Innovative Computing Review (ICR)
Volume 1 Issue 2, Fall 2021
ISSN(P): 2791-0024 ISSN(E): 2791-0032
Journal DOI: https://doi.org/10.32350/icr
Issue DOI: https://doi.org/10.32350/icr/0102
Homepage: https://journals.umt.edu.pk/index.php/icr

Journal QR

Article QR

Muhammad Uzair

Article: **Computational Techniques and their Usage to Optimize Car Design for Better Performance**

Author(s): Muhammad Uzair Riaz[1], Tareq Manzoor[2], Muhammad Umair[1], Hasan Ali[3], Ammar Iqbal[3]

Affiliation: [1]University of Erlangen-Nuremberg, Erlangen, Germany
[2]Energy Research Centre, COMSATS University, Lahore, Pakistan
[3]Department of Mechanical Engineering, COMSATS University, Sahiwal, Pakistan

Citation: M. U. Riaz, M. Tareq, U. Muhammad, A. Hasan, and I. Ammar, “Computational Techniques and their Usage to Optimize Car Design for Better Performance”, *Innova Comput Rev*, vol. 1, no. 2, pp. 27–43, 2021. https://doi.org/10.32350/icr/0102/02

Estd. 1990

A publication of the
School of Systems and Technology
University of Management and Technology, Lahore, Pakistan

Computational Techniques and their Usage to Optimize Car Design for Better Performance

Muhammad Uzair Riaz[1], Tareq Manzoor[2], Muhammad Umair[1], Hasan Ali[3], Ammar Iqbal[3]

ABSTRACT: This paper describes the use of different computational fluid dynamics (CFD) models and other techniques that can be used to get low coefficient of drag. The upper body of a car is designed using surface modeling in CATIA. It is analyzed using K-epsilon realizable model in ANSYS Fluent. Changes in design are made based on the high coefficient of drag at high speeds. Then, K-epsilon realizable model is used for analysis. To get better accuracy, a refined mesh for the wake region is used. Afterwards, k-omega standard model is used. At the end, the upper body of the passenger car and its tyres are examined to get the final results of the coefficient of drag. The same design with the same meshing conditions is also analyzed in Star CCM+ to verify the results.

INDEX TERMS: ANSYS Fluent, computational fluid dynamics (CFD), coefficient of drag, Star CCM+

I. INTRODUCTION

Aerodynamic drag is the resistance force opposite to the direction of motion acting on a body moving through air. Drag force reduces the performance and fuel efficiency of road vehicles or racing cars as it is applied against the direction of their motion. “About 50% to 60% of total fuel energy is consumed to overcome the drag force, 30% to 40% to tackle road resistance and only about 10% to 20% to operate electrical appliances” [1]. Thus reduction of aerodynamic drag is of great importance for the vehicle aerodynamics, and research based efforts are being made to ensure better fuel economy [1]. Different studies on aerodynamics of vehicles has shown that “fuel economy of vehicle is improved by 1% because of 2% reduction in drag

[1]University of Erlangen-Nuremberg, Erlangen, Germany
[2]Energy Research Centre, COMSATS University, Lahore, Pakistan
[3]Department of Mechanical Engineering, COMSATS university, Sahiwal, Pakistan
*Corresponding Author: tareqmanzoor@cuilahore.edu.pk

force"[1]. Now-a-days, researchers are working on the shape of vehicles in order to improve fuel economy. The objective of aerodynamic research is to minimize drag force for faster running with low fuel consumption.

Following is the equation for drag force[2].

$$F_d = \frac{C_d \rho A V^2}{2}$$

Where

C_d= Coefficient of drag
A=Frontal area (Area projected on plan normal to direction of flow)
ρ=Density of fluid
V= velocity of fluid

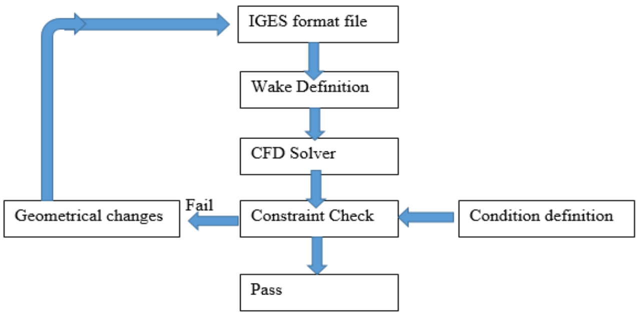

Fig. 1. Schematic Diagram

The parameter that can be varied for producing low drag force without changing the speed, is the coefficient of drag and it directly effects the fuel consumption. "It is the dimensionless number that quantifies the drag force"[3]. As it has been shown that 40% of the drag coefficient depends on the external shape. Therefore, this paper contains only an analysis of the external body of car [4]. Change in design, use of appropriate Turbulence model and mesh refinements are the parameters that can help obtain accurate values of coefficient of drag using CFD analysis and all of these will be implemented in this paper. K-epsilon model and K-omega models are frequently used for the external flows in CFD analysis [5]. So, this paper discusses the use of these two different CFD models for predicting the value of coefficient of drag. Scheme shown in Fig. 1 is used in this paper [6].

II. Initial Design and Analysis

The first design is made by using surface modeling in CATIA. Design is shown in Fig. 2.

