

Aerodynamic development of land speed record car engineering essay



**ASSIGN
BUSTER**

This project is focused on the baseline aerodynamic analyses and optimisation of a Land Speed Record vehicle in terms of performance of drag and aerodynamic stability, and thereby, speed the vehicle can attain. The vehicle 'Stay Gold' shown below belongs to the David Tremayne, a Formula One journalist. It is his plan to break the current British Land Speed Record by achieving a speed of about 350 m/s. A photograph of the vehicle in its current form can be seen below.

C: UsersCecilDesktopDSC07489. JPG

Figure 1, (20)- Photograph of the Land Speed Record vehicle in its current form.

In aerodynamic analyses, the prohibitive costs of conventional wind testing along with the advent of computing power, characterised by its decreasing cost has brought the applications of Computational Fluid Dynamics (CFD) to the fore front of research and industrial applications, (2). The complications of actual wind tunnel testing ranging from differences in boundary conditions to the scale and mounting of the object is covered well in existing literature, (1). CFD is a method for analysing complex fluid flow problems using numerical methods to solve the Navier-Stokes governing equations.

This report begins with the study of high speed vehicle aerodynamics to understand the important physical phenomena taking place, i. e. Air flows linked with different parts of the vehicle. CFD has been established as a proven tool used to perform baseline simulations with different turbulence models, boundary conditions and grids. Consequently, the influence of compressibility by running cases with progressively faster operating speeds

up to Mach 0.5 and the effects of varying the geometry of nose and body of the vehicle using low-drag fairings are studied.

Literature Review:

High Speed Vehicle Aerodynamics:

Aerodynamics is the study of various forces when a body is in motion. As we know, any vehicle moving through a fluid experiences forces induced by the fluid on it. When a vehicle moves forward it displaces the static air in front of it and hence disrupts the air flow around the body. These induced fluid forces can be characterised into three main forces which can be represented on a Cartesian coordinate system as shown in the figure below, (19).

Figure 2, (19)- Forces acting on a vehicle represented on Cartesian coordinate system

The vertical force along the Z axis is called lift. In this context, the vertical force is usually pushing the car towards the ground. Lift force then would be negative. Instead, the term downforce will be used, which is the positive vertical force towards the ground.

The horizontal force moving in the opposite direction as the vehicle along the X axis is called drag force. Drag force is created by the vehicle's resistance to motion moving through the air. Drag will always be negative with this axis system, although in the results it will be displayed as positive, (19).

The horizontal lateral force along the Y axis is called side force which occurs due to strong cross winds or by vehicles being in proximity to each other.

The magnitude of these forces depends on various factors like the geometry & speed of the vehicle, mass of the fluid, viscosity & compressibility. These
<https://assignbuster.com/aerodynamic-development-of-land-speed-record-car-engineering-essay/>

three forces are the basic aerodynamic forces that act on a moving solid body. We concentrate on the reducing the drag force alone to achieve higher speeds.

Aerodynamic Flows:

There are various kinds of aerodynamic flows considered important in a vehicle ranging from flows associated with the external shape of the vehicle to the flows existing in the lubrication and cooling systems of the vehicle which are called external and internal flows respectively, (3).

C: UsersCecilDesktopUntitled. jpg

Figure 3, (4) – Attached flows and separated flows over a body.

The streamlines are the curves associated with a pictorial description of a fluid motion, in this case air particles move along the streamlines, (4). Using this definition we can differentiate the flows. When the streamlines near the solid surface follow the shape of the body, the flow is considered to be attached and if the flow does not follow the shape of the body, the flow is considered to be separated, (4). As seen from the figure above, separated flows leave behind trailing vortices which result in an unsteady wake flow which can be seen in the figure below.

C: UsersCecilDesktop1-s2_0-S0167610501001611-gr11. gif

Figure 4, (5) – Trailing vortices in the wake of a conventional fastback car

It is also important for Race Car engineers to know whether the flow is laminar or turbulent since features such as flow separation and vehicle drag can change dramatically within these two flows, (4). When a body travels in an undisturbed environment, the flow can be considered laminar. Conditions such as winds or the motion of other vehicles directly affects the flow causing turbulence. Turbulence is a chaotic and random state of motion develops in which the velocity and pressure change continuously with time, [22].

