Aerodynamics Analysis of Passenger Vehicles using CFD Modeling.

DipakShelar[1], Abhishek Dabb[2], Aniket Phatangare[3], Rohit Gupta[4]

[1]*Asst. professor, Mech. Engg. Dept, DYPIET Maharashtra India ,**shelardipak08@email.com***
[2] *Asst. professor, Mech. Engg. Dept, DYPIET Maharashtra India **abhishek.280489@gmail.com***
[3]*Asst. professor, Mech. Engg. Dept, DYPIET Maharashtra India, **aniphatangare@gmail.com***
[4]*Asst. professor, Mech. Engg. Dept, TIEIT Bhopal MPIndia ,**r.guptajnv2008 @gmail.com***

ABSTRACT

Now a days computational fluid dynamic tools are very extensively used to minimize the drag force. Passenger vehicle experiences the drag force at high velocity even though lift force is of no significance for passenger vehicle. Also CFD tool provides solution to complex problem related most of area. This tool is very accurate and gives most accurate results. In thesis work aerodynamics effects on the passenger vehicles are presented. The study incorporates model making of passenger vehicles, Computational Fluid Dynamic analysis and simulation to predict the best aerodynamically performing vehicle. As drag force increases it lead to the consumption of more fuel so we try to minimize the drag force. some techniques are enlisted in this dissertation work. This work provides basic fundamental procedure to make aerodynamic analysis of any vehicle. This will helpful for researcher those who are working in this area.

Keywords: Model Making, CFD Simulation, Aerodynamic Performance..

---

1. INTRODUCTION

1.1Basic Principle of Lift and Drag

An important principle of aerodynamics is Bernoulli's principle. It states that as there is an increase in flow velocity of a gas there is a decrease in pressure for a fixed volume. Also energy conservation statement support to this. This law is used extensively in aeronautical applications. The shape of an airplane wing causes the air to flow faster over the top surface than the bottom one. According to the Bernoulli's principle lower pressure on the top surface compared to the bottom surface get generates and due to this lift is created.Reverse phenomena is used in automobile vehicle. if the wing were turned upside down then the down force will be generated and is useful in vehicles design as it pushes the tires onto the road giving aerodynamic grip which is different from mechanical grip.

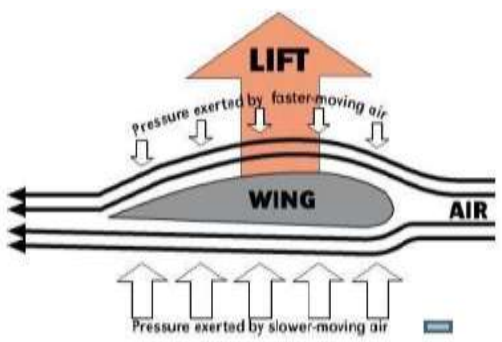


Fig.1 Lift and Drag Phenomena

The diagram above illustrates the Bernoulli principle on an aircraft wing. [8] Automotive aerodynamics is the study of the air behavior on the road vehicles. Its aim are reducing drag and wind noise, minimizing noise, creation of down forces and other causes of aerodynamic instability at high speeds. For some classes of racing vehicles, it may also be important to produce down force to improve traction. An aerodynamic Vehicle will integrate the wheel arcs and lights into the overall shape to reduce drag. It will be streamlined; no sharp edges crossing the wind stream above the windshield and will feature a sort of tail called lift back, try to reduce rear area. It will have smooth floor to support the Venturi effect and produce desirable downwards aerodynamic forces. The air that rams into the engine is used for cooling, combustion, then reaccelerated by a nozzle and then ejected under the floor. These vehicles need a seal between the low pressure region around the wheels and the high pressure around the gearbox. They all have a closed engine bay floor. The suspension, door handles, the antenna, and roof rails can have a streamlined shape. The side mirror can only have a round fairing as a nose. Air flow through the wheel-bays is said to increase drag though vehicles need it for brake cooling and many vehicles emit the air from the radiator into the wheel bay.