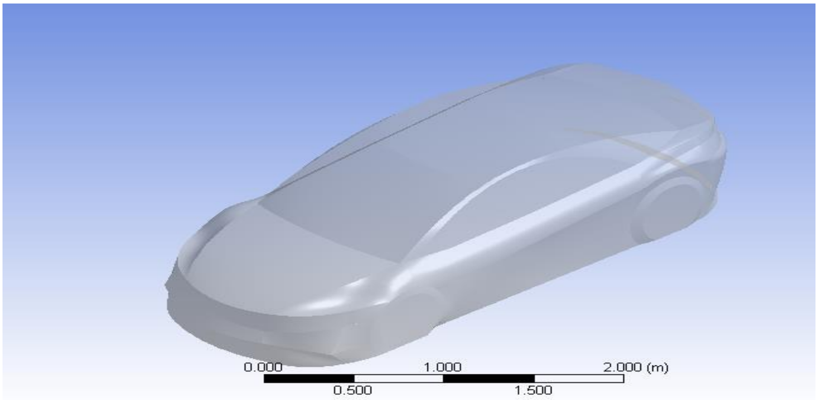

Fig. 1. Initial Car Design

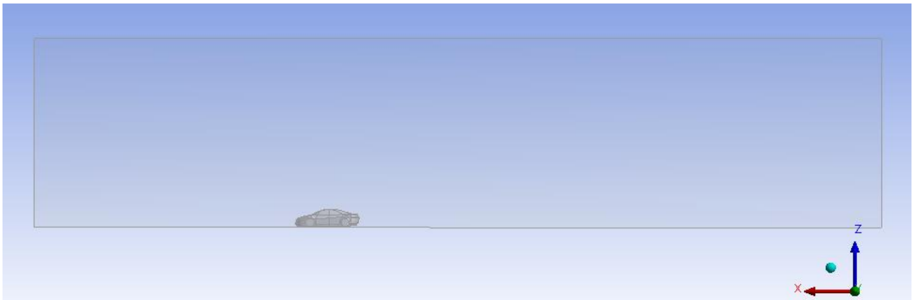

Fig. 2. Enclosure

The enclosure shown in Fig. 3 is made using Design modeler of ANSYS Fluent to create a control volume around the vehicle and the dimensions of the enclosure are as follow:.10L for the back of the car, 5L for the front of the car, 5H above of the car and 5W for the left and right sides [7]. The left side is named as Inlet velocity, the right as outlet pressure, the top as roof, the bottom as road and the sides are named as symmetry sides. The ground clearance with scaled value as that of scaling of the car is given in the analysis by giving the offset value between the car geometry and enclosure and this ground clearance is kept the same for the complete project. Car design is axisymmetric so the entire project is conducted by dividing the design into half and then utilizing this half design in order to save memory and time [8].

The design is meshed using medium meshing using the command of medium mesh in ANSYS. Mesh is prominent part of simulation. Structured mesh is used because it is simple and more popular these days [9]. Five prismatic layers are added to the upper surface of the car and five prism layers are added to the road in order to get better accuracy [10].

Then the design is analyzed using K-epsilon model because some researchers use this model [11]. Moreover, Realizable model with non-equilibrium wall function is used. They are used for external flows like flow over vehicles. “It gives us better prediction of the behavior of the turbulent boundary layer including flow separation without a significant increase in the CPU time or dynamic memory”[8]. Air is used as fluid flowing over the surface within the control volume.

A convergence criteria of 0.001 is used in the simulation. In the solution methods of ANSYS Fluent, the Least Squares cell based gradient method is used along with the coupled pressure velocity scheme. Momentum is kept first order upwind for 100 iterations and the pressure is always kept second order. After 100 iterations, momentum is also changed into second order. Turbulent kinetic energy and turbulent dissipation rate is always kept second order upwind in the setting of fluent [12]. Setting for the analysis with intensity to viscosity ratio and solving parameters are shown in Table 1[13]. Zero outlet pressure is adjusted and inlet velocity is varied on every analysis. In the first analysis, it is 50kmph.All other values are used as default values in ANSYS.

TABLE 1

PARAMETERS SETTINGS

Velocity Inlet	*Magnitude (Measured normal to Boundary)*	*Variation*
Turbulence specification method	*Intensity and viscosity ratio*	
Turbulent intensity (%)	*1%*	
Turbulent viscosity ratio	*10*	
Pressure Outlet	*Gauge pressure magnitude*	*0 pascal*
Gauge pressure direction	*normal to boundary*	
Turbulence specification method	*Intensity andvViscosity ratio*	
Turbulent viscosity ratio	5 %	

Velocity Inlet	*Magnitude (Measured normal to Boundary)*	*Variation*
Turbulent viscosity ratio	10	
Wall zones	*NosSlip*	
Fluid properties	*Fluid type*	*Air*
Density	*ρ = 1.225 (kg/m^3)*	

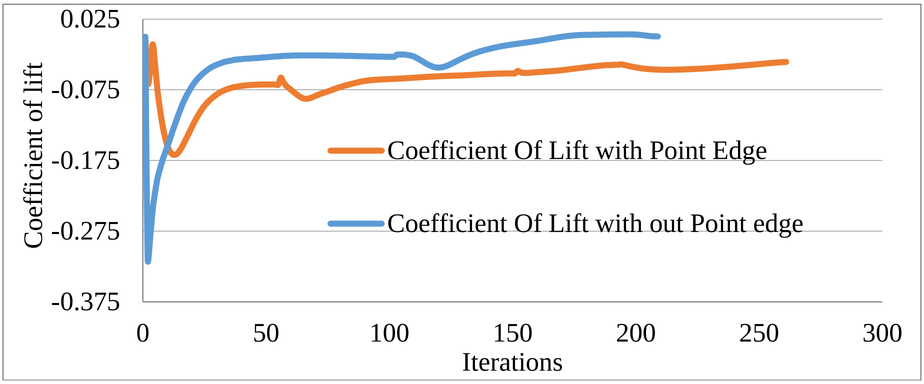

Fig. 4. Coefficient of lift with or without point edge

This analysis has been performed on design and then to reduce the coefficient of drag we made the edges on the front bumper of the car. Its frontal area is decreased by making sharp edges. Results of coefficient of drag and coefficient of lift for both designs are shown in Fig. 4 and Fig. 5.

