Design and Analysis of Lift and DragForce for a Passenger Car

R.Sridhar[1],Sathishkumar G[2], S.Sivabalan[3], S. Sivaganesan[4]

[4]Associate professor, Department of Mechanical Engineering

[1,2,3] Assistant Professor, Department of Mechanical Engineering,

Vels Institute of Science Technology & Advanced Studies-Chennai

*Email:gsk032sathish@gmail.com, sivamech1990@gmail.com,

srisampangy@gmail.com

Abstract

Numerical prediction of incompressible turbulent flow has been performed on a passenger car body moving with a velocity of 11.11 m/s (40 km/hr). CATIA, 3D modeling software was used to model 3D surface modeling of the car. FLUENT, the computational fluid dynamics code, which in-corporate k-ε turbulence model and segregated implicit solver was used to perform computation. The aerodynamic analysis was performed to study the flow behavior of the air over the car body. The analysis includes the contours of pressure and velocity that impacts the car body followed by an evaluation of the coefficient of lift and drag.

In the present work, model of generic passenger car has been developed in CATIA and generated the wind tunnel and applied the boundary conditions in FLUENT platform then after testing and simulation has been performed for the evaluation of drag coefficient for passenger car. In another case, the aerodynamics of the most suitable design of tail plate is introduced and analyzed for the evaluation of drag coefficient for passenger car. The addition of tail plates results in a reduction of the drag-coefficient and lift coefficient in head-on wind. Rounding the edges partially reduces drag in head-on wind but does not bring about the significant improvements in the aerodynamic efficiency of the passenger car with tail plates, it can be obtained. Hence, the drag force can be reduced by using add on devices on vehicle and fuel economy, stability of a passenger car can be improved.

Keywords: *3D surface modeling, Aerodynamic analysis, Aerodynamic efficiency*

1. Introduction

The main purpose of the design of a car body is for containing and protection of the engine and accessories as well as the passenger. Layer of heavy gumming material is sprayed or brushed to the interior of the body panels. Usually the car bodies become less subjected to temperature changes due to these material acts as heat insulators. Initially an automobile body furnishing seats for the passengers was considered sufficient. But then closed car bodies became popular. With the passage of time, riding comfort with reference to seating, heating and ventilation became the target of attention. Further due to increased cars, operation speeds of motor vehicles increased, which in turn necessitated special attention to streamlining is the process of shaping the body to reduce air resistance as the engine move forward. In this case, curves instead of angles and flat surfaces are used on the body shaping.

In the earlier models, the vertical front sections of the radiator and the windscreen offered considerable resistance to the car movement. Moreover, in the back of the car, air eddies formed tend to produce a drag. To overcome this air resistance, considerable power was required at intermediate and high speeds. Wind test and actual road tests had conclusively proved that old styled closed cars had great resistance of wind of the car movement. It had been proved that if heavy end of the body (ie. rear) were placed to the front, the car would be operated more economically, by streamlining the body of the car, better speed characteristics and fuel efficiency were expected. The sloping lines provided on the streamlined car further permits it body to move more smoothly through the air due to pushing of the air up and around the car. In comparisons to earlier models, air eddies are not formed behind the body of the modern car. In case of racing cars, where the speeds are of paramount importance, streamlining has great influence. For cars operating 70 to 80 kmph the advantages due to streaming are small. The fuel saving is also worthwhile only when the vehicle is operated continuously at high speed. With computational fluid dynamics codes it it is possible to study the behavior of airflow over the bodies and this will help to take critical decisions in the design processes of car body and other objects.

1.1. History of Car

The first working steam-powered vehicle was designed and most likely built by Ferdinand Verbiest, a Flemish member of a Jesuit mission in China around 1672. It was a 65-cm-long scale-model toy for the Chinese Emperor that was unable to carry a driver or a passenger. It is not known if Verbiest's model was ever built.