1.2 Comparison with aircraft

There is difference in automotive aerodynamics and aircraft aerodynamics in several ways. First one the characteristic shape of a road vehicle is much less streamlined compared to an aircraft. Second one is the vehicle operates very near to the ground, rather than in free air. Third one is the operating speeds are lower. Fourth one is a ground vehicle has fewer degrees of freedom than an aircraft, and its motion is less affected by aerodynamic forces. Fifth one passenger and commercial ground vehicles have very specific design constraints such as their intended purpose, high safety and certain regulations

1.3Methods to Study Aerodynamics

Automotive aerodynamics is studied using a) computer Modeling b) wind tunnel testing c) track test. Since drag force is the function of velocity of air; force increase with the square of velocity, down force increases with the square of the vehicles speed and requires a certain minimum speed in order to produce a significant effect. But some vehicles have rather unstable aerodynamics, such that a minor change in angle of attack or height of the vehicle can cause large changes in the down force. In the very worst cases this can cause vehicles to experience lift, not down force.

2. LITERATURE REVIEW

2.1 Primary Requirements

Primarily requirements before performing CFD analysis on a vehicle: (1) A closed volume must be present. (2) No internal component is needed for external flow analysis (3) No intersecting surfaces or overlapping should be present. Residual is of the order of 10^-3 to 10^-4 residual units. Author used residual of the order 10^-3 to10^-5 which is very low and sufficient. Author used flow condition like: segregated flow, turbulent, steady flow, constant density flow, K-epsilon turbulence Model which is more reliable. He also suggested that constant density flow is better than ideal flow. His Model is not used for high match number. Drag and down force were calculated from the frontal area and from which drag and lift coefficient are fond out. To minimize these forces different angle of attack are used for the rear and front wing. He placed the both wing for formula 1 vehicle at different angle which give clear indications of which will minimize the drag and gives sufficient down force. Wings are rotated from 3 to 7 degree by 1 degree increments from their initial positions. So that it will increase the down force. He neglected the modification below 3 degree because less drag and down force is generated. As far rear wing is concern it is not greatly affected on the down force but drag is going to minimize slowly. Therefore higher angle may be suitable. Conclusion from this is that rear wing is positioned at more than 5 degree angle of attack. Front wing does not show any significant roll on drag and down force. [1]

Wing system consist of span wise camber, various combinations of basic lifting surfaces, ring wings and C-wings.It also consist of winglets ,endplates, other tip devices. Thick wings optimized for minimum lift-induced drag or other parameters are sometimes called warped wings, because they have a distribution of camber and twist to fulfill the condition of equidistribution of the wing loading. In this paper aerodynamic performance of straight wing and non-planer c wing are compared on the basis of parameter like drag, lift, lift to drag ratio, induced drag of various angle of attack, lifting area using the computational fluid dynamics technique. From these simulations we found that lift to drag ratio for the c wing is greater than straight wing for all angle of attack. But overall performance of c wing is better than the straight wing in the following manner like structural characteristics, span constrained air-plane. It suffers the problem of large torsional inertias and coupling. To avoid this multiple ailerons are used. C-wing also improves weight on wing which leads to a small improvement in total drag, still the L/D ratio is high, and since tip vortices is reduced. C-Wing has weak over straight wing like maintenance complexity and manufacturing which may be neglected due better aerodynamic performance.[2]

2.2 Modification Made to Improve Performance

The first solid Model is actual basic SAE vehicle with approximate dimension. In second Model there is cutout in the firewall. In third Model front wing as well as cutout in the firewall is present. Velocity Contours: Higher velocity for Model 2 is observed at cut out section. And for the Model 3 it is at the stagnation point near the base. Along Symmetry: air flow is gets obstructed by firewall, but Rounded edges at the front increases the air flow, total pressure is found to less for Model 2 at the driver sea and for the Model 3 it is very less. As actual or Model 1 has more firewall area so it opposes more air to flow so it create more drag force. This is overcome in Model 2 by providing the cutout in the firewall.so it get more space to flow the air. This lowers the pressure and raises the velocity of air at firewall. More pressure is generated near the front wheel.

