CFD ANALYSIS OF SUPER UTILIY VEHICLE TO DETERMINE AERODYNAMIC BEHAVIOUR

PERUOPGU MANOJ KUMAR[1],B. VENKATA SIVA[2]
[1]PG Scholar Department of mechanical Engineering,Narasaraopeta Engineering College,Narasaraopet,Guntur, AP, India.
[2]Assistant Professor Department of mechanical Engineering,Narasaraopeta Engineering College, Narasaraopet, Guntur, AP, India.

ABSTRACT

A steady increase in global energy demand has a direct influence on the fuel prices. This together with the environmental problems caused by the exhaust gases of cars is the main motives behind needs to reduce fuel consumption of roads vehicles. Reducing aerodynamic drag can lead to reduction in fuel consumption leading to less environment problems.

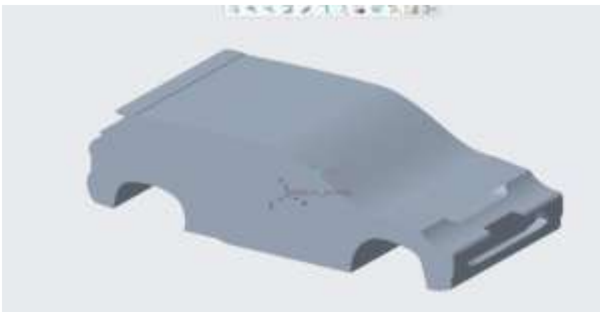
1.0 INTRODUCTION

Aerodynamic,the study of the motion of air, partically it interaction with a solid object such as an airplane wing. Aerodynamic is a sub-field of fluid dynamic and gas dynamics, is often used synonymously with gas dynamics, the difference being that "gas dynamics" applies to study of the motion of all gases, and is not limited to air. The formal study of aerodynamic began in the modern sense in the eighteen centuary, although observations of fundamental concepts such as aerodynamic drag were recorded much earlier. Most of the early efforts in aerodynamic were directed towards achieving heaviour-than-air flight, which was first demonstrated by Wilbur & Orville Wright in 1903. Since then , the use of aerodynamics through mathematical analysis, empirical approximations, wind tunnel experimentation, and computer simulation has formed a rational basis for the development of heavier-than-air flight & a number of other technologies. Recent work in aerodynamics has focused on issues related to compressible flow, turbulence, & boundary layers and has become increasingly computational in nature.

A vortex is created by the passage of an aircraft wing, revealed by smoke. Vortices are one of the many phenomena associated with the study of aerodynamics

2.0 MODELING OF EXTERNAL BODY OF SUV

The external body models of SUV's Brezza&Ecosport are modelled. The model are modified by adding lip kits to the front bumper The analyses are carried out using a commercial CFD solver, ANSYS Fluent. The solver is based on finite volume method with second order discretization. The convergence criteria for continuity, momentum and other parameters were set to 10-3 , while the convergence of energy equation was set to 10-6 . In most cases, the momentum and other residuals were less than 10-5 and the highest residual was 7*10-4 .



Layout of Steam Power Plant

3.0. CFD ANALYSIS ON SUV MODELS

CFD analysis is performed on all the models of SUV and compared for the better model by observing results of pressure, velocity, lift and drag.

Turbulence models are known to replace the time-dependent Navier–Stokes equations by averaging them and simplifying the equations to reduce the complexities in calculation of the required quantities. Though these turbulence models are simplified and averaged, these models are able to predict the effects of turbulence accurately in many applications that are developed and implemented within commercial CFD software. The 'two-equation' models are most common and widely used models. These two equations represent two transport equations to solve for turbulent properties of the flow. Generally, one of the turbulent properties is the mean turbulent kinetic energy 'k' and the second property depends on the type of turbulence model. It is either dissipation rate 'e', for k-e turbulence model or the specific dissipation, 'x', which is a measure of the inverse time scale of the eddies, for k-x turbulence model.