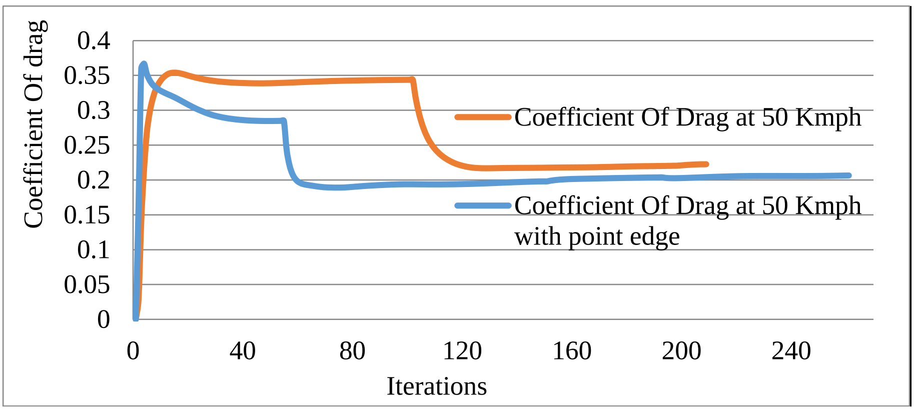

Fig. 5. Coefficient of drag with or without point edge

The analysis shows that the coefficient of drag decreases with the formation of sharp edges on bumper. But these sharp edges are not required in a passenger car. These were the analysis for the half car and in this way coefficient of drag needs to be reduced. So we focused on changing design of the car.

III. FINAL DESIGN

Then the design is made in leopard shape with smooth curves so that the coefficient of drag and pressure on the roof can be controlled. The design is shown in Figs. 6 and Fig. 7. K- Epsilon model with the above-described parameters is used in these analysis and speeds are varied. Design is analyzed using the above-described methodology at an increased speed. Change in design provides us significant decrease in coefficient of drag.

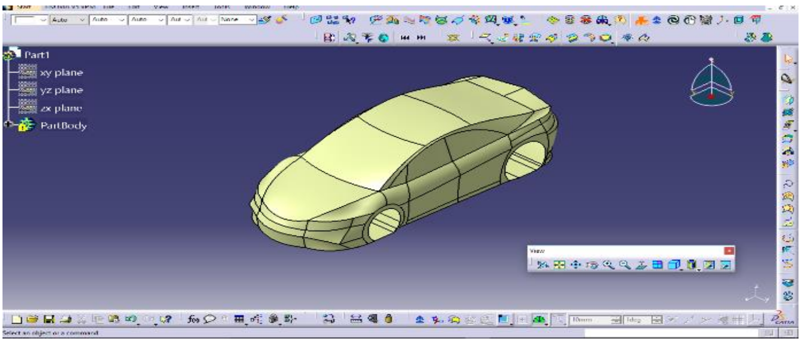

Fig. 6. Final design of car without tyres

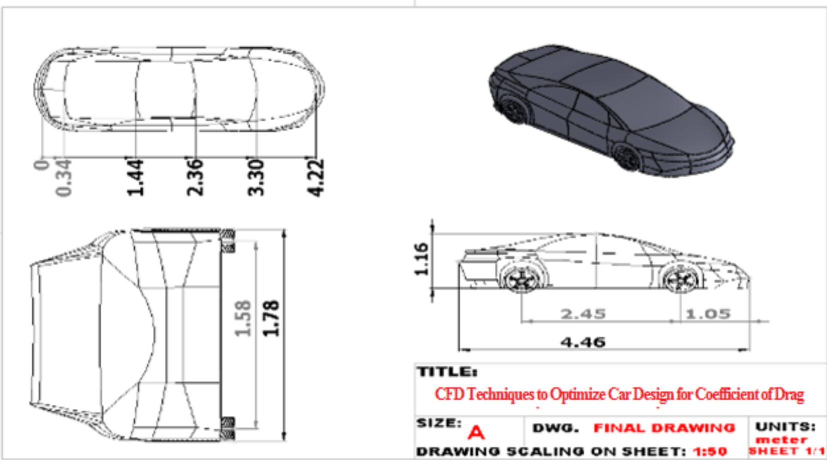

Fig. 7. Drawing representing dimensions of car

Coefficient of lift is not discussed in the complete research paper because coefficient of drag is most prominent part in the car design and the former is always kept below zero. The graphs for coefficient of drag and lift are shown in Figs 8 and 9.