Nicolas-Joseph Cugnot is widely credited with building the first full-scale, self-propelled mechanical vehicle or automobile in about 1769. He created a steam-powered tricycle. He also constructed two steam tractors for the French Army, one of which is preserved in the French National Conservatory of Arts and Crafts. His inventions were, however, handicapped by problems with water supply and maintaining steam pressure. In 1801, Richard Trevithick built and demonstrated his Puffing Devil road locomotive, believed by many to be the first demonstration of a steam-powered road vehicle. It was unable to maintain sufficient steam pressure for long periods, and was of little practical use. The development of external combustion engines is detailed as part of the history of the car, but often treated separately from the development of true cars. A variety of steam-powered road vehicles were used during the first part of the 19th century, including steam cars, steam buses, phaetons, and steam rollers. Sentiment against them led to the Locomotive Acts of 1865.

In 1807 Nicéphore Niépce and his brother Claude created what was probably the world's first internal combustion engine (which they called a Pyréolophore), but they chose to install it in a boat on the river Saone in France. Coincidentally, in 1807 the Swiss inventor François Isaac de Rivaz designed his own 'de Rivaz internal combustion engine' and used it to develop the world's first vehicle to be powered by such an engine. The Niépces' Pyréolophore was fuelled by a mixture ofLycopodium powder (dried spores of the Lycopodium plant), finely crushed coal dust and resin that were mixed with oil, whereas de Rivaz used a mixture of hydrogenand oxygen. Neither design was very successful, as was the case with others, such as Samuel Brown, Samuel Morey, and Etienne Lenoir with his hippomobile, who each produced vehicles (usually adapted carriages or carts) powered by internal combustion engines.

1.2. Development Of Cars

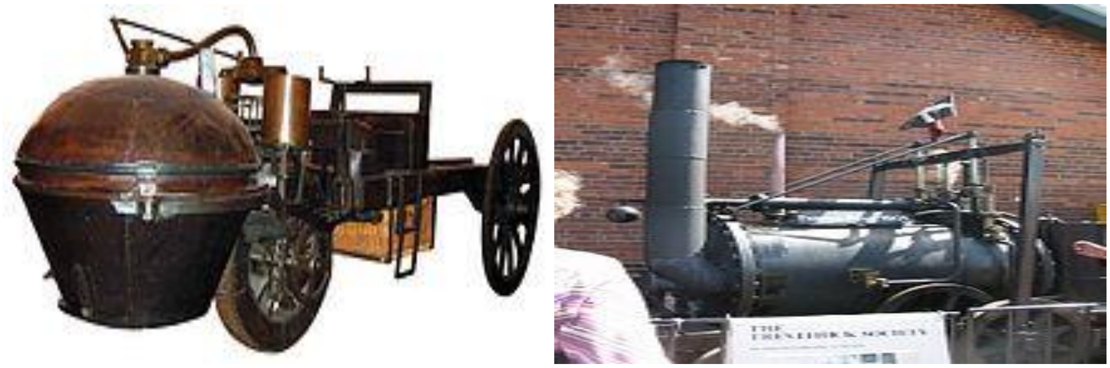


Figure 1.17th- 18th-19th century Steam-powered vehicles, Road locomotive Puffing Devil

