DESIGN AND ANALYSIS OF LIFT AND DRAG FORCE FOR A PASSENGER CAR

G.Sathishkumar[1] & T.Thirunavukkarasu[2]

[1]PG Scholar Department of Mechanical Engineering, Gojan School of Business and Technology, Chennai
[2]UG student, Department of Mechanical Engineering, Gojan School of Business and Technology, Chennai
Corresponding author e-mail: thiru_74@rediffmail.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
.

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.

Numerical Method:

CFD 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.

HOW DOES CFD WORK?

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.

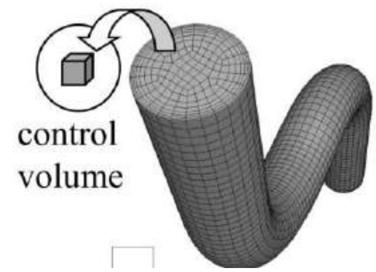
$$\frac{\partial}{\partial t}\int \rho\emptyset dV + \oint \rho\emptyset V.dA = \oint \Gamma\nabla\emptyset.dA + \int S_{\emptyset}dV$$

I Term- Unsteady

II Term-Convection

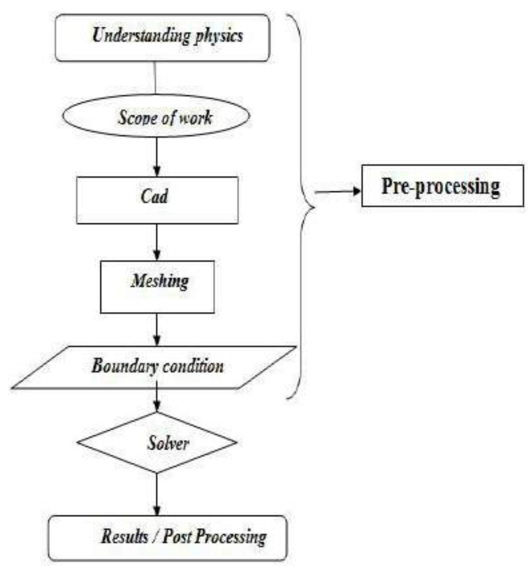
III Term-Diffusion

IV Term- Generation are discretized into algebraic equations.



Control Volume

FLOWCHART



CAR MODEL

The main geometric parameters of the car model are referred to Figure. 13 and the length unit is m. CATIA, 3D modeling software was used to model 3D surface modeling of the car. 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.

AERODYNAMIC FORMULAE

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 ($1.225kg/m^3$)

v – velocity (40kmph)

s = span area

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 ($1.225kg/m^3$)

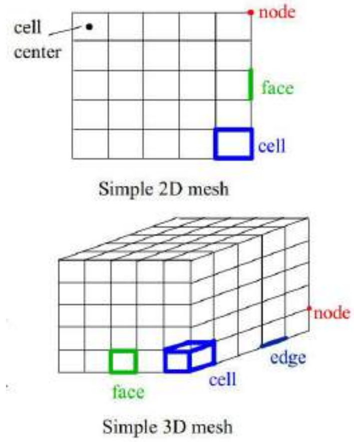
v – velocity (40kmph)

s = span area

MESHING

READING MESH COMPONENTS

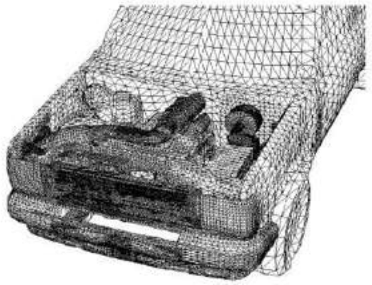
Components are defined in preprocessor
Cell = control volume into which domain is broken up
computational domain is defined by mesh that represents the fluid and solid regions of interest.
Face = boundary of a cell Edge = boundary of a face Node = grid point
Zone = grouping of nodes, faces, and/or Cells.



Meshes description

TRI / TET VS. UAD/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.
For complex geometries, quad/hex meshes show no numerical advantage, and you can save meshing effort by using a tri/tet mesh.



Unstructured Mesh

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 te tool CATIA AND HYPERMESH.

Mesh type	Un-Structured
Grid	TRIA
Cells	1649723
Faces	3423260
Nodes	339215
Partitions	1
Growth rate	1.2

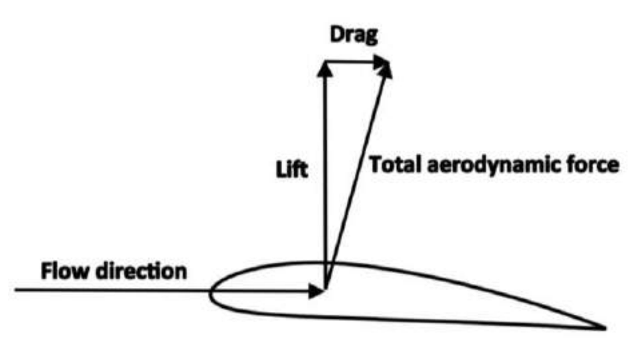
NUMERICAL ACCURACY ANALYSIS

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.The performance of a grid sensitivity analysis confirmed that the grid resolution used here is sufficient.