Characteristic of Aerodynamic flows:

External Automotive flows can be characterised as those involving excessive flow separation, transitional flows, strong cross flows and streamline curvature with a turbulent wake interacting with the ground boundary layers, (5). The prevailing areas where the separation of air flow takes place are the front and rear windshields. This separation of air flow leads to change in pressure over the surface of the vehicle which constitutes the aerodynamic drag of the vehicle. Pockets of high and low velocities are created around the vehicle because of this separation. The variation of pressure over a vehicle is shown in the figure below and is measured by a coefficient of pressure, denoted by C_p . According to Bernoulli's equation, the low pressure region denotes high velocity and high pressure region denotes low velocity. C_p is given by the ratio of the difference in pressure on any point of the vehicle to the dynamic pressure.

C: UsersCecilDesktop109741_3mg. jpg

<https://assignbuster.com/aerodynamic-development-of-land-speed-record-car-engineering-essay/>

Figure 5, (4) – Variation of pressure over a vehicle

Boundary Layer:

The layer between the vehicle and the moving air where the fluid flow is stagnant or less is called the boundary layer and is a significant aspect at high speeds. When the body is in motion, a relative velocity is created between the vehicle and the air around it due to the fluid viscosity. Boundary layers may be either laminar or turbulent depending on the value of the Reynolds number. For lower Reynolds numbers, the boundary layer is laminar and the velocity changes uniformly as one moves away from the wall and for higher Reynolds numbers, the boundary layer is turbulent and the velocity is characterized by unsteady (changing with time) swirling flows inside the boundary layer, (7). In real environment it is inevitable that the boundary layer detaches from the solid body which results in a large increase in the drag on the body. So at high speeds, it is important to maintain an attached and laminar boundary layer with a streamlined shape (4).

C: UsersCecilDesktopboundlay. gif

Figure 6, (7) – Boundary layer on a surface of a vehicle

Compressibility effects:

Compressibility is the measure of change in volume of the air relative to the speed. We are dealing with subsonic speeds (less than Mach 1) where the air acts as if it's an incompressible fluid meaning the density will remain constant though the velocity and pressure are variable, (6). By Bernoulli's principle when air enters a body or part of the vehicle, air must travel faster

to get to the other side as the bypass air varying the pressure and velocity. The velocity and pressure return to their original form at the outlet.

Importance of the speed of sound:

Sound is the pressure disturbances radiating in all directions from the vehicle. In subsonic flight sound waves radiate from all points on the vehicle and can travel faster than the vehicle itself as shown on the figure on the left.

C: UsersCecilDesktopasw. png

Figure 6, (6) - Propagation of sound waves in subsonic and supersonic speeds
C: UsersCecilDesktopas. png

As the vehicle travels at higher speeds, these sound waves pile up at the nose of the aerofoil and create shock waves as shown in the figure on the right. These shock waves are created due to change in pressure & velocity of air flow and these waves cannot get ahead the originating point at the speed of sound. There are different kinds of shock waves which are discussed below. Oblique Shock waves are formed on sharp edges of the body with the air surface changing in the direction of air flow, basically on leading and trailing edges of the airfoil, (6). Normal shock waves are formed in front of a blunt body or on the body itself. The molecules pile up at the front and form a detached wave called the 'bow' wave, (6). Expansion shock waves are formed in the regions of separation on the body or airfoil. Shock waves are very important in high speed aerodynamics as it affects the change in direction of the fluid flow and are relatively negligible in subsonic flows.

Relevance of Aerodynamic Drag:

In aerodynamics, drag is defined as the force that opposes forward motion of the vehicle through the atmosphere and is parallel in the direction of free stream velocity of the air flow which can be overcome by thrust in order to achieve forward motion, (8). Generally in racing it is important to have to downforce to keep the vehicle stable on the ground. When going at speeds over 100 mph, the real drag is experienced.

