Flow Simulation Analysis of Aerodynamic Drag Reduction of Design Chassis

Raghwani Khushal[1], Dheeraj Yadav[2], Kiran Pimple[3], Chetan Pawar[4]
[1, 2, 3, 4] Dept of Mechanical
[1, 2, 3, 4] D Y Patil College Of Engineering, Ambi.

Abstract- *To save the energy and to protect the Global environment, fuel consumption reduction is a primary concern of the modern car manufacturers. Drag reduction is essential for reducing the fuel consumption. Designing a vehicle with a minimized Drag resistance provides economical and performance advantages. Decreased resistance to forward motion allows higher speed for the same power output, or lower power output for the same speed. The shape is an important factor for drag reduction. To design an efficient shape of the car that will offer a low resistance to the forward motion, the most important functional requirement today is the low fuel consumption. The resistance, termed as the drag force (or the drag coefficient in non-dimensional terms), is a strong function of the shape of the car. This suggests it is important how the fluid particles move about the car and how fast they move along their path.*

The main intention behind this project is to compute the Drag co-efficient, Drag force and moments on low mass vehicle by using CFD software (computational fluid dynamics).

I. INTRODUCTION

Embracing the 21st century in stride, virtually all manufacturers have adopted some form of computer aided design, Engineering, Manufacturing and Analysis. It is a common belief that they can stay ahead by continuously introducing new products that are differentiated by the latest Technology revolution, innovative designs, higher functionality and superior quality. The best technology to integrate expertise knowledge and companies unique practice is the CAD/CAM and analysis software packages; a comprehensive suite of solution to meet the present day demand.

II. OBJECTIVE

Decreasing the fuel consumption of road vehicles, due to environmental and selling arguments reasons, concerns car manufacturers. This gives steps of analysis for flow simulation in solidwork Flow simulation.

Decreased resistance to forward motion allows higher speeds for the same power output, or lower power output for the same speeds.

The main aim for reducing drag resistance is:

- Fuel consumption reduction and Performance increasing.

Table 1 shows the $C_D$for various objects

Circular cylinder normal to stream	0.35 to 1.2, depending on Reynolds number
Flat plate normal to stream	2.0
Rectangular section	0.9 to 3, depending on aspect ratio
Sphere	0.1 to 0.4 depending on Reynolds number
Disk normal to stream	1.2
Cube	1.1
Saloon car	0.35
Articulated container truck	0.7

DRAG

For bluff bodies where the drag is mainly due to direct stresses, the drag coefficient is defined in terms of a dimension normalto the flow. This is in contrast to streamlined bodies where the drag is mainly due to shear stress, where a dimension parallel to the flow is used. This is discussed further in the Appendix on drag coefficients and their definitions.

The drag is sometimes divided into forebody drag, due to the pressure distribution around the front, and base drag, due to that round the back. This is useful because, by and large what goes on in the wake does not depend to any extent on the forebody, provided separation occurs at the same place, and similarly the flow over the forebody is not influenced much by changes in the base region.

A simple correlation that seems to hold for a variety of two dimensional shapes is that the drag coefficient increases as the angle between the separating shear layers increases.

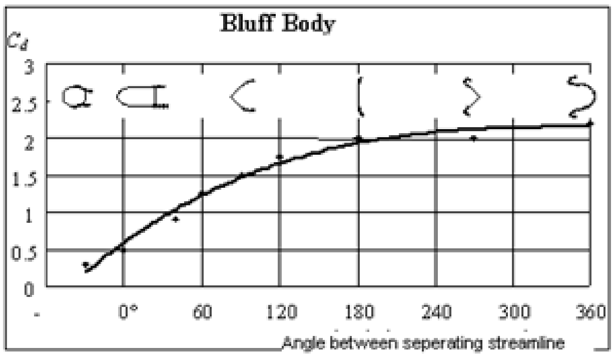


Fig.6: Co-efficient of drag versus angle between separating streamlines

For example, a 90° angle with its apex upstream has a C_D of about 1.7, a flat plate (180°) is about 2.0, and an angle with its apex downstream (270°) is about 2.1. Further examples are given in the following graph;

a similar relationship seems to hold for axially symmetric bodies - cones, disks, spheres.

III. CFD (COMPUTATIONAL FLUID DYNAMICS)

CFD (Computational fluid dynamics) is a set of numerical methods applied to obtain approximate solution of problems of fluid dynamics and heat transfer

According to this definition, CFD is not a science by itself but a way to apply methods of one discipline (numerical analysis) to another (heat and mass transfer).Computational fluid dynamics(CFD) has come out as a modern alternative for reducing the use of wind tunnels in automotive engineering. CFD is now being intensively applied to various stages of aerodynamic design of automobiles.

IV. DRAG REDUCTION METHOD

Aerodynamic drag is the main obstacle to accelerate a vehicle when it moves in the air. About 50 to 60% of total fuel energy is lost only to overcome this adverse aerodynamic force. Reduction of aerodynamic drag has become one of the prime concerns in vehicle aerodynamic.

Analysis.