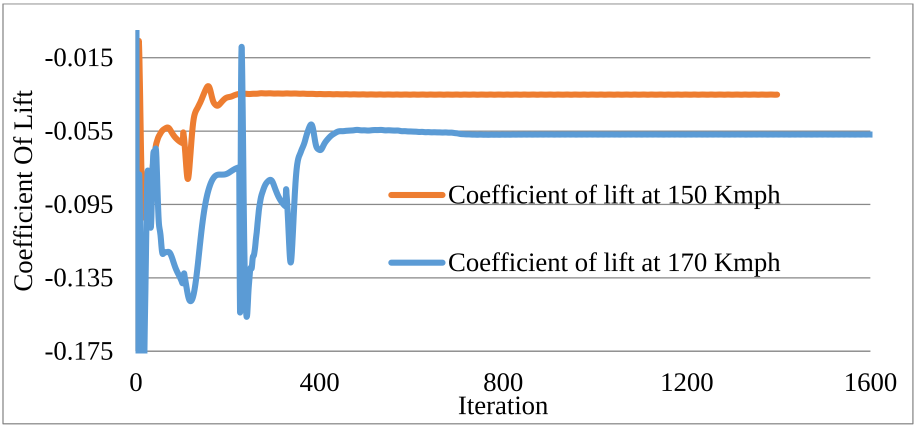

Fig. 8. Coefficient of lift at different speed

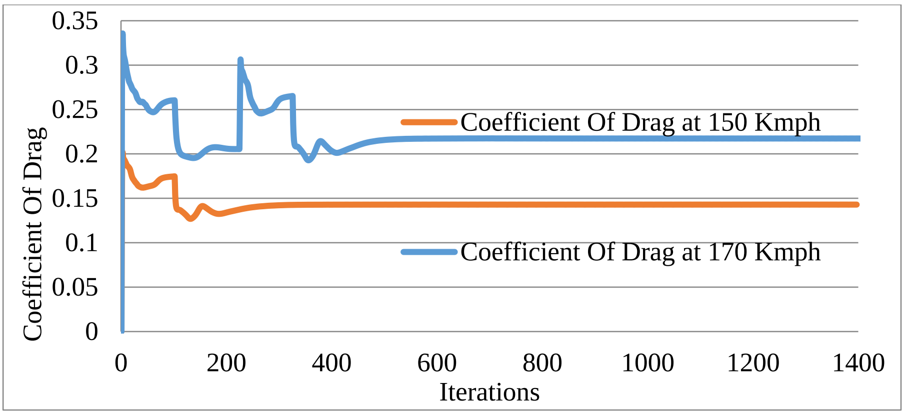

Fig. 9. Coefficient of drag at different speed

A. Mesh Refinement

Then we used the technique of mesh refinement in order to get better accuracy. Good quality meshing produces better solution. An enclosure is designed within the first control volume, and refined because the properties are changing significantly near the vicinity of the car [12]. Mesh of element size 0.01m is created for the car geometry and mesh of element size 0.1m is made for the small control volume. Other space of the control volume is meshed similarly. Mesh refinement decreases the coefficient of drag of car. Fig 10 shows the mesh refined geometry of car. This produced a significant change in the coefficient of drag and coefficient of lift of the half body car.

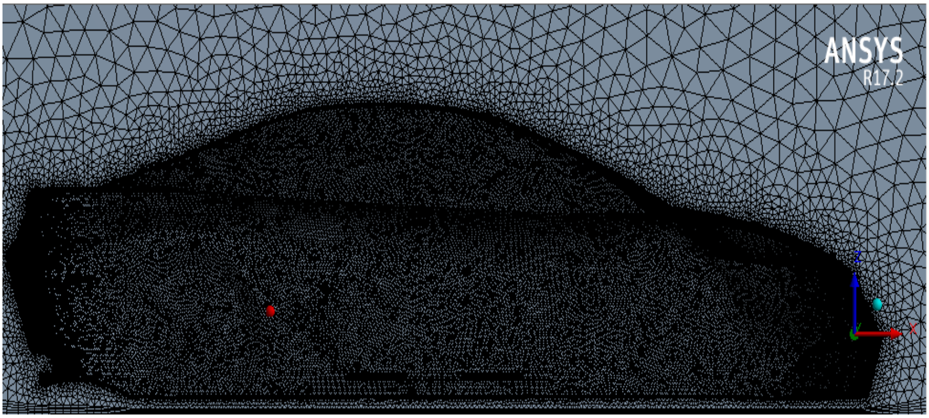

Fig. 10. Mesh refinement

The graphs developed by the data points of results are shown in Figs 11 and 12. The numbers of elements and of nodes are 1555806 and 374087 and the maximum skewness is 0.92403. These numbers show that mesh quality is good enough to conduct the analysis.

