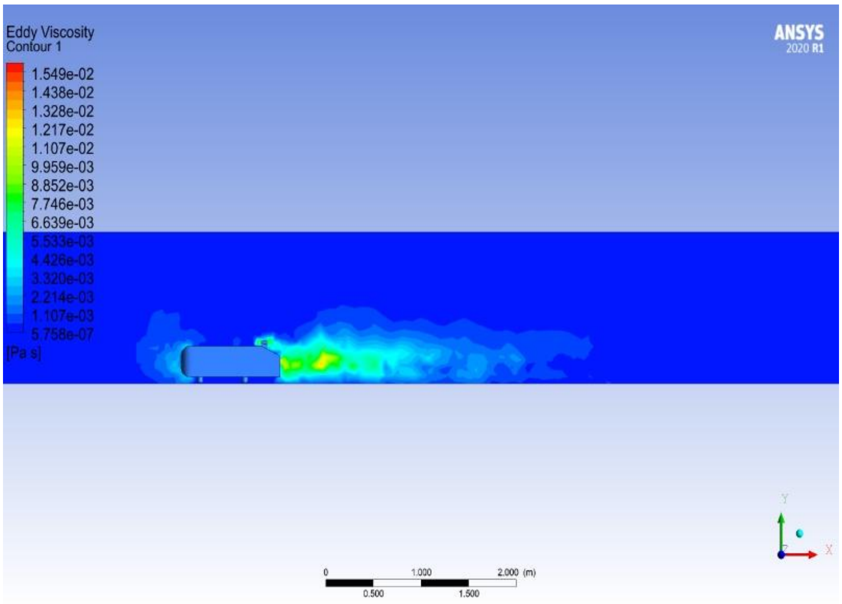


Figure 12 eddy viscosity of model 1

Below figure (13) shows that the wake structure of the model 2 and is maximum eddy viscosity is 1.627e-02. wake length is reduce compere to the model 1.in this model 2 we mount the component (A) and component is increase the eddy viscosity, this eddy viscosity through wake length is reduce.