Model 1 Model 2 Model 3

Fig. 2 Model Modification

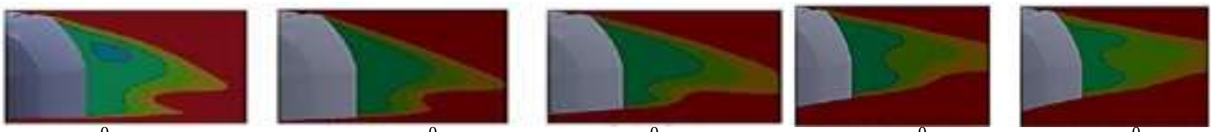
Down force is created due to Underside of the race vehicle. Floor panel height of the vehicle should be reduced to reduce the lift. Stability is achieved due to Front wing. It also seen that co-efficient of lift is decreased from 0.2 for the Model 1 to –0.25 for the Model 3 with front wing. It is because the floor panel height of the first 2 Model is higher than the Model 3 and it affects on the lift coefficient. Vehicle stability is achieved by reducing the lift. Cd always closely related to the on shape of the vehicle. Shape of vehicle is modified by cutting the firewall and providing the font wing. From simulation Cd for Model 3 is lower, as that of standard race vehicle. Cd for Model 3 is to 0.7 and for Model 2 is 0.75, Model 1it is 0.85. Cutting firewall provides the passage to the air as well as air to flow with surface. Modifications are done in order to increase the aerodynamic performance of SAE car. Comparative study is done on three vehicle Models by using CFD analysis. Cd is getting reduced from 0.85 for Model 1 to 0.70 for Model 3, whereas negative lift. Coefficient is increased from 0.2 for Model 1 to 0.25 for the Model 3. The pressure at firewall

 ISSN: 2321-8169
833 – 839

is reduced for the Model 2 and Model 3 due to cutting the firewall, and flow is attached to body helps to reduce the drag. Velocity of air is increase below the stagnation point of vehicle from 26m/sec to 32m/sec for Model 3. Due to this pressure at the driver head region is reduced from 340Pa to 80 Pa. for Model 3. Model 3 gives less drag and lift shows better aerodynamics characteristics than other two Models. [3]

Main advantages of CFD simulation are accuracy and robustness. In these simulations Galerkin or least square based CFD solver is used. It yields accurate result on highly distorted grid. CFD for the Ahmed body and the Asimo have been performed, which gives good result for pressure coefficients and drag. The results show a noticeable difference in the lift force between the non-deformable and the deformable rear wing. [4]

As we know aerodynamic drag is more important to accelerate the vehicle. 50% to 60% energy is lost to overcome the drag. Racing car need faster acceleration and this can be achieved by aerodynamic shape. Some methods are used such as exhaust gas redirection towards the rear end zones and rear under body modification. Before modification drag is 0.3233 and after under body modification it is reduced by 22.13% and by exhaust gas recirculation it is by 9.5% reduced.3.3% drag reduction can be achieved by exhaust gas recirculation. This technique will help researcher to improve aerodynamic performance drag is also reduced at rear end. As slicing is done at rear under body end more air is flow so low pressure zone get reduced. Hence drag force get reduced. Next method is that exhaust gas recirculation at the low pressure zone. So that it can reduce the negative pressure and drag force.



B=0^0 B=2.5^0 B=5^0 B=10^0 B=12.5^0

Fig.3 Rear Underbody Modification

Main thing have to consider in aerodynamic design is that streamline should be attached to body there should not be separation. Also smooth surface, avoid sharp fillet and few joint can improve aerodynamic performance. [9]

2.3 Aerodynamic Parameter Effects on Bridge under Crosswinds And Dynamic Response Of The Vehicle

In this paper how aerodynamic parameter effects on bridge under crosswinds and dynamic response of the vehicle are discussed.

(1) Dynamic response of bridge and vehicle are effects on the buffering force of bridge. This are considered in analysis of the vibration wind–vehicle–bridge system

(2) Aerodynamic parameters of the vehicle have no impact on the responses of the bridge but have a great impact on the responses of the vehicle

(3) To predict the aerodynamic interference between vehicle and the bridge under wind loads it is very necessary to estimate the aerodynamic parameters of the bridge and the vehicle. [6]

3. AERODYNAMIC ANALYSIS

3.1 Steps Followed In Analysis

Here in this paper I made 4 Models of passenger vehicle. First Model is simple extruded Model it does not required any special effort. But Model2, Model3, Model4 needs effort and very careful attention at each every steps. Those who want to do aerodynamic analysis of passenger vehicle needs closed control volumes of passenger vehicle.