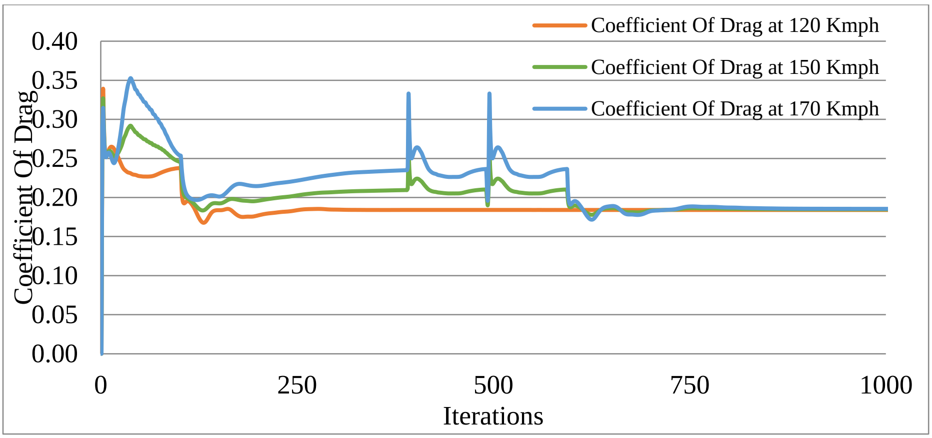

Fig. 11. Coefficient of drag with refined mesh

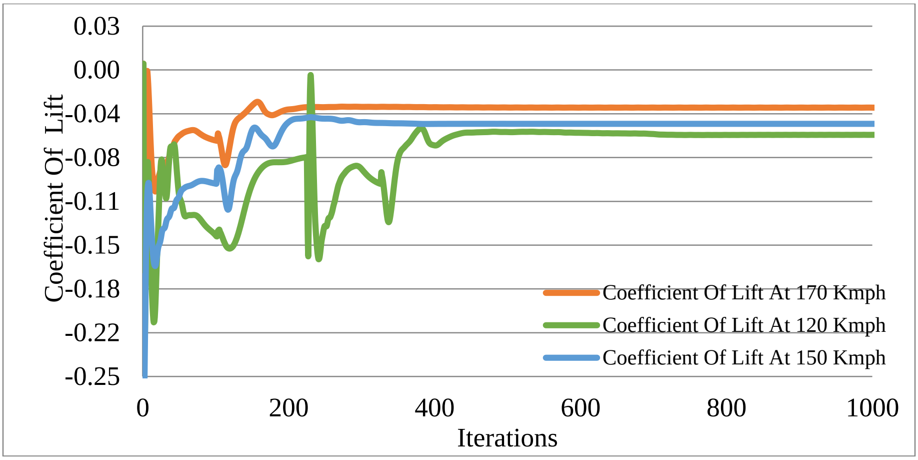

Fig. 12. Coefficient of lift with refined mesh

B. Temperature Change

Following this, the t effect of change in temperature on coefficient of drag and coefficient of lift is observed. Same values are given and mesh is used as medium mesh as was used in the first design. All other values are used as that of the above analysis. It showed that temperature does not have significant effect on coefficient of drag and coefficient of lift. Figs 13 and 14 are the graphs of analysis with change in temperature. These graphs show more value because they are performed at a high speed. But this speed is maintained for all temperatures.

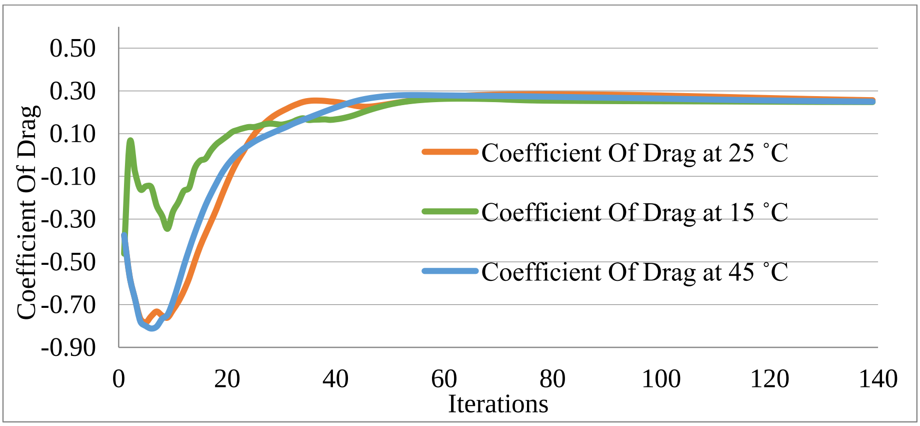

Fig. 13. Coefficient of drag at different temperature

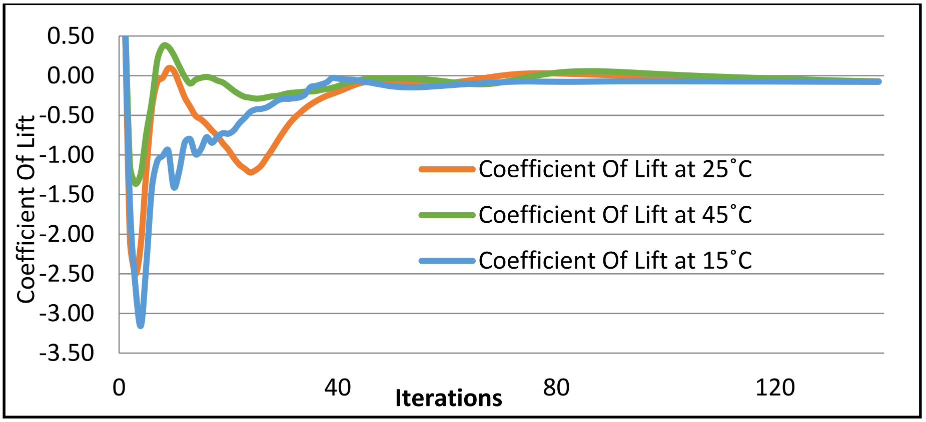

Fig. 13. Coefficient of lift at different temperatures

C. Analysis with Standard K-omega Model

Some researchers use the standard k-Omega model for analysis. Therefore, it is used for the analysis of the same design. This model provides better results. All other default values are used for the analysis. It is used to solve two equations. Turbulent kinetic energy and turbulent frequency are the parameters related to kinetic viscosity in K-Omega Model [13]. Mesh with refinement has been used for this analysis. Convergence criteria has also been changed in this analysis. For running the results to the high number of iterations we shifted the convergence criteria from .001 to .00001.