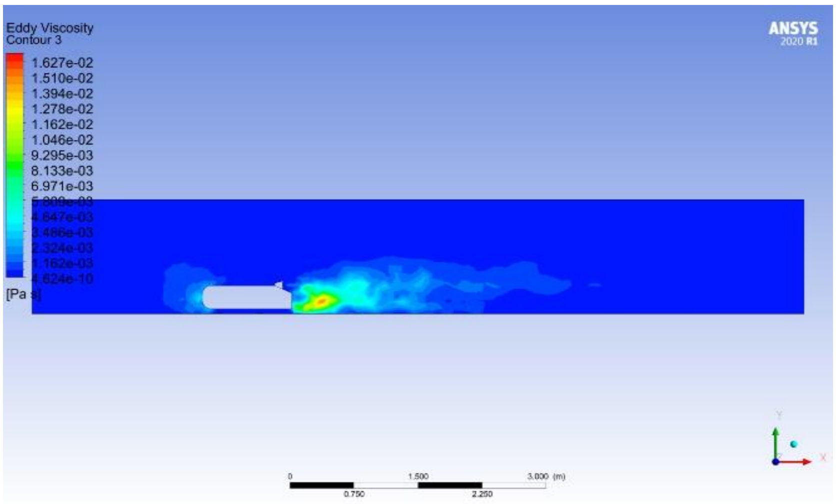


Figure 13 eddy viscosity of the model 2

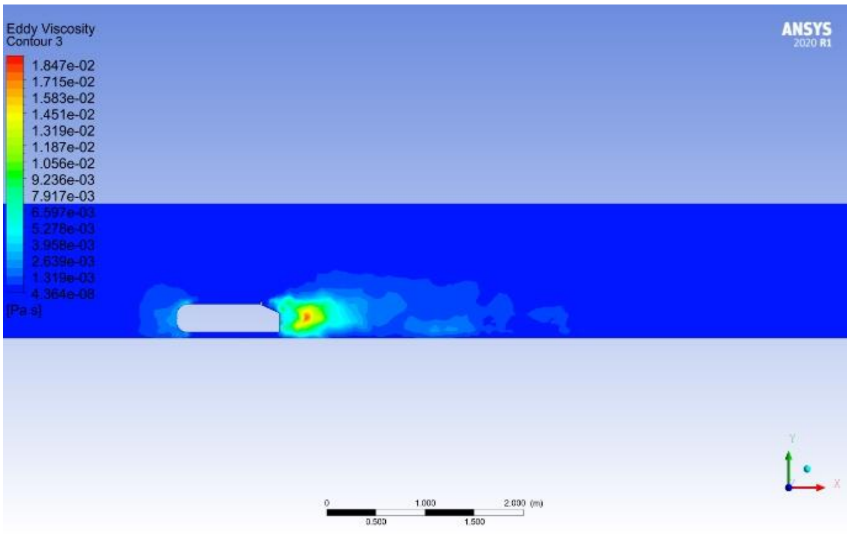
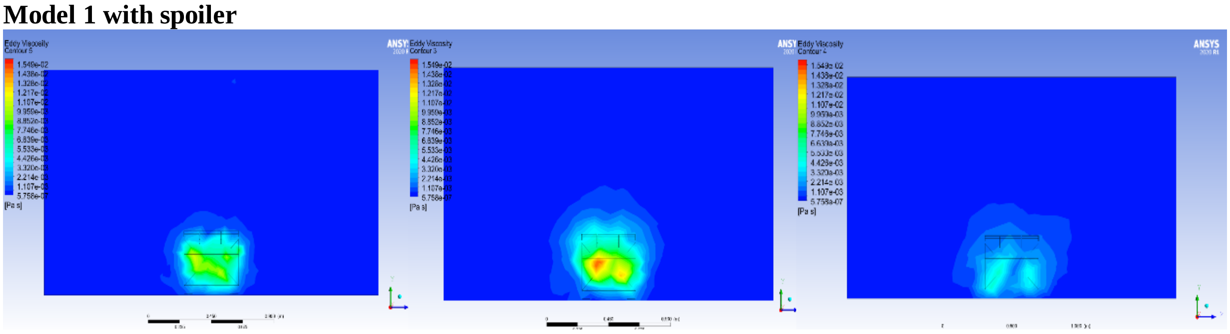


Figure 14 eddy viscosity of the model 3

This figure (14) shows that wake structure of the model 3 and shown the various eddy viscosity level. Component B is increase the eddy viscosity, this eddy viscosity is effect the vortex and is create resistance between two layer air in the vortex. The higher eddy viscosity is reduce the wake length.

2.2.1 Wake level with different plane

Model 1 with spoiler

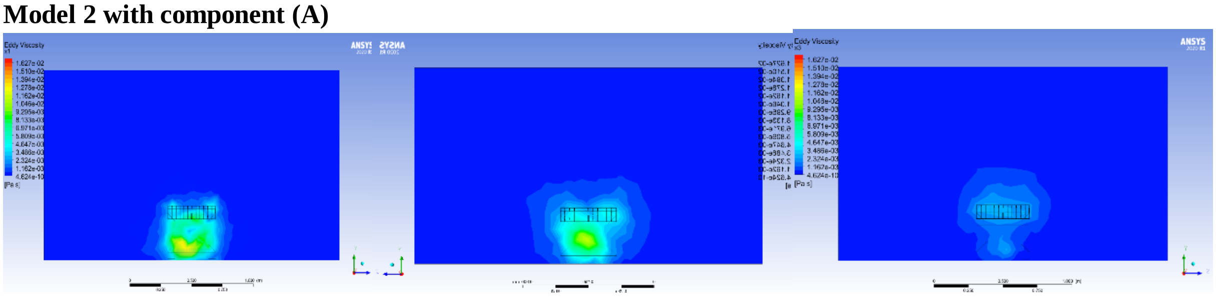


(a) 0.7m (b) 1m (c) 2m

Figure 15 (a) eddy viscosity at 0.7m plane behind the model 1 (b)eddy viscosity at the 1m plane behind the model 1 (c) eddy viscosity at the 2m plane behind the model1

Above figure show that the wake structure with different y-z plane. Three y-z plane are create behind the model according to the 0.7m, 1m, 2m on distance. Figure (a) is 0.7m y-z plane and it is show that wake level at the 0.7m, in this plane maximum eddy viscosity is 1.217e-02. Figure (b) is show the second plane at the 1m behind the model 1 and in this plane maximum eddy viscosity is 1.549e-02. Figure (c) is the third plane at 2m of model 1and in this eddy viscosity level is 4.429e-03 .