Figure 2.1888, first electrical car

Figure 3.1885-built first car

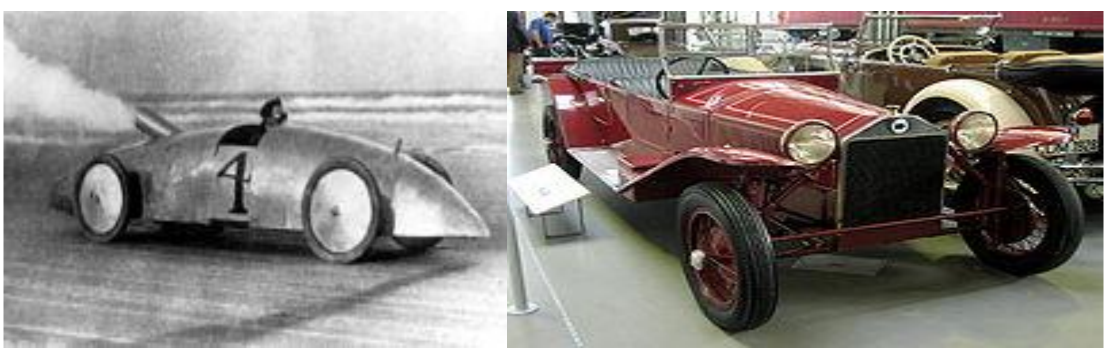


Figure 4. 1923 Lancia Lambda


Figure 5. 1926 Bugatti Type 35 **Figure 6. Ford V-8 (Model B)**


Figure 7. Volkswagen Beetle Figure 8. Modern cars (A 2011 Toyota Corolla

2. Materials and Methods

2.1. Numerical Method

Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving mathematical equations that represent physical laws, using a numerical process.

Conservation of mass, momentum, energy, species, ...

The result of CFD analyses is relevant engineering data:

- conceptual studies of new designs
- detailed product development
- troubleshooting
- redesign
- CFD analysis complements testing and experimentation.
- Reduces the total effort required in the laboratory.
- solvers are based on the finite volume method.
- Domain is discretized into a finite set of control volumes or cells.
- General conservation (transport) equation for mass, momentum, energy, etc.

$$\frac{\partial}{\partial t}\int \rho \emptyset dV + \oint \rho \emptyset V.dA = \oint \Gamma \nabla \emptyset . dA + \int S_{\emptyset} dV$$

- ✓ I Term- Unsteady
- ✓ II Term-Convection
- ✓ III Term-Diffusion
- ✓ IV Term- Generation are discretized into algebraic equations.

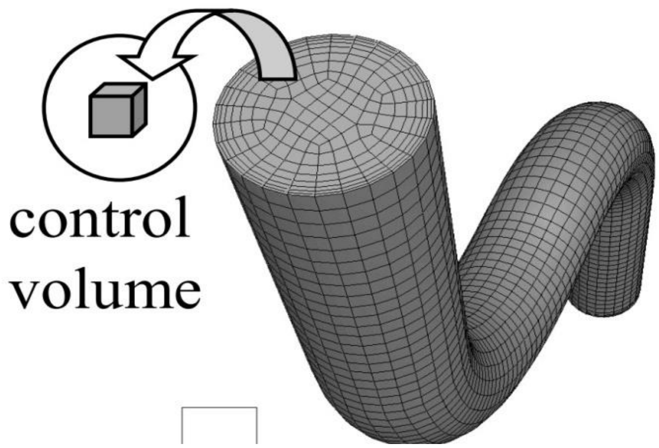


Figure 9. Control Volume

The computation is performed using the finite-volume technique with upwind discretization to solve the two-dimensional compressible RANS equations. The air is considered to be a calorically perfect gas with constant ratio of specific heat. The space discretization is performed by a cell-centered formulation. To account for the directed propagation of information in the inviscid part of the equations, the advection upstream splitting method flux vector splitting is applied for the approximation of the convective flux functions. Higher-order accuracy for the upwind discretization and consistency with the central differences used for the diffusive term is achieved by the monotonic upstream scheme for conservation laws extrapolations. Time integration is performed by an explicit five stage Runge-Kutta time-stepping scheme. To enhance convergence, a multi-grid method, implicit residuals smoothing, and local time stepping are applied.

2.2. Numerical Accuracy Analysis

The solution can be considered as converged after approximately 500 iterations, where the Courant number is 0.5. At this stage, the continuity, x-velocity, y-velocity and energy residuals, reach their minimum values after falling for over four orders of magnitude. The turbulence (k and Ɛ) residual have a five orders of magnitude decrease. An additional convergence criterion enforced in this current analysis requires the difference between computed inflow and outflow mass flux to drop below 0.5 per cent. The evaluation was performed using the UN-structured mesh.he performance of a grid sensitivity analysis confirmed that the grid resolution used here is sufficient.