Keeping the surfaces floating in air means it does not have closed volume. Such body is can't be analyses in the CFD. So, special care has to take while modeling the geometry. To do analysis in less time tries to avoid the more curves in geometry. Here all four models are creative.

There are also way to analyze the exiting vehicle by making the model. But those who want to analyze the new model which is inventive they needs hand on skill on modeling. Modeling is the main step in aerodynamic analysis of passenger vehicle. Further generation of mesh and applying the boundary conditions are also important.

Here I listed detailed step followed in CFD.

ISSN: 2321-8169
833 – 839

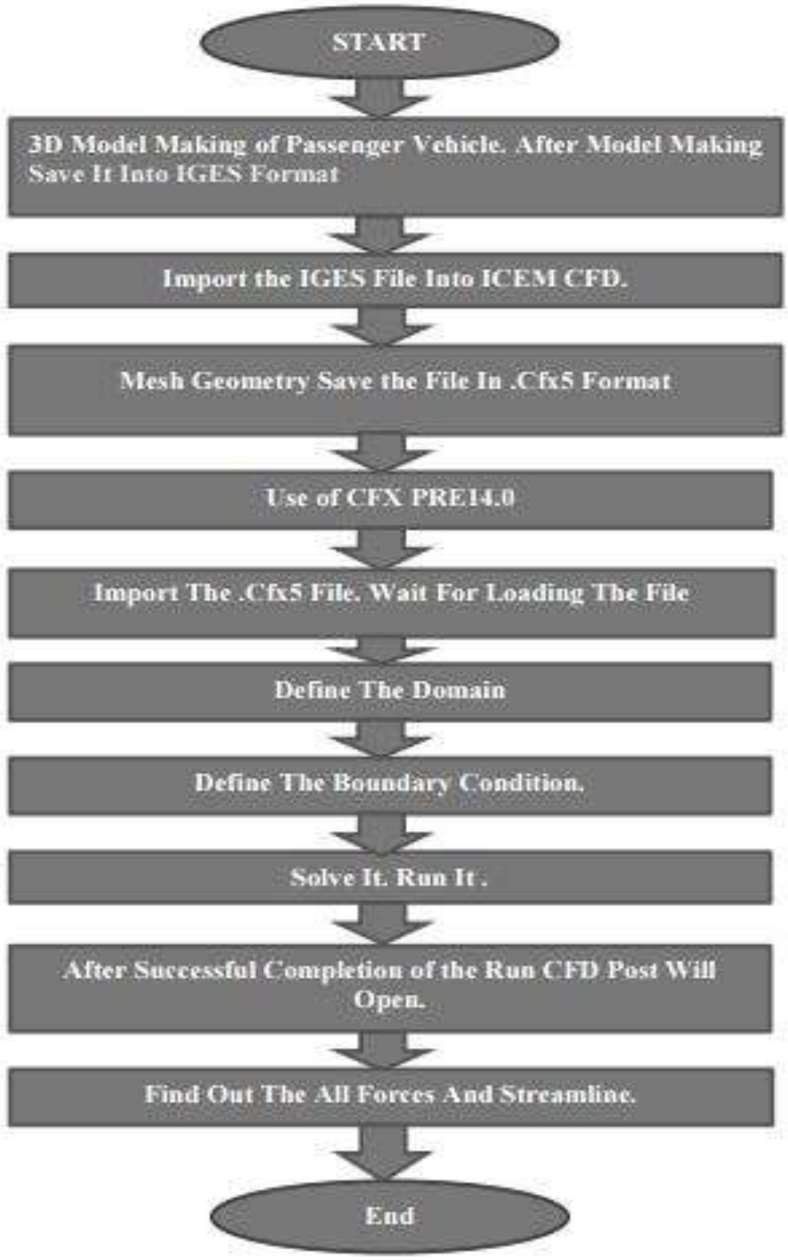


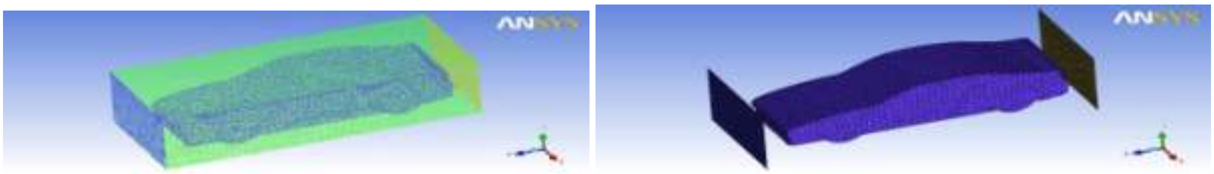
Fig. 4 algorithm for Aerodynamic Analysis of Passenger Vehicle

3.2Model Making and Simulation

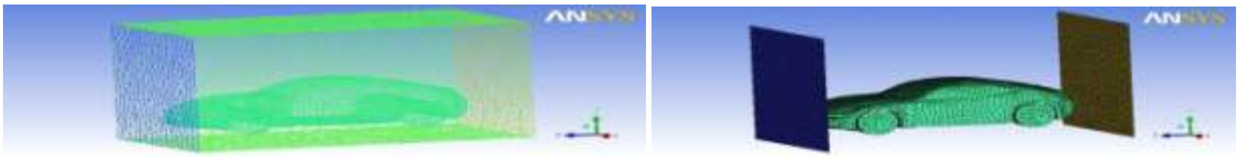
Model-1 **Model-2**

Model- 3 **Model-4**

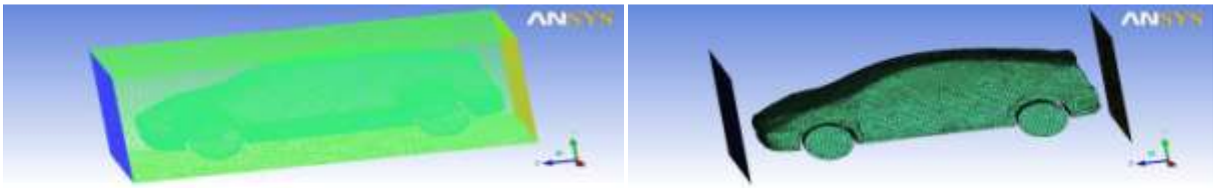
Fig. 5 Creative Model in CREO Parametric 2.03.2 Meshing of Models.



Model 1



Model 2



Model 3

Model 4