Model 2 with component (A)

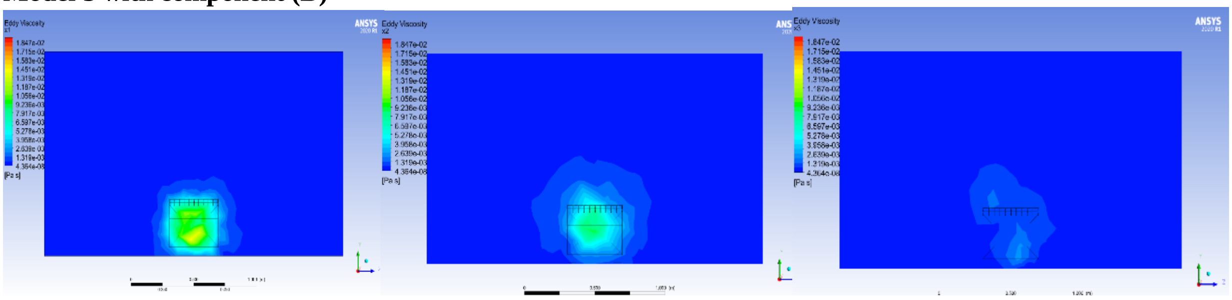


(d) 0.7m (e) 1m (f) 2m

Figure 16 (d) eddy viscosity at 0.7m plane behind the model 2 (e)eddy viscosity at the 1m plane behind the model 2 (f) eddy viscosity at the 2m plane behind the model 2

Above figure(16) show that wake level behind the model 2. Figure (d) is the plane of 0.7m behind the model 2 and it is show eddy viscosity level is 1.278e-02(maximum). Figure (e) is the second plane at 1m distance of model 2 and In this 1.278e-02 eddy viscosity generate. Figure (f) is the 2m plane behind the model 2 and in this eddy viscosity level is 2.324e-03.

Model 3 with component (B)



(g) 0.7 m (h) 1m (i) 2m

Figure 17 (g) eddy viscosity at 0.7m plane behind the model 3 (h)eddy viscosity at the 1m plane behind the model 3 (i) eddy viscosity at the 2m plane behind the model 3

Above figure shows that wake level behind the model 3. Figure (g) shows the 1.451e-02 eddy viscosity (maximum) level at the 0.7m behind the model 3. Figure (h) in eddy viscosity level is 9.238e-03(maximum). Figure (i) in eddy viscosity level is 2.639e-03.

Table 2 comparison of eddy viscosity in different plane of the models

	0.7 m plane	1m plane	2m plane
Model 1	1.217e-02	1.549e-02	4.429e-03
Model 2	1.278e-02	1.278e-02	2.324e-03
Model 3	1.451e-02	9.238e-03	2.639e-03

Above comparison through we have observe the model 3 is higher wake length is reduce compare to the model 1 and model 2. (model 3)In the y-z plane at distance of the 0.7m higher eddy viscosity is generate, because of this model 3 in air velocity is reduce and wake and vortex is also reduce.

III. CONCLUSION

The Ahmed body four models are investigated using the larger eddy simulation model in the Ansys software and we are observed the various wake structure and various wake length. We also see the different eddy viscosity level of the wake from different distance. All the result data is given below table 3.

Table 3 comparison of the Cd, CL and max vortex of models

	Model	Maximum vortex (kg/m-s)	Co-efficient of drag (Cd)	Co-efficient of lift (Cl)
1	Original Ahmed body	0.0066955	0.3766	0.2641
2	Model 1 with spoiler	0.0215807	0.4140	-0.005060
3	Model 2 & component A	0.0029987	0.3488	-0.2071
4	Model 3 &component B	0.0041811	0.4133	-0.0861

With the help of the table from the upper we know that low maximum vortex energy produce by the model 2 and also low Cd produce by the model 2 but model 2 is high Cl compare to the model 1and 3. Lowest Cl produce by the model 1 (spoiler model) but this replace by the wake component A & B , this two between lowest Cl is the model 3. Above comparison through low length of vortex produce by model 3.

- Component A is reduce the maximum vortex energy of model 3, model 3 is the lower Cd compare to the model 2.