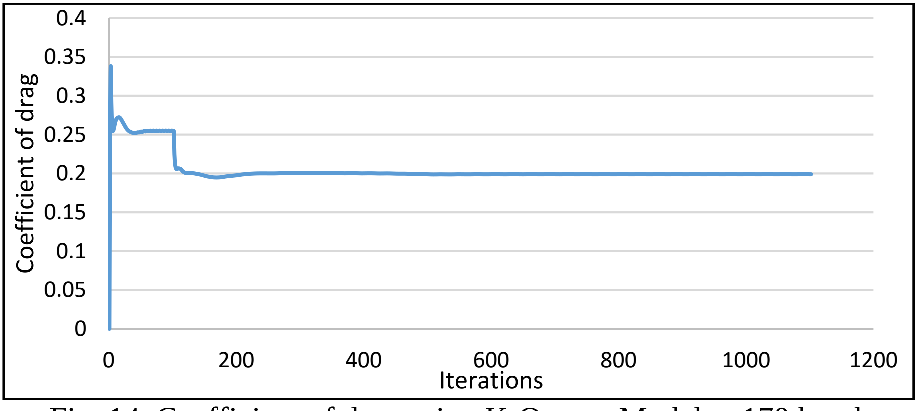

Fig. 14. Coefficient of drag using K-Omega Model at 170 kmph

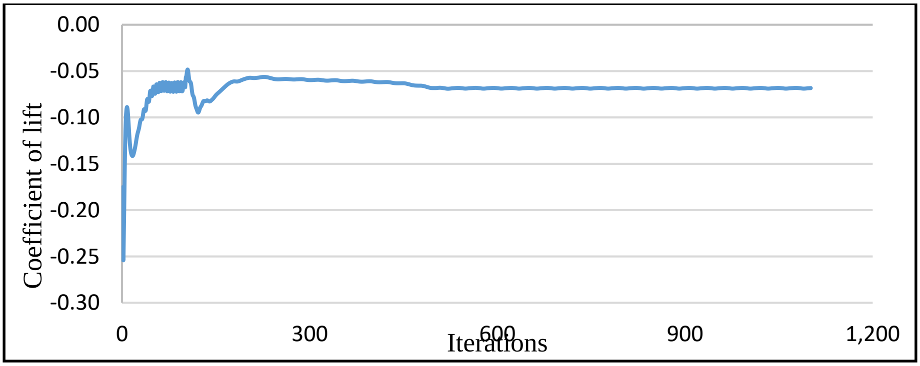

Fig. 16. Coefficient of lift using K-Omega Model at 170 kmph

In all the previous analysis, it was used as default value. There is slight change in the values of coefficient of drag and coefficient of lift. But coefficient of drag obtained by using K- Omega model is greater than that the one obtained by using K- Epsilon Model and coefficient of lift obtained by using K-Omega model is lower than that obtained by using K-epsilon model [14]. Graphs shown in Figs 15 and 16 are plotted by using data points of the results of coefficient of drag and coefficient of lift with standard K-omega model.

D. Car with Tyres

Finally, car with tyres is analyzed in ANSYS Fluent by using standard K-omega model and the graph is drawn by using the data points of results.

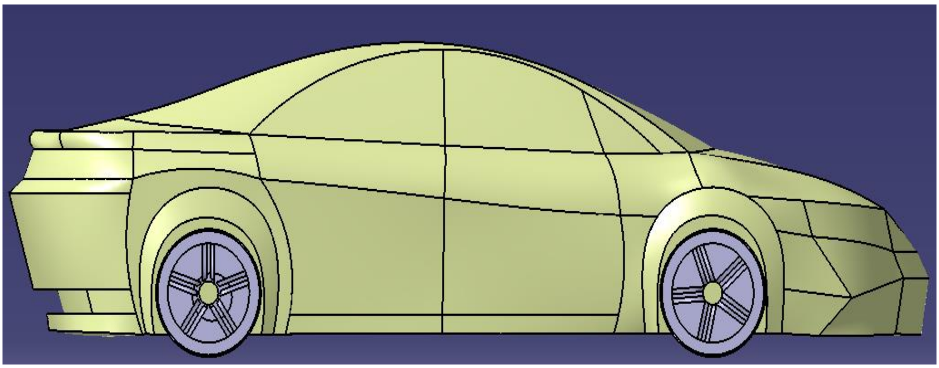

Fig. 17. Car with tyres

Fig 17 is car with tyres. In both analysis of with and without tyres using K-omega standard model, refined mesh is used with parameters as described above. Graph of coefficient of drag obtained by the results of the analysis of car with tyres is shown in Fig 18. The same design with tyres is also analyzed using Star CCM+ software in order to check the design. Coefficient of drag is increased due to the addition of tyres in design. K-Omega standard is also used in Star CCM+.