The aerodynamic drag is denoted by C_d and is given by the formula,

$$C_d = \text{Drag force} / (\text{Dynamic pressure} * \text{Area})$$

A body moving through a fluid experiences drag which can be divided into two components, frictional or viscous drag and pressure drag, (11). Frictional Drag is developed due to friction of fluid and the surface it is flowing on, commonly associated with development of boundary layers, (11). Pressure drag is formed from the eddying (turbulent) motions set up by the fluid as it passes over the body which is associated with the formation of wake behind the vehicle. Hence the geometry of nose and body shape plays an important role in reducing drag on vehicle, (12).

The figure below shows the driving force required to propel the vehicle forward at a constant speed as a function of the aerodynamic drag. We can see that the aerodynamic drag increases proportional to the square of speed.

C: UsersCecilDesktopUntitled. png

Figure 7, (11) - Driving Speed Vs Vehicle Speed

It is also established that the drag prediction over the front of the vehicle, especially involving attached boundary layers and, subsonic flow is far easier and more accurate than the analysis of the rear of the vehicle, (11).

Review of existing research, (13):

Extensive research has been made in this area with detailed experimentation on the widely known Ahmed model, (12). The Ahmed model is a simple geometric body that retains the main flow features, especially the vortex wake flow where most of the drag is concentrated. This model is used as a reference model to compare our results with. An illustration of this model is shown in the figure below.

C: UsersCecilDesktopUntitled. png

Figure 8 – Left: Geometric dimensions of an Ahmed model, Right: Computational domain

The Ahmed reference model is a general car type bluff body shape which is enough for accurate for flow simulations. boundary conditions used for this problem are uniform flow at the inlet & no slip on the surface of the body and a non-structural tetrahedral grid approach is applied to this geometry at $Re = 4.25 \times 10^6$, (13) . This flow was solved using incompressible Navier-Stokes formulations and the drag and pressure were measured. From the figure below, it is observed that the total pressure drag is minimal at the front portion and is high the rear slanted portion of the body. Subsonic interactions are fairly weak as the length of the body is long. C:

UsersCecilDesktopUntitled. png C: UsersCecilDesktopUntitled. png

<https://assignbuster.com/aerodynamic-development-of-land-speed-record-car-engineering-essay/>

Figure 9A,(13) – Contour fill field pressure Figure 9B, (13) – Contour fill field velocity

And as for the pressure measurements, the presence of vortices at side edges of the slant surface appears to be two dimensional with parallel isobars running over the surface

C: UsersCecilDesktopUntitled. png C: UsersCecilDesktop2. png

Figure 10A – Flow behind the rear side of the body, Figure 10B – Streamlines in the wake

Turbulent flows are completely three dimensional and unsteady. Using a time average flow, some sort of macrostructure appears to govern the pressure drag on rear end. Figure 10A shows the different wakes created due to different shapes and edges and Figure 10B shows the streamlines in wake structure.

Land Speed Record (LSR) Racing:

Ever since the inception of automobiles, there has always been an inherent drive to push the automobile to its limits in terms of performance and speed. Land Speed Record is highest speed achieved by an automobile on land. There are different classes and organisations with respect to the configurations of the vehicle, operating speeds and environment. The current vehicle belongs to the unlimited class, which is a special class for thrust powered vehicles which may be propelled using turbo jet engines and without any limitations over wheeled power, (14).

Existing research:

The closest vehicle comparable to 'Stay Gold' LSR is the JCB dieselmax (shown below) which holds the land speed record for a diesel-powered vehicle having been driven to over 350mph breaking the world record at the Bonneville Salt Flats. The aerodynamics of the car was designed entirely using ANSYS Fluent by aerodynamicist Ron Ayers whose goal was to achieve an optimal balance between aerodynamic drag, skin force and downforce, (15). It has been observed that the Cd of the vehicle was 0.17. After running a number of simulations, it is shown that higher downforce generated by the wings increases the drag on the vehicle drastically, (16).