Fig. 6 Meshing the Models in ICEM CFD 14.0

3.3Boundary Conditions Applied for All Models

- *Boundary Condition for all Models*

All boundary condition applied in the CFX Pre 14.0

- Domain1=Body,

Domain Type= Solid,
Boundary Type =Wall, Material = Aluminium,
Domain Motion =Stationary Heat Transfer = None.

Boundary in domain 1
Boundary name= body,
Boundary type=wall,
Location= select all body.

- Domain 2 = All except Body,

Domain Type= Fluid,
Material = Air At 25^{0}c. Turbulence Model =K-ε Model

Boundary in domain 2
Boundary name =inlet, Boundary type=inlet,
Location= select inlet surface. Normal speed =25m/s.
Boundary name= outlet, Boundary type=outlet,
Location=select outlet surface, Static pressure =0 atm.
Boundary name =wall,

ISSN: 2321-8169
833 – 839

Boundary type=wall,
Location=select all remaining 4 surface of CV, Mass momentum option=free slip wall.
Solver control=Residual used $=10^{-4}$

4. RESULTS

All result are obtained in the CFD post 14.0

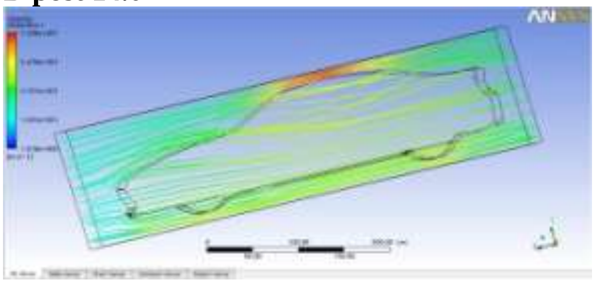


Fig. 7 Velocity streamline at velocity 25m/s =90 km/hr. Formodel 1

Similarly results are obtained for all 3 models.