Table 1. Boundary Conditions

velocity inlet	11.11 m/s
Pressure outlet	0 pascal

Total temperature	218 K
Operating conditions	101325 pascal

2.3. Aerodynamic Calculation

For lift

$$C_L = \frac{L}{1/2\rho V^2 S}$$

c_l − coefficient of lift

l − lift per unit span

ρ − density at std. atm (1.225kg/ı)

v − velocity (40kmph)

s = span area

$$C_L = \frac{48.181}{\frac{1}{2} * 1.225 * 11.11^2 * 6.718}$$

$$C_L = 0.094$$

For drag

$$C_d = \frac{D}{1/2\rho V^2 S}$$

c_d − coefficient of drag

D − lift per unit span

ρ − density at std. atm (1.225kg/ı)

v − velocity (40kmph)

s = span area

$$C_d = \frac{109.869}{\frac{1}{2} * 1.225 * 11.11^2 * 2.517}$$

$$C_d = 0.430$$

2.4. Reading Mesh Components

Components are defined in preprocessor

Cell = control volume into which domain is broken up

Computational domain is defined by mesh that represents the fluid and solid regions of interest.

Face = boundary of a cell Edge = boundary of a face Node = grid point

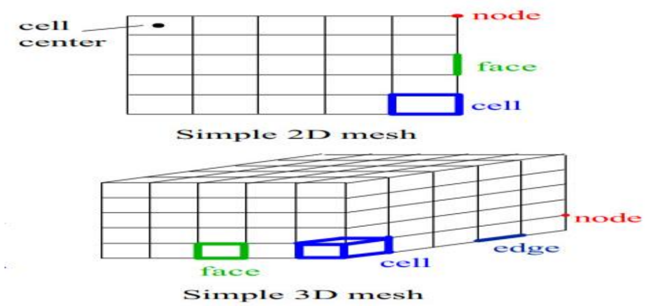
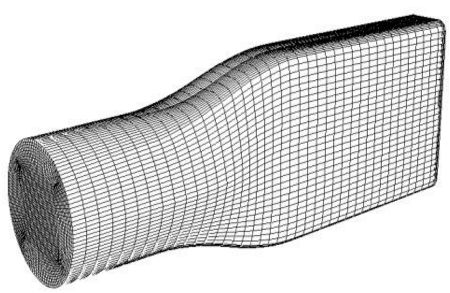


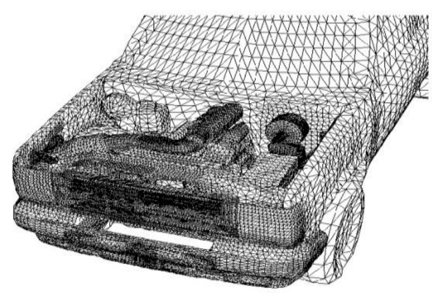
Figure 10. Meshes description

TRI / TET VS. QUAD/HEX MESHES

For simple geometries, quad/hex meshes can provide high-quality solutions with fewer cells than a comparable tri/tet mesh.

Align the gridlines with the flow.





Fi

gure 11. Structured Mesh **Figure 12. Unstructured Mesh**

For complex geometries, quad/hex meshes show no numerical

The computations have been performed using the un-structured grid. Our car model and domain is a 3-D. The dimension has been taken from the thesis, where the dimensions are in m. The meshing and cad data was done by the tool CATIA and HYPERMESH.

3. Results An Discussion

Though a considerable amount of data was gathered mainly in terms of line plots of static pressure, contours of velocity and turbulence contour. In the line plots, static pressure has been normalized with the free stream value. These plots are presented for the surfaces of car model, i.e. tyres, upper surface and domain. The aerodynamic phenomena such as lift and drag and flow separation are discussed.