3.1 .BOUNDARY CONDITIONS:

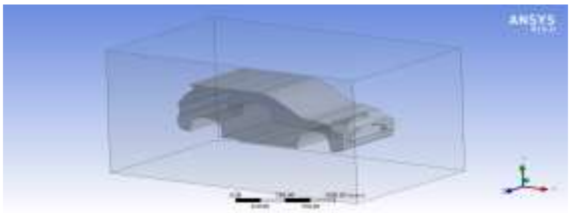
Analysis is performed by varying the speed of air speed of air -80km/hr, 120kmlhr & 160km/hr

3.2 .BREZZA ORIGINAL MODE

→→Ansys → workbench→ select analysis system → fluid flow fluent → double click

→→Select geometry → right click → import geometry → select browse →open part → ok→

Select Tools – Select Enclosure and enter dimensions



Imported geometry of Brezza original model with enclosure from Creo 6.0

3.3. RIGHT CLICK ON SETUP IN MAINWINDOW AND EDITTHE SOLVER SETUP INCLUDES

following steps:-

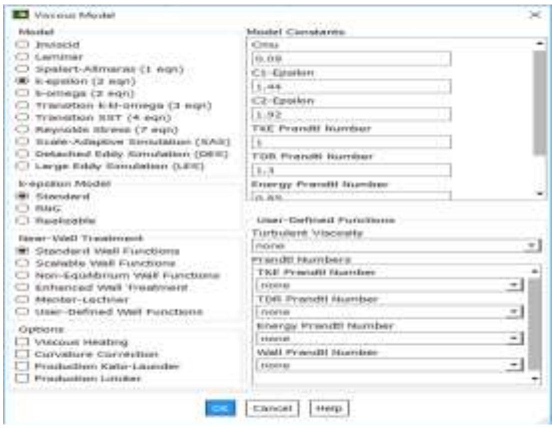
1. Type of analysis: For aerodynamic analysis 3D model analysis which pressure based steady state analysis with relative velocity formulation is carried out.
2. Models: here the type of numerical model used for the analysis is defined. For aerodynamic analysis Viscous based numerical model. The type of viscous based model used is realizable k-epsilon and scalable wall function model
3. Materials: here the material for the model is defined. In this work the analysis is carried out on Air which isthe surrounding fluid. Here properties of the material can be retrieved from ANSYS Library.4. Boundary conditions: this is the core part of ANSYS Solver here the initial boundary conditions are defined.The initial boundary values for our analysis are:-

a. Inlet fluid velocity: - 80km/hr, 120km/hr, 160km/hr.
b. Wall: - Stationary

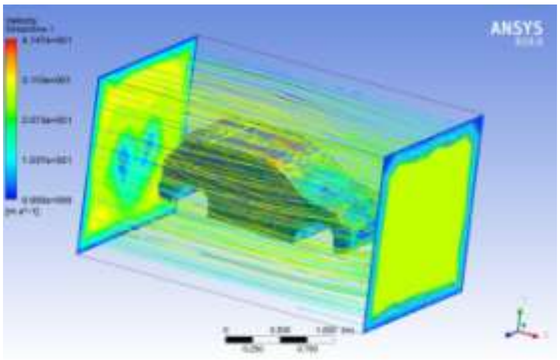
5. Solution methods :-

a. Pressure-Velocity Coupling
b. Scheme: Simple
c. Gradient: - Least Square Cell Based
d. Pressure: - second order
e. Momentum: - second order upwind

After carrying out all these initial set-ups run the calculation with required number of iterations. More the number ofiterations the obtained result will be more close to the actual values.



k-epsilon is considered as viscous model

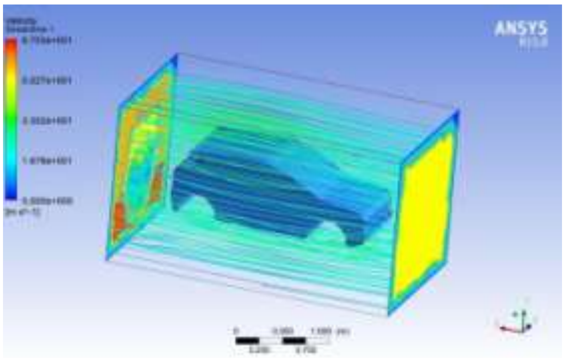


Air flow path streamlineonBrezza original model at 80km/hr

4.0. CALCULATIONS

4.1 ECOSPORT MODIFIED MODEL

Velocity – 22.22m/s (80Km/hr)



Air flow path streamlineonEcosport modified model at 80Km/hr

Drag force

Zone Viscous	Forces (n)			Coefficients		
	Pressure	Viscous	Total	Pressure	Viscous	Total
wall-kit (00 0)	(0 0 0)	(000)	(0 0 0)	(0 0 0)	(0 0 0)	(0 0 0)
wall-car (0 0 0)	(0 0 0)	(0 00)	(0 0 0)	(0 0 0)	(000)	(0 0 0)

wall-enclosure (296.47998 -95.926269 -2.0658168) (23.68417 0.096242478 0.024856249) (320.16415 -95.830026 -2.0409606) (484.04894 -156.61432 -3.3727622) (38.668032 0.15713058 0.040581631) (522.71698 -156.45719 -3.3321806)