Sr. No. (A)	Velocity Unit (m/sec) (B)	$.5*\varrho*A*V^2$ Unit(N) (C)	Force In Z Direction Unit(N) (D)	Coe. of Drag. C_d
1	16.67	255.89	92.12	0.360
2	19.45=70km/hr.	348.36	128.54	0.369
3	22.23=80km/hr.	455.06	127.01	0.378
4	25.00=90km/hr.	575.53	218.125	0.379
5	27.78=100km/hr.	710.65	270.75	0.381

Table 1Drag Force In Z Direction At Different VelocityFor Model 1

Sr. No. (A)	Velocity Unit (m/sec) (B)	$.5*\varrho*A*V^2$ Unit(N) (C)	Force In Z Direction Unit(N) (D)	Coe. of Drag. C_d
1	16.67=60km/hr.	306.27	82.08	0.268
2	19.45=70km/hr.	416.91	112.98	0.271
3	22.23=80km/hr.	544.60	150.8	0.277
4	25.00=90km/hr.	688.78	192.85	0.280
5	27.78=100km/hr.	850.40	245.76	0.289

Table 2Drag Force In Z Direction At Different Velocity For Model 2

Sr. No. (A)	Velocity Unit (m/sec) (B)	$.5*\varrho*A*V^2$ Unit(N) (C)	Force In Z Direction Unit(N) (D)	Coe. of Drag. C_d
1	16.67=60km/hr.	374.95	98.46	0.283
2	19.45=70km/hr.	473.68	137.8	0.291
3	22.23=80km/hr.	618.77	183.77	0.297
4	25.00=90km/hr.	782.59	242.6	0.310
5	27.78=100km/hr.	966.31	329.5	0.341

Table 3 Drag Force In Z Direction At Different Velocity For Model 3

Sr. No. (A)	Velocity Unit (m/sec (B)	$.5*\varrho*A*V^2$ Unit(N) (C)	Force In Z Direction Unit(N) (D)	Coe. of Drag. C_d
1	16.67=60km/hr.	322.4	99.88	0.310
2	19.45=70km/hr.	438.64	141.68	0.323
3	22.23=80km/hr.	572.99	194.81	0.340
4	25.00=90km/hr.	724.68	250.73	0.346
5	27.78=100km/hr.	894.81	314.0	0.351

Table 4 Drag Force In Z Direction At Different Velocity For Model 4

Sr. no.	Velocity (m/sec)	C_d for Mc	C_d for Moc	C_d for M	C_d for Moc
1	16.67	0.360	0.268	0.283	0.310
2	19.45	0.369	0.271	0.291	0.323
3	22.23	0.378	0.277	0.297	0.340
4	25.00	0.379	0.28	0.310	0.346
5	27.78	0.381	0.289	0.341	0.351

Table 5 Comparison Chart On The Basis of CdValues

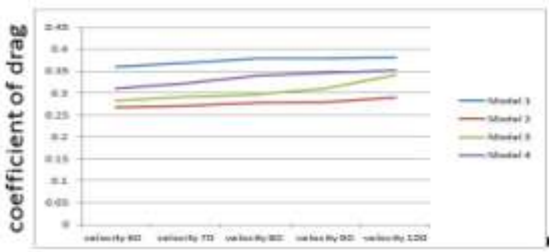


Fig No8. Graph C_d vs. Velocity

5. CONCLUSIONS

Overall summary of thesis is to get insight of aerodynamics of passenger vehicle. This thesis work gives detail method of obtaining the optimum aerodynamic performance. It is important in the aspect of fuel economy and acceleration of vehicle. Also methodology is enlisted so sophistically, so researcher can understand it easily. Some difficulties in this are skill over the software tools those are used to Model, simulate. Making the accurate geometry, meshing it are the main steps. There are governing equations of fluid dynamics behind the simulation. In turbulence flow there are two components of each velocity one is fluctuating and other is normal which leads to additional stresses. So it create closure problem. So resolve this issue we used the K-ε Model are to reach to the final result of aerodynamic performance. Some techniques are very rarely used to minimize the drag are exhaust gas recirculation at outlet. This helps to minimize the drag. Also lots of conventional methods are also explained. Finally from the result obtained after simulation in CFX Post we can conclude that,