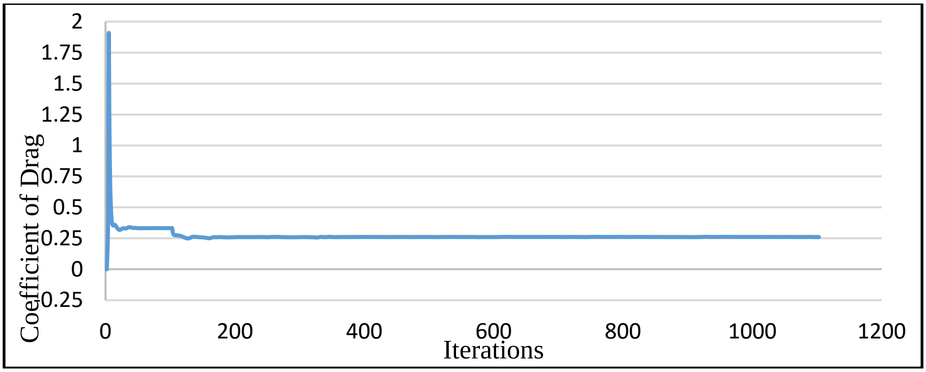

Fig. 18. Coefficient of drag of car with tyres using ANSYS Fluent at 170 kmph

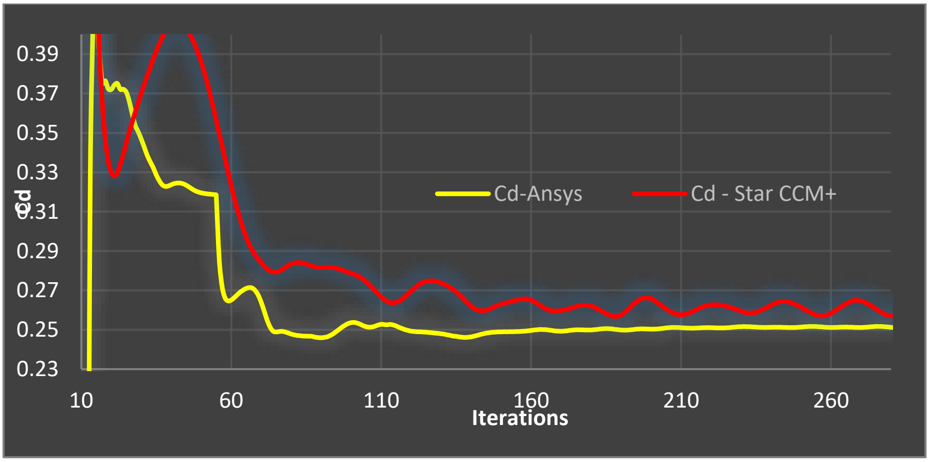

Fig. 19. Superimposed results of ANSYS FLUENT and Star CCM+ at 170 kmph

The results were compared between two software for accuracy of design performance and errors involved. Coefficient of drag of final design was determined using ANSYS. Furthermore, it was predicted using Star CCM+. Fig 19 shows the superimposed graph of both ANSYS FLUENT and Star CCM+.

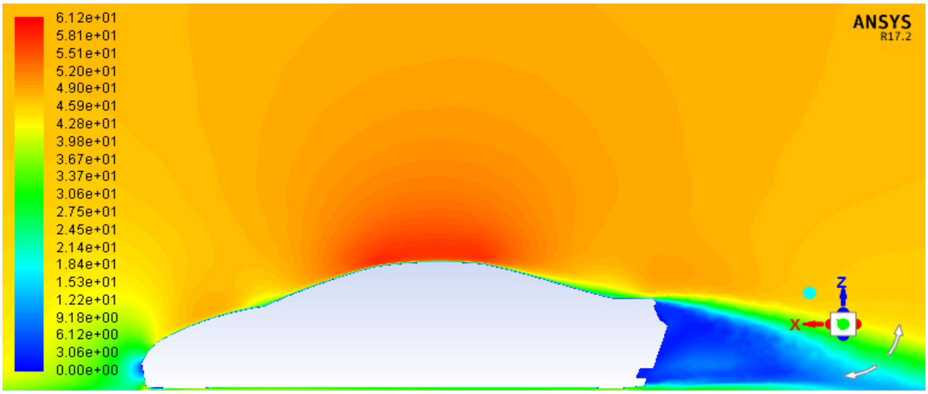

Fig. 20. Velocity contour in ANSYS FLUENT

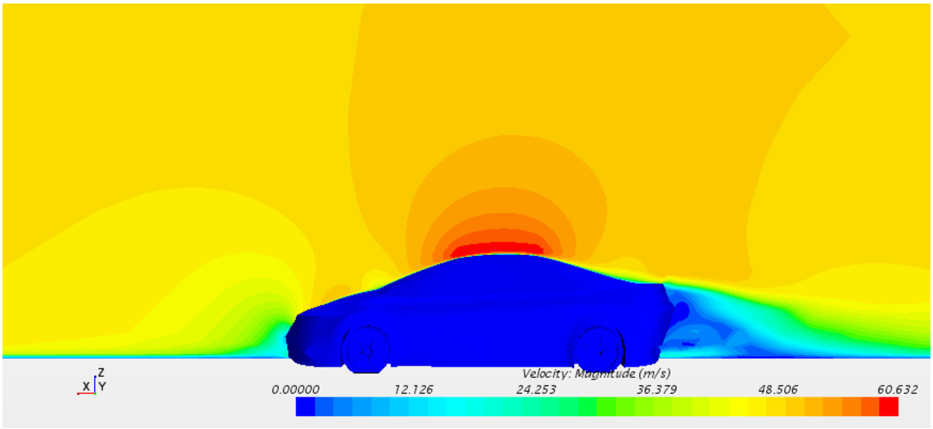

Fig. 21. Velocity contour in STAR CCM+

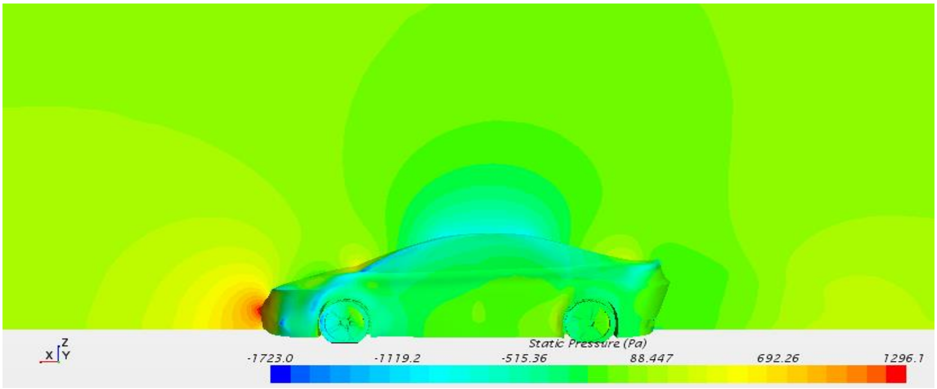

Figure 22: Static pressure contour in STAR CCM+

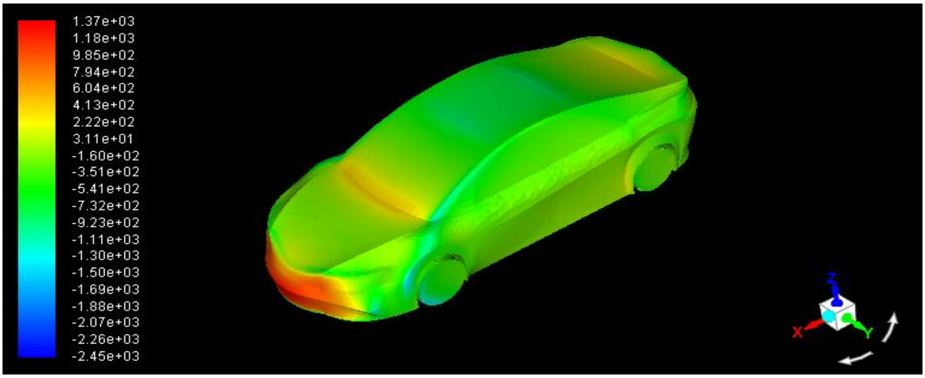

Fig. 23 Pressure contour in ANSYS

FLUENTThe results are close to each other. Contours of pressure and velocity of the last analysis obtained by both software ANSYS and Star CCM+ are shown in Figs 20-23.