In this analysis we will reduce the maximum vortex length by model 3 but model 3 in increasing the vortex energy. Longest length of vortex is create by original Ahmed body.

REFERENCES

[1]. Anish1, S. V. (2017). MODELLING AND ANALYSIS OF A CAR FOR REDUCING AERODYNAMIC FORCES. international journal of engineering trend and technology.
[2]. Bayraktar, I. (2001). Experimental and Computational Investigation of Ahmed Body for Ground Vehicle Aerodynamics. SAE.
[3]. Suraj Pal Singh1, A. S. (2016). Design and Simulate an Aerodynamic Car Body for The Maruti Suzuki 800 With Less Coefficient of Drag. International Research Journal of Engineering and Technology .
[4]. Rubel Chandra Dasa, ∗. M. (2017). CFD Analysis of Passenger Vehicleat Various Angle of Rear End Spoiler. ELSEVIEER,.
[5]. A.rao, G. (2018). on the two flow states in the wake of a hatchback ahmed body . Journal of Wind Engineering & Industrial Aerodynamics.
[6]. aljure, D. (2014).). flow and turbulernt structures around simplified car models. elsevier.
[7]. Alvin Gatto1, A. •. (2020). Influence of Rotating Wheels and Moving Ground Use on the Unsteady Wake of a Small Scale Road Vehicle. springer.
[8]. bearman, p. (1984). some observations on road vehicle wakes. SAE international .
[9]. Beaudoin, J.-F. (2008). Drag and lift reduction of a 3D bluff body using flaps. springer .
[10]. Becker, H. L. (2003). Flow and Turbulence Structure in the Wake of a Simplified Car Model. SAE.
[11]. c. arrighi, j. a.-h. (2015). drag and lift contribution to the incipient motion of partly submerged flooded vehicles. journal of fluids and structures.
[12]. Conan, B. (2010). Experimental aerodynamic study of a car-type bluff body. springer.
[13]. Cooper, k. (1993). BLUFF-BODY AERODYNAMICS AS APPLIED TO VEHICLES . Journal of Wind Engineering and Industrial Aerodynamics.
[14]. Dasa, R. C. (2017). CFD Analysis of Passenger Vehicleat Various Angle of Rear End Spoiler . Elsevier.
[15]. Davidson, S. K. (2004).). Large-Eddy Simulation of the Flow around Simplified Car Model. SAE.
[16]. G. Bonnaviona. (2017). On multistabilities of real car's wake. Journal of Wind Engineering and Industrial Aerodynamics.
[17]. Gavin Dias1, N. R. (2016). Aerodynamic Analysis of a Car for Reducing Drag Force. . IOSR Journal of Mechanical and Civil Engineering ,.
[18]. George, E. G. (1993). Measurements in the Unsteady Near Wakes of Ground Vehicle Bodies . SAE international.
[19]. H. Lienhart, C. S. (2002). Flow and Turbulence Structures in the Wake of a Simplified Car Model (Ahmed Modell). springer.
[20]. Hitoshi Fukuda, K. Y. (1995). Improvement of vehicle aerodynamics by wake control . ELSEVIER.
[21]. KANG, S. O. (2012). ACTIVELY TRANSLATING A REAR DIFFUSER DEVICE FOR THE AERODYNAMIC DRAG REDUCTION OF A PASSENGER CAR.
[22]. kurec, K. (2019). The influence of different aerodynamic setup on enhancing a sports cars braking. elsevier.
[23]. muyl, F. (2004). Hybrid method for aerodynamic shape optimization in automotive industry. ELSEVIER.
[24]. ONORATO, M. (1986). EXPERIMENTAL ANALYSIS OF VEHICLE WAKES . Elsevier.
[25]. S. R. Ahmed, G. R. (1984). Some Salient Features of the Time -Averaged Ground Vehicle Wake . SAE international.
[26]. S.R.Ahmed. (1983). Influence of Base Slant on the Wake Structure and Drag of Road Vehicles . ASME.
[27]. T.AVADIAR, M. t. (2018). charaterisation of the wake of the drivaer estate vehicle. journal of wind engineering and industrial aerodynamics.
[28]. tsai, c.-H. (2009). Computational aero-acoustic analysis of a passenger car with a rear spoiler. Elsevier.