Net (296.47998 -95.926269 -2.0658168) (23.68417 0.096242478 0.024856249) (320.16415 -95.830026 -2.0409606) (484.04894 -156.61432 -3.3727622) (38.668032 0.15713058 0.040581631)

(522.71698 -156.45719 -3.3321806)

Forces - Direction Vector (1 0 0)

Zone Viscous	Forces (n)			Coefficients		
	Pressure	Viscous	Total	Pressure	Viscous	Total
wall-kit (00 0)	(0 0 0)	(00 0)	(0 0 0)	(0 0 0)	(0 0 0)	(0 0 0)
wall-car (0 0 0)	(0 0 0)	(0 00)	(0 0 0)	(0 0 0)	(00 0)	(0 0 0)

wall-enclosure (296.47998) (23.68417) (320.16415)
(484.04894) (38.668032) (522.71698)

Net (296.47998) (23.68417) (320.16415)
(484.04894) (38.668032) (522.71698)

5.0.RESULTS AND DISCUSSIONS

From CFD analysis, the pressure, velocity, drag and lift forces are observed for all the models of SUV.
The results are tabulated as follows:

Speed (Km/hr)	Pressure (Pa)	Velocity (m/s)	Drag (N)	Lift (N)
80	229.2	36.79	395	89.82
120	514.4	40.34	883	200.05
160	912.5	53.84	1557.68	364.28

CFD Results of Brezza Original model at different speeds

Speed (Km/hr)	Pressure (Pa)	Velocity (m/s)	Drag (N)	Lift (N)
80	223.5	26.96	395.19	91.611
120	502.7	40.48	879.54	208.189
160	892.9	53.99	1557.12	372.22

CFD Results of Brezza Modified model at different speeds

By comparing results of original and modified Brezza models, the pressure & velocity are decreasing and lift is increasing for the modified model (attaching lip kit for the front bumper) when compared with original model. The change in drag is very less for the modified model. The change in velocity is also very less for higher speeds 120km/hr and 160km/hr.
For the modified model, the pressure is reducing by about 2.5%, the velocity is reducing by about 26.7%, and the lift is increasing by about 2%.

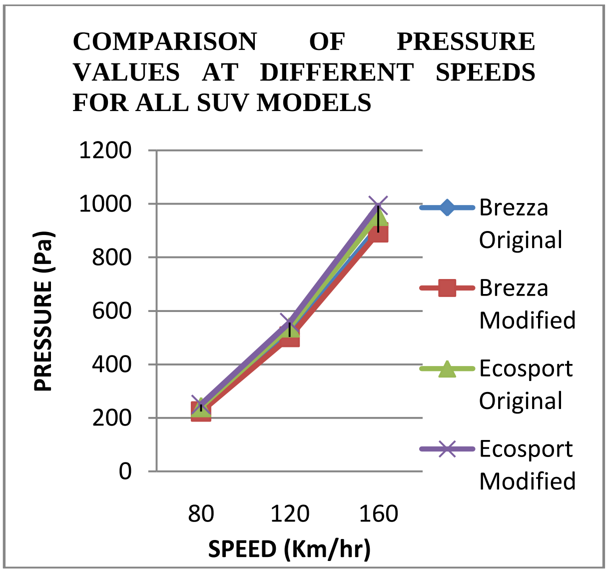
Speed (Km/hr)	Pressure (Pa)	Velocity (m/s)	Drag (N)	Lift (N)
80	241.1	28.42	492.4	77.29
120	539.1	42.65	1097.43	84.99
160	956	56.01	2226.90	176.16