1) Model 1 one has worst aerodynamic performance as compare to the other Model; reason behind this is surface of the Model 1 is not so smooth, No smooth curves are there, and Model looks like the box shape so it creates the more drag. Its maximum drag value are 0.380
2) Model 4 looks very attractive, good aesthetics view are obtained in Modeling, but as we look over the front it has shape such that it opposes the most of the air. So it is not good but satisfactory combination between drag value and aesthetics. Its maximum drag coefficient vale are 0.351
3) Model 2 and Model 3 has nearly similar values of the drag coefficient. As we look over the Model shape, surface finish, curves are so smooth so streamline are get attached to surface, it flows smoothly over the surface. Creates the minimum drag. Maximum values of drag coefficient values for Model2 and Model 3 are 0.289 and 0.341.

REFERENCES

[1] Satyan Chandra, Allison Lee, Steven Gorrell and C. Greg Jensen “Computer-Aided Design & Applications”, PACE (1), 2011, 1-14© 2011 CAD Solutions, LLC..

[2] C.Suresh, K.Ramesh, V.Paramaguru “Aerodynamic Performance Analysis Of A Non-Planarc-Wing Using CFD” Aerospace Science and Technology 40 (2015) 56–61H. Poor, *An Introduction to Signal Detection and Estimation*. New York: Springer-Verlag, 1985, ch. 4.

[3] Sneh Hetawal, Mandar Gophane, Ajay B.K., Yagnavalkya Mukkamala “Aerodynamic Study of Formula SAE Car” 12th Global Congress On Manufacturing And Management, GCMM 2014procedia Engineering 97 (2014) 1198 – 1207.

[4] Dr. Marc Ratzel, Thomas Ludescher, “Streamlining Aerodynamic CFD Analyses” Nafems World Congress 2013 June 9-12, 2013, Salzburg, Austria

[5] Darko Damjanović Dražan Kozak Marija Živić Željko Ivandić Tomislav Baškarić “CFD analysis of concept car in order to improve aerodynamics” A jövő járműve I 2011 01/02..

[6] Yan Han , C.S. Cai, Jianren Zhang , Suren Chen , Xuhui He “Effects of aerodynamic parameters on the dynamic responses of road vehicles and bridges under cross winds” J. Wind Eng. Ind. Aerodyn. 134 (2014) 78–95.

[7] Matteo Beccaria , Guido Buresti , Alberto Ciampa , Giovanni Lombardi ,Wolfgang Hans-Georg Paap , Andrea Vicer´e “High-performance road-vehicle optimize aerodynamic design: Application of parallel computing to car design”;Future Generation Computer Systems 15 (1999) 323–332

[8] A. Sunanda, M. Siva Nayak, “Analysis of NACA 2412 for Automobile Rear Spoiler Using Composite Material,” International Journal of Emerging Technology and Advanced Engineering Volume 3, Issue 1, January 2013.

[9] S.M. Rakibul Hassan, Toukir Islam, Mohammad Ali, Md. Quamrul Islam “Numerical Study on Aerodynamic Drag Reduction of Racing Cars” Procedia Engineering 90 (2014) 308 – 313

[10] Nikolay A. Vinnichenko , Alexander V. Uvarov , Irina A. Znamenskaya , Herchang Ay, Tsun-Hsien Wang, “Solar car aerodynamic design for optimal cooling and high efficiency” Solar Energy 103 (2014) 183–190 .

[11] Mahmoud Khaled, HichamElHage , FabienHarambat , HassanPeerhossaini , “Some innovative concepts for car drag reduction :A parametric analysis of aerodynamic forces on a simplified body” J. WindEng.Ind.Aerodyn.107–108(2012)36–47.

[12] Anderson, John D., Jr., “Fundamentals of Aerodynamics”, 2nd Edition McGraw-Hill,New York,1991

[13] H. K. Versteeg And W. Malalasekera, “An Introduction To Computational Fluid Dynamics”, The Finite Volume Method Second Edition 2007.

[14] Dr.Shrinivas Jayanti,NPTEL online course on the “computational fluid dynamics”