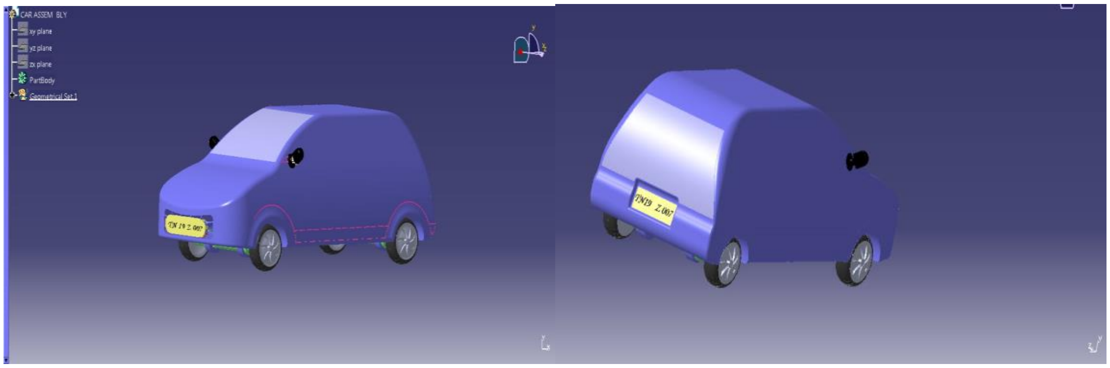
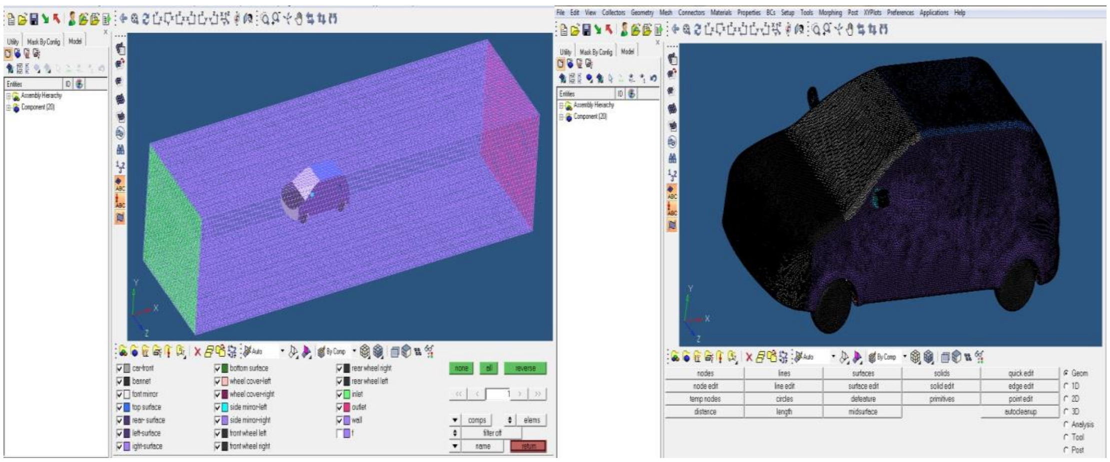
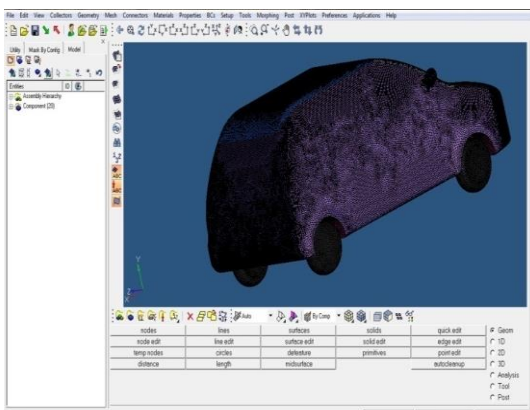