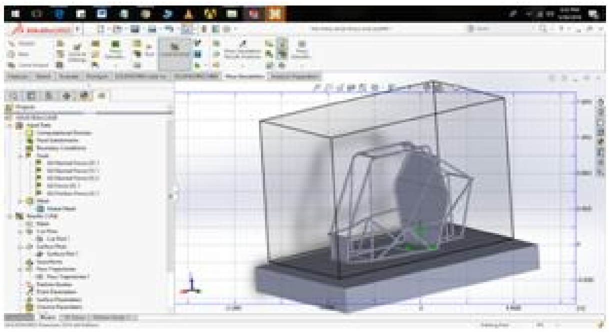
<u>STEP 1</u>: GEOMETRY GENERATION USING CATIA V5R20 SOFTWARE

<u>STEP 2</u>: Analysis In SOLIDWORKS Flow Simulation.

PRE- PROCESSING IN FLOW SIMULATION

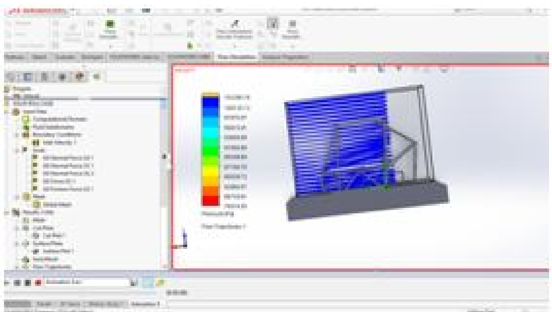
Initially we have to generate domain on which we are going to work .As shown in fig:

- APPLYING BOUNDARY CONDITIONS AND DOMAIN MATERIAL SPECIFICATION WITH THE FOLLOWING STEP:
- First save the project into specific folder.
- Go to wizard in toll bar.
- Insert project name.
- Click next
- Apply unit in which your inputs are based on.
- Select analysis type as External > Consider closed cavity select both option.
- Click next
- Select input domain flow > Gases > Air > Next.

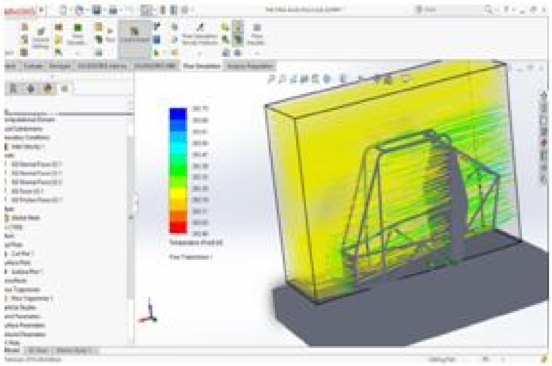


PROCESSING OPERATIONS IN FLOW SIMULATION:

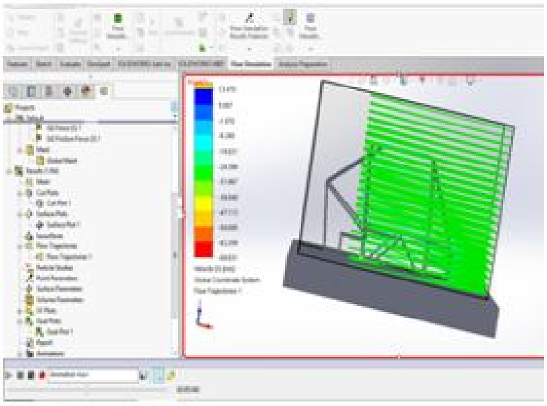
Pressure distribution along the body of chassis.



Velocitydistribution along the body of chassis.



Temperaturedistribution along the body of chassis:



Calculation of drag force:

Drag force is mainly depends upon frontal projected area and co-efficient of drag.

FD = 0.5*CD*A*V2

Where,FD = Drag force, CD = Co-efficient of drag. A = Projected area V = Velocity

V. CONCLUSION AND FUTURE SCOPE

The aerodynamic design of vehicles is an area where a lot of improvements will appear in the near future, in concern of drag reduction. The guidelines pointed out in the text are of a general nature that can be implemented in most modern road going vehicles.

REFERENCE

[1] Aerodynamics of Road Vehicles From Fluid Mechanics to Vehicle Engineering,Edited by Wolf-Heinrich Hucho,introduction to automobile aerodynamics.

[2] W. H. Hucho, Aerodynamics of Road Vehicles, 4th edition, Warrendale: Society of Automotive Engineers (SAE), 1998

[3] Elsevier Ltd,Aerodynamic study of Human Powered Vehicles.FirozAlama*, Pedro Silvaa, Gary Zimmer b.

[4] Numerical Study on Aerodynamic Drag Reduction of Racing Cars.Joseph Katz, Race Car Aerodynamics-Designing for Speed,firsted.,Bently Publishers, 1995

[5] Mogre, M. R. (2012). Comparative study between Automatic and Manual transmission car. In International conference on mechanical, Automobile and Biodiesel Engineering (pp. 308-312) , "International conference on Mechanical Aoutomobile and biodiesel Engineering(ICMA BE 2012 DUBAI) "