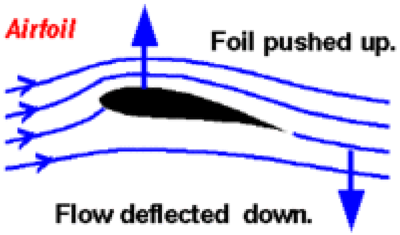
Velocity inlet	11.11 m/s
Pressure outlet	0 pascal
Total temperature	218 K
Operating conditions	101325 pascal

LIFT

A fluid flowing past the surface of a body exerts a force on it. Lift is the component of this force that is perpendicular to the oncoming flow direction. It contrasts with the drag force, which is the component of the surface force parallel to the flow direction. If the fluid is air, the force is called an aerodynamic force. In water, it is called a hydrodynamic force.



FLOW DEFLECTION AND NEWTON'S LAWS



Newton's third law says that for every action there is an equal and opposite re-action. When an airfoil deflects air downwards, the air exerts an upward force on the airfoil.An airfoil generates lift by exerting a downward force on the air as it flows past. According to Newton's third law, the air must exert an equal and opposite (upward) force on the airfoil, which is the lift. In the case of an airplane wing, the wing exerts a downward force on the air and the air exerts an upward force on the wing. The air changes direction as it passes the airfoil and follows a path that is curved. Whenever airflow changes direction, a reaction force is generated opposite to the directional change. Newton's second law, F=ma, tells us that the lift force exerted on the air is equal to its mass times its downward acceleration. This is often more conveniently expressed as the rate of momentum change over time. The downward turning of the flow is not produced solely by the lower surface of the airfoil, and the flow following the upper surface contributes strongly to the downward-turning action. In some versions of this explanation, the tendency of the flow to follow the upper surface is referred to as the Coandă effect. This is a controversial usage of the term.

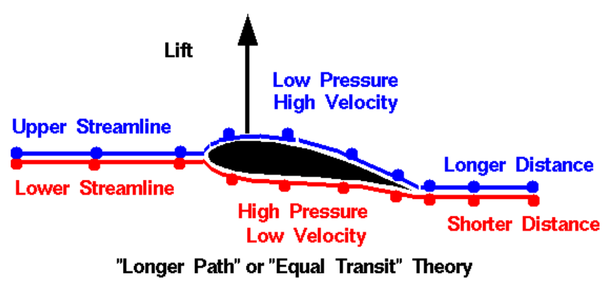
LIFT COEFFICIENT

If the lift coefficient for a wing at a specified angle of attack is known (or estimated using a method such as thin airfoil theory), then the lift produced for specific flow conditions can be determined using the following equation:[58]

$$L = \tfrac{1}{2}\rho v^2 A C_L$$

where

- L is lift force,
- ρ is air density,
- v is true airspeed,
- A is planform area, and
- C_L is the lift coefficient at the desired angle of attack, Mach number, and Reynolds number



DRAG

In fluid dynamics, drag (sometimes called air resistance, a type of friction, or fluid resistance, another type of friction or fluid friction) refers to forces acting opposite to the relative motion of any object moving with respect to a surrounding fluid. This can exist between two fluid layers (or surfaces) or a fluid and a solid surface. Unlike other resistive forces, such as dry friction, which are nearly independent of velocity, drag forces depend on velocity. Drag force is proportional to the velocity for a laminar flow and for a squared velocity for a turbulent flow. Even though the ultimate cause of a drag is viscous friction, the turbulent drag is independent of viscosity. Drag forces always decrease fluid velocity relative to the solid object in the fluid's path.

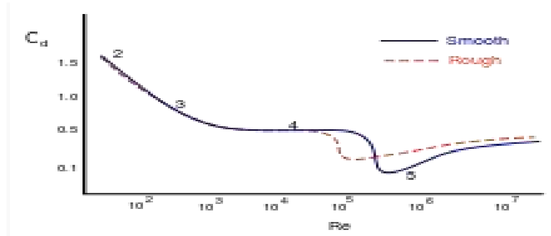
Types of DRAG

Types of drag are generally divided into the following categories:

- parasitic drag, consisting of
- form drag, skin friction,
- interference drag,
- lift-induced drag, and
- Wave drags (aerodynamics) or wave resistance (ship hydrodynamics).

The phrase parasitic drag is mainly used in aerodynamics, since for lifting wings drag is in general small compared to lift. For flow around bluff bodies, drag is most often dominating, and then the qualifier "parasitic" is meaningless. Form drag, skin friction and interference drag on bluff bodies are not coined as being elements of "parasitic drag", but directly as elements of drag.

Further, lift-induced drag is only relevant when wings or a lifting body are present, and is therefore usually discussed either in the aviation perspective of drag, or in the design of either semi-planing or planing hulls. Wave drag occurs when a solid object is moving through a fluid at or near the speed of sound in that fluid—or in case there is a freely-moving fluid surface with surface waves radiating from the object, e.g. from a ship.