Figure 13.Sketch of car





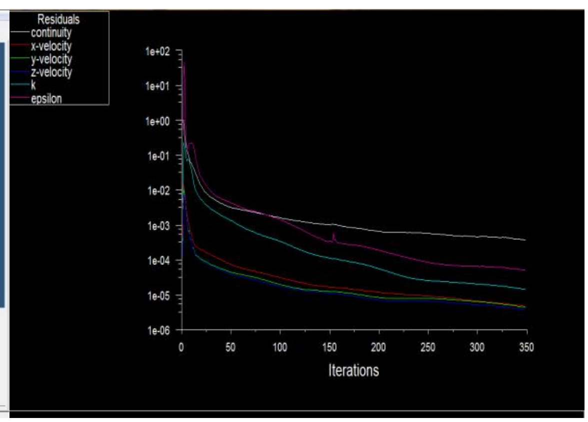


Figure14.un-Structured Grid for the Design computation

Figure 15. Residuals for

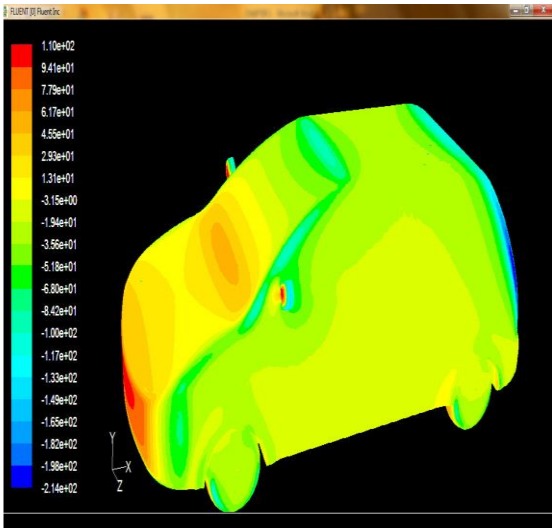


Figure 16.Static Pressure Contour

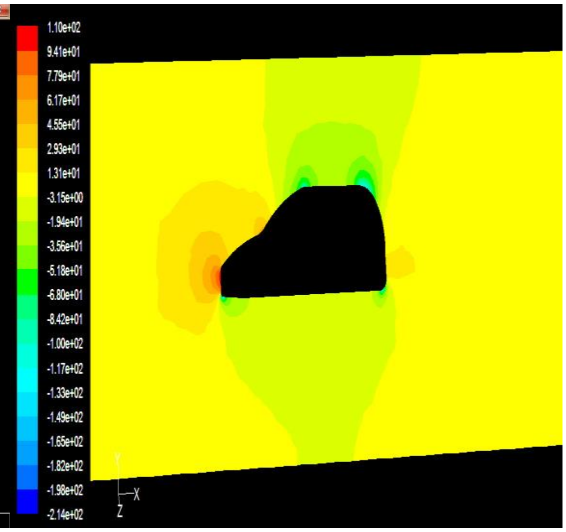


Figure 17.Pressure Contour in plane view

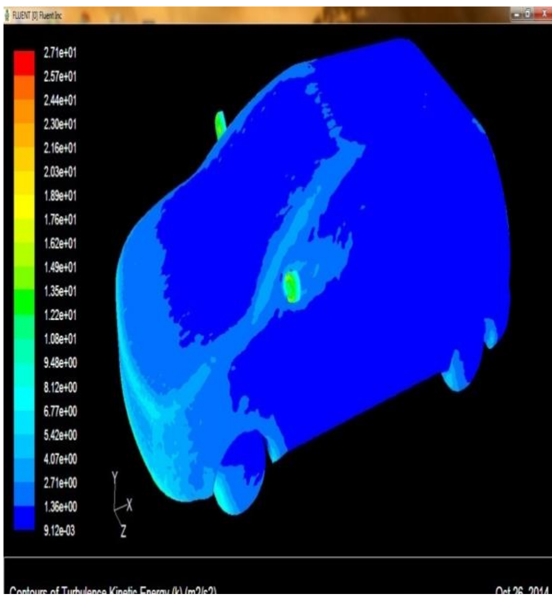


Figure 18.Turbulence Contour

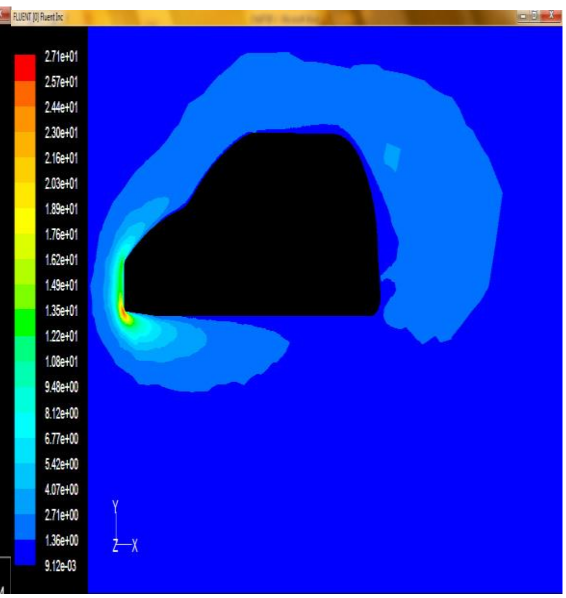


Figure 19.Turbulence Contour in plain view

4. Conclusion

The main objective to estimate the drag coefficient and flow visualization is achieved. Aerodynamics drag for my car is 0.430 at ranging velocity between 40km/h. The analysis shows aerodynamics drag in term of drag forces or drag coefficient proportionally increased to the square of velocity. The contour plot of velocity and pressure were shown the in aerodynamics drag analysis as a visualization analysis. The pattern of visualization for every velocity depict quite same either for velocity contour plot or pressure contour plot.Here my project doesn't end and the optimization of my vehicle is to be continued in the project (phase-2). The lift in the current passenger car is 0.094 and the drag is 0.430. there are different techniques to reduce drag in the automobile car the one way is changing the car design according to aerodynamic considerations and the another way by providing tail plates, spoiler at the rear end to avoid eddies and wake behind the vehicle.