CFD Results of Ecosport Original model at different speeds

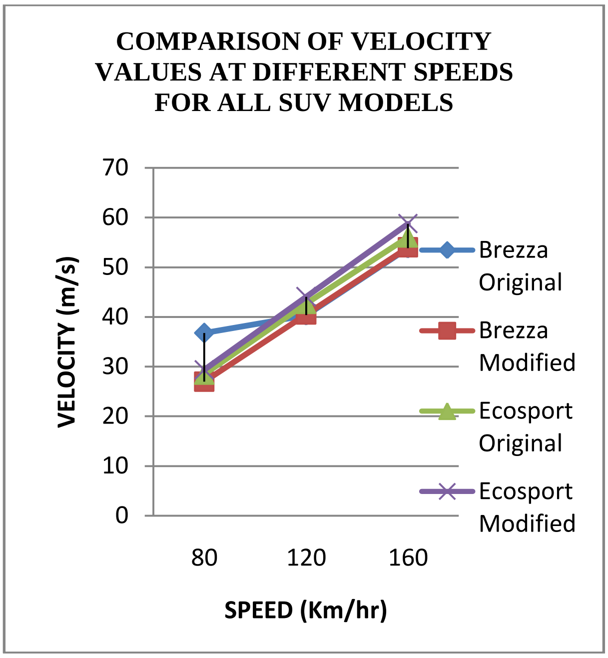
Speed (Km/hr)	Pressure (Pa)	Velocity (m/s)	Drag (N)	Lift (N)
80	249.9	29.37	522.71	156.45
120	556.2	44.07	1163.08	344.27
160	993.5	58.76	2059.85	601.94

CFD Results of Ecosport Modified model at different speeds

By comparing results of original and modified Ecosport models, the pressure, velocity, drag and lift are increasing for the modified model (attaching lip kit for the front bumper) when compared with original model.
For the modified model, the pressure is increasing by about 3.5%, the velocity is increasing by about 3%, the lift is increasing by about 50.6% and the drag is increasing by about 5.8%.



Comparison of pressure values at different speeds for all SUV models



Comparison of velocity values at different speeds for all SUV models

6.0 CONCLUSION

CFD analysis is done by applying different inlet speeds 80Km/hr, 120Km/hr and 160Km/hr. the original models of Brezza and Ecosport are modified by attaching a lip kit for front bumper.

The modification of the models by attaching lip kit showed significant difference on Ecosport model than Brezza model. The lift has been increased and drag force has been reduced by the attachment for Ecosport model. But the pressure and velocity are increasing for modified Ecosport model in reverse with the Brezza model where pressure and velocity are decreased for modified model.

7.0 REFERENCES

1. Pramod Krishnani, CFD Analysis of Drag Reduction for a Generic SUV, Proceedings of IMECE2009 2009 ASME International Mechanical Engineering Congress and Exposition November 13-19, 2009, Lake Buena Vista, Florida, USA IMECE2009-10170

2. Vipul Kshirsagar, Jayashri V. Chopade, Aerodynamics of High Performance Vehicles, International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056, Volume: 05 Issue: 03 | Mar-2018 www.irjet.net p-ISSN: 2395-0072

3. Pikula, Boran&Filipovic, Ivan &Kepnik, Goran. (2011). Research of the external aerodynamics of the vehicle model, ResearchGate.

4. Angel Huminic&Gabriela Huminic, Aerodynamic study of a generic car model with wheels and underbody diffuser, International Journal of Automotive Technology volume 18, pages397–404(2017)

5. ShobhitSenger and S.D. Rahul Bhardwaj, Aerodynamic Design of F1 and Normal Cars and Their Effect on Performance, International Review of Applied Engineering Research. ISSN 2248-9967 Volume 4, Number 4 (2014), pp. 363-370

6. Ms.SujataShenkar, Mr. Sanjay D. Nikhade, Mr.Sagarkumar Banerjee, Mr.AmarjeetKunal, Mr. Santosh Mote, Mr. MustufaSadikot, Computational Aerodynamics Research and Vehicle Engineering Development (CAR-VED), International Journal of Engineering Science Invention, ISSN (Online): 2319 – 6734, ISSN (Print): 2319 – 6726, Volume 5 Issue 11|| November 2016 || PP. 66-67

7. Gavin Dias, Nisha R. Tiwari, Joju John Varghese, Graham Koyeerath, Aerodynamic Analysis of a Car for Reducing Drag Force, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 13, Issue 3, Ver. I (May- Jun. 2016), PP 114-118

8. A.Anish, Suthen.P.G, Viju.M.K, Modelling and analysis of a car for reducing aerodynamic forces, International Journal of Engineering Trends and Technology, Volume 47 Number 1 May 2017

9. HalilSadettinHamut , Rami Salah El-Emam , Murat Aydin , Ibrahim Dincer, Effects of rear spoilers on ground vehicle aerodynamic drag, International Journal of Numerical Methods for Heat & Fluid Flow, ISSN: 0961-5539, 1 April 2014.

Rajat Rohit, Chandrakant R Kini and Srinivas G, Recent trends in aerodynamic performance developments of automobile vehicles: A Review, Journal of Mechanical Engineering Research and Developments (JMERD) 42(4) (2019) 206-214