Drag coefficient Cd for a sphere as a function of Reynolds number Re, as obtained from laboratory experiments. The solid line is for a sphere with a smooth surface, while the dashed line is for the case of a rough surface.

Drag depends on the properties of the fluid and on the size, shape, and speed of the object. One way to express this is by means of the drag equation:

$$F_D = \frac{1}{2} \rho v^2 C_D A$$

where

FD is the drag force,
ρ is the density of the fluid,[10]
v is the speed of the object relative to the fluid,
A is the cross sectional area, and
CD is the drag coefficient – a dimensionless number.

The drag coefficient depends on the shape of the object and on the Reynolds number:

$$R_e = \frac{vD}{\nu}$$

Where D is some characteristic diameter or linear dimension and ν is the kinematic viscosity of the fluid (equal to the viscosity μ divided by the density). At low Reynolds number, the drag coefficient is asymptotically proportional to the inverse of the Reynolds number, which means that the drag is proportional to the speed. At high Reynolds number, the drag coefficient is more or less constant. The graph to the right shows how the drag coefficient varies with Reynolds number for the case of a sphere.

For high velocities (or more precisely, at high Reynolds number) drag will vary as the square of velocity. Thus, the resultant power needed to overcome this drag will vary as the cube of velocity. The standard equation for drag is one half the coefficient of drag multiplied by the fluid mass density, the cross sectional area of the specified item, and the square of the velocity.

Wind resistance is a layman's term for drag. Its use is often vague, and is usually used in a relative sense (e.g.
a badminton shuttlecock has more wind resistance than a squash ball).

DESIGN AND CALCULATION:

AERODYNAMIC CALCULATION

1. For Lift with Tail plate

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

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

$$\boxed{C_L = 0.094}$$

For Drag with Tail plate

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

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

$$\boxed{C_d = 0.430}$$

2. For lift without Tail plate:

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

$$C_L = \frac{35.22}{\frac{1}{2} * 1.225 * 11.11^2 * 5.7}$$

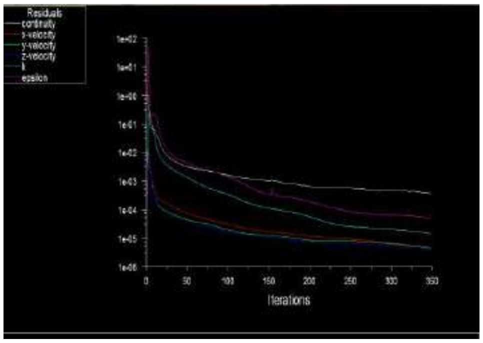
$$\boxed{C_L = 0.074}$$

For drag

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

$$C_d = \frac{120.45}{\frac{1}{2} * 1.225 * 11.11^2 * 1.11}$$

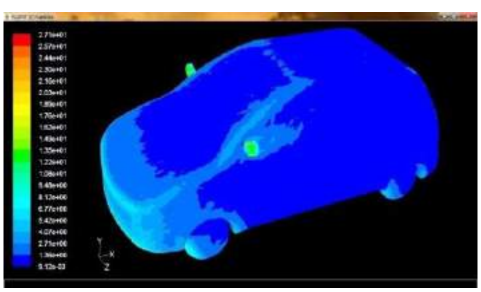
$$\boxed{C_d = 0.352}$$



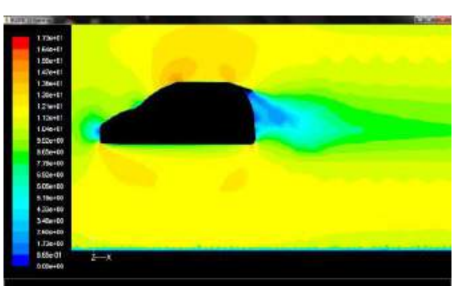
Residuals for computation

The residual approximately converged at 500 iterations and solving of equations such as

- Continuity
- x- velocity
- y-velocity
- z-velocity
- k turbulence model
- ε turbulence



Turbulence Contour without tail plates



Velocity Contour with tail plates

CONCLUSION

The main objective to estimate the drag coefficient and flow visualization is achieved. Aerodynamics drag for my car is 0.430 and with tail plate is 0.352 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.The project of CFD simulation over a passenger car for aerodynamic drag reduction is done and the drag is reduced by 0.078%.. Hence it is achieved by a popular software packages like CATIA V5r18,

<table>
<tr><th>CONTENTS</th><th>COEFFICIENT OF</th><th>COEFFICIENT OF</th><th>% DRAG REDUCTION</th></tr>
<tr><td>WITH TAIL PALTE</td><td>0.094</td><td></td><td rowspan="2">0.078</td></tr>
<tr><td>WITHOUT TAIL PLATE</td><td>0.074</td><td>0.352</td></tr>
</table>

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.*