In future, the aerodynamics of the most suitable design of tail plate is introduced and analysed for the evaluation of drag coefficient for passenger car. And the comparative results will be given in the final phase of the project.

References

[1] Wolf-Heinrich Hucho. Aerodynamic of Road Vehicle. Fourth Edition. Society of Automotive Engineers, Inc. 1998.

[2] Heinz Heisler. Advanced Vehicle Technology. Second Edition. Elsevier Butterworth Heinemann. 2002.

[3] Rosli Abu Bakar, Devarajan Ramasamy, Fazli Ismail, Design and Development of Hybrid Electric Vehicle Rear Diffuser, Science, Technology & Social Sciences 2008 (STSS), Malaysia.

[4] Guido Buresti. The Influence of Aerodynamics on the Design of High- Performance Road Vehicles. Department of Aerospace Engineering University of Pisa, Italy. 19 March 2004.

[5] Luca Iaccarino. Cranfield University Formula 1 Team: An Aerodynamics Study of the Cockpit. School of Engineering. Cranfield University. August 2003.

[6] Wong H.M, D. Ramasamy, Series Hybrid Electric Vehicle Cost-Effective Powertrain Components Development, RDU 070305, UMP Research Grant2007.

[7] Mark Coombs and Spencer Drayton, Proton Service and Repair Manual, Haynes Ptd. Ltd, P Ref 1, 2003, USA

[8] Amir Shidique, Simulation and Analysis of Hybrid Electric Vehicle (HEV) by Addition of a Front Spoiler, p39, Thesis, Universiti Malaysia Pahang, 2007.

[9] Bruce R. Munsan. Donald F. Young and Theodore H. Okiishi. Fundamental of Fluid Mechanics. Fifth Edition. John Wiley & Sons (Asia), Inc. 2006.

[10] Dr. V. Sumantran and Dr. Gino Sovran. Vehicle Aerodynamics. Society of Automotive Engineers, Inc. 1996.