IV. RESULTS

Following are the results of our work.

A. Initial Design

For Initial design coefficient of drag was 0.21 using k-epsilon realizable without tyres at 50 kmph.

B. Final Design

Values of coefficient of drag for the final design are shown below with columns describing specifications of the analysis.

TABLE 2

COEFFICIENT OF DRAG

Software	Mesh settings	Turbulence Model	Tyres	Speed Kmph	Coefficient of drag
ANSYS Fluent	Medium Mesh	Realizable K-epsilon	Without	170	0.22
ANSYS Fluent	Refined mesh	Realizable K-epsilon	without	170	0.185
ANSYS Fluent	Refined mesh	K-Omega standard	without	170	0.198
ANSYS Fluent	Refined mesh	K-Omega standard	with	170	0.251
Star CCM+	Refined mesh	K-Omega standard	with	170	0.259

V. CONCLUSIONS

Following conclusions can be drawn from the above research.

1. By reducing coefficient of drag, we can increase the efficiency of fuel.
2. Temperature has no effect on the coefficient of drag and the coefficient of lift.
3. The coefficient of drag and the coefficient of lift vary significantly with shape.
4. Making an edge shape bumper causes reduction in the coefficient of drag.
5. Mesh refinement decreases the value of the coefficient of drag and produces accurate results.
6. The coefficient of drag and the coefficient of lift are always different during the simulation of cars, with and without tyres.
7. The coefficient of drag increases during the simulation of car with tyres.
8. The value of the coefficient of drag is greater for the standard K-Omega model as compared to the K-epslion realizable model.
9. The drag coefficient is significantly similar for two different software.

REFERENCES

[1] P Ramya , A Hemanth Kumar , Jaswanth Moturi , N.Ramanaiah, "Analysis of Flow over Passenger Cars using Computational Fluid Dynamics," *Int J Eng Trend Tech (IJETT)*, vol. 29, no. 4, ISBN 2231-5381, pp. 170-176, November 2015.

[2] Yunus A. Cengel, John M. Cimbala, *Fluid Mechanics fundamental and applications*, New york: McGraw-Hill, 2006.

[3] Bruce R Munson, Donald F Young, *Fundamentals Of Fluid Mechanics, Fifth Edition*, USA: John Wiley and sons, Inc, 2006.

[4] W.H. Hucho, *Aerodynamics of road vehicles*, London: Butterw2orth, 1997.

[5] Charlie Matsubara, Tim Kuo, Helen Wu, "Comparison of the Effects of k-ϵ, k-ω, and Zero Equation Models on Characterization of Turbulent Permeability of Porous Media," *J Water Res Hydra Eng*, vol. 2, no. 2, pp. 43-50, June 2013.

[6] G. Lombardi, F. Beux, S. Carmassi, "*Aerodynamic Design of High Performance Cars: Discussion and*

*Examples on the Use of Optimization Discussion and Examples on the Use of Optimization procedur*e," SAE Technical Papers, July 2002.

[7] Pramod Nari Krishnani, "cfd study of drag reduction of a generic sport utility vehicle," MS Thesis, Mumbai India, 2009.

[8] Akshay Parab, Ammar Sakarwala, Bhushan Paste, Vaibhav Patil, Amol Mangrulkar, "Aerodynamic Analysis of a Car Model using Fluent- Ansys 14.5," *Int J Recent Tech Mech Elect Eng (IJRMEE),* vol. 1, no. 4, pp. 7-13, November 2014.

[9] N. p. Weatherill, Michael J Marchant, D. A. King, "An Introduction To Grid Generation using Multiblock Approach," in *Notes on Numerical Fluid Mechanics*, vol. 44, 1993. https://doi.org/10.1007/978-3-322-87881-6_2

[10] Rao V Garemilla, Mark S. Shephard, "Boundry Layer mesh Generation for viscous Flow simulations," *Int J num Method Eng*, vol. 49, no. 1-2, pp. 193-218, sept 2000. https://doi.org/10.1002/1097-0207(20000910/20)49:1/2<193::AID-NME929>3.0.CO;2-R

[11] Vihar Malviya, Naresh Gundala, Rakesh Mishra, "effect of cross wind on aerodynamic coefficients of ground vehicles," in *Computing and Engineering Annual Researchers' Conference*, University of Huddersfield, Huddersfield, 2009.

[12] Cakir Mustafa, "*CFD study on aerodynamic effects of a rear wing/spoiler on a passenger vehicle*," Masters Theses. Paper 1, 2012.

[13] Saha, Sambit Majumder and Somnath, "A Method of Drag Reduction of a Vehicle by Computational Investigation and Geometric Modification," *Int J Appl Eng Res*, vol. 9, no. 6, pp. 687-699, 2014.

[14] Praveen Padagannavar and Manohara Bheemanna, "automotive computational fluid dynamics simulation of a car using ansys," *Int J Mech Eng Tech (IJMET),* vol. 7, no. 2, p. 91–104, March-April 2016.