C: UsersCecilPicturesjcb_dieselmax. jpg

Figure 11, (17) – Picture of a JCB dieselmax streamliner

Another example is the Buckeye Bullet 3 (BB 3), which operates on a battery. This vehicle was developed using an alternate aerodynamic method. In order to test the body shape and geometry a new wind tunnel model was constructed though it is time consuming and costly. The vehicle's frontal area was significantly reduced allowing the driver to be placed at an inclined position keep safety in mind, (18) though the driver in middle (DIM) configuration used in BB1 & BB 2 has better performance compared to the former. BB 3 also achieves more stability because it maintains negative pitch over higher speeds ensuring the normal loads on the tyres are not reducing, improving traction and yaw stability, (18). BB 3 had a 17 % reduce in the Cd compared to its predecessors.

C: UsersCecilPicturesBuckeye-Bullet-3-thumb-450×255. jpg

<https://assignbuster.com/aerodynamic-development-of-land-speed-record-car-engineering-essay/>

Figure 12, (18) – Photograph of a Buckeye Bullet 3 at the Bonneville Salt Flats

Computational Fluid Dynamics:

Introduction:

Computational fluid dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and associated chemical reactions by means of computer based simulation, [21]. CFD is used in wide range of industries. CFD contains a set of codes structured around the Navier-Stokes Equations (NSE) that are used to solve fluid flow problems. NSE are the governing equations of Fluid dynamics which are shown below.

C: UsersCecilPicturesnseqs. gif

Figure 16, [22] – Navier-Stokes equations of three dimensional fluid flows

The NSE consists of time-dependant continuity equation for conservation of mass, time dependant conservation of momentum equations and conservation of energy equations, [22]. The integral forms of these equations are solved using Finite Volume Method (FVM) which is the discretisation method ANSYS Fluent uses. Discretisation is a method of converting higher order integral equations into a system of algebraic equations. These set of algebraic equations are later solved by an iterative method, [23].

As mentioned earlier, at higher Reynolds numbers flows are observed to become turbulent. We used turbulence models in CFD to approximately

model the turbulence in real time environment and results obtained are deemed near accurate.

All codes in CFD contain 3 main elements: 1.) A pre-processor, 2.) A solver, 3.) A post processor, [22].

Pre-processing:

The activities involved in this stage are

Definition of geometry in the computational domain

Grid generation - discretising the domain into smaller cells (grid or mesh)

Define fluid properties and specify the appropriate boundary conditions

Solving:

There are different techniques to solve numerical equations. CFD uses finite volume method which is the most established method in different softwares.

The steps involved in this stage are

Integration of NSE over control volumes of the domain

Discretisation of resulting integral equations into a set of algebraic equations

Solution of the algebraic equations using an iterative method

Post-processing:

A large amount of work has been put into CFD packages to visualize the data with outstanding graphics due to the increasing demand in the engineering field. Some of the most popular data visualisation tools are

Domain geometry and grid display

2D and 3D surface and vector plots

Streamlined and shaded contour plot

At present, almost all Formula 1 teams use CFD to constantly optimise the aerodynamics of their cars for better performances in a race. The bottleneck of CFD was quick and efficient construction of a functional grid which has become more user friendly in modern times which makes it easy for meshing, [22].

Conclusions:

Aerodynamics & CFD play a key role in the optimisation of a Land Speed Record vehicle. The geometry, powerplant, wheel configurations and vehicle dynamics are important parameters in achieving reduced drag. Open wheel configurations cause more overall drag on the vehicle. Also the tyre's effective radius changes with speed and is necessary to test the relationship between the vehicle speed and tyre's geometric configuration due to constant downforce exerted on the vehicle by the wings. This was the case revealed in the JCB streamliner.

Project Plan:

Study Vehicle Aerodynamics - Understand the various aerodynamic flows involved in a vehicle. Get a good grip on High Speed Aerodynamics. Read about Importance of the speed of sound in achieving high speeds and influence of compressibility.

Computational Fluid Dynamics (CFD) - Reading on the background of CFD and understand how Navier-Stokes equations are derived, Numerical discretisation of equations using Taylor series. Understand turbulence modelling, grid sensitivity and boundary conditions relevant to the problem.

CFD Tutorials –

Tutorial 1 - Create basic geometry for backward facing step (2D).

Tutorial 2 - Create basic geometry for lid driven cavity (2D).

Tutorial 3 - Create 3D model of cylindrical body with a rectangular grid and run simulations.

Gantt chart:

C: UsersCecilDownloadsAerodynamic Development of LSR (1). png

Progress to date:

With the help of Dr. Carl Gilkeson's CFD tutorials, creating meshes of basic geometrical shapes and running simulation on them was possible. The first tutorial involves creating the geometry in Design Modeller for lid driven cavity. The mesh was created in ANSYS Mesh and running simulations with specified boundary conditions, velocity & pressure was done in ANSYS Fluent.

In the figure below, the mesh for the lid driven cavity is shown.

C: UsersCecilDesktopCavity- Vel Contours. png

Figure 13A - Mesh for lid driven cavity Figure 13B - Velocity contours on the top wall of the cavity C: UsersCecilDesktopCavity. png

The mesh was solved for 1000 iterations and were run on the top wall with a transitional velocity of $1.4607e-03$ ($Re = 100$), ensuring the flow is laminar. The velocity contours are filled and can be seen in figure 13B. The second tutorial involves creating a backward facing step. This model is solved (1000 iterations) for turbulent conditions so a turbulence model (k-epsilon) was created and specified with an inlet velocity of 40m/s. Also this model is discretised to second order upwind for momentum, turbulent kinetic energy and turbulent dissipation rate. The results are shown below.

C: UsersCecilDesktopUntitled. png

Figure 14A- Mesh for Back Facing Step Figure 14B- Velocity contours on the backstep C: UsersCecilDesktopMesh - Backw step. png

It is noted that the horizontal component of the velocity is negative behind the step as the flow reattaches after detaching due to high Reynolds number, [20].

3D model of a cylinder:

I: Ansys projectscylinder. jpg

Figure 15A & 15B(below) – Mesh for 3D cylinder in a rectangular grid

In the figure above, a cylindrical mesh in a rectangular grid was created in ANSYS Mesh. A rectangular grid is created in the domain to solve finite volume system of equations and to get more accurate solutions. The model <https://assignbuster.com/aerodynamic-development-of-land-speed-record-car-engineering-essay/>

was solved for constant pressure at the inlet to get reversed flow on all faces using hybrid initialisation as seen below.

I: Ansys projects12. jpg

References

- [1] P. R. Spalart, ' Strategies for turbulence modelling and simulations', Boeing Commercial Planes(Feb 1999).
- [2] W. H. Hucho, ' Aerodynamics of Road vehicle' - 4th edition.
- [3] `New Directions in Race car aerodynamics', Joseph Katz.
- [4] Joseph Katz, `Race Car Aerodynamics', 2nd edition.
- [5]] Ahmed, S. R. , Gawthorpe, R. G. and Mackrodt, P. -A.(1985) ' Aerodynamics of Road- and Rail Vehicles', Vehicle System Dynamics, 14: 4, 319-392
- [6] `High Speed Aerodynamics', Seminar, Harry L Whitehead.
- [7] <http://galileo.phys.virginia.edu/classes/311/notes/fluids2/node11.htm>
- [8] http://www.pilotfriend.com/training/flight_training/aero/drag.htm
- [9] http://www.princeton.edu/~asmits/Bicycle_web/blunt.html
- [10] Miles Jackson , B. Taylor Newill and Perry Carter , 'Racecar Aerodynamic Optimization for an E-1 Class Streamliner Using Arbitrary Shape Deformation' , SAE Technical paper 2007-01-3858.

[11] Hiroyuki Ozawa, Dai Higashida, 'Development of Aerodynamics of a Solar Race Car', Honda R&D, 1998 SAE.

[12] SiniÅija Krajnovic, Lars Davidson, 'Flow Around a Simplified Car Part 1: Large Eddy Simulation',

[13] 'CFD Modelling of Flow around the Ahmed vehicle model', Gerardo Frank and Jorge D'Elia, Centro Internacional de Metodos Compucionales en Ingeneria.

[14] <http://www.landspeed.com/archive/classroom/classlsrbasics.html>.

[15] http://www.newmaterials.com/News_Detail_Aerodynamics_of_jcb_dieselmex_car_designed_entirely_with_cfd_code_fluent_9408.asp#axzz2H86gSGFW

[15] http://www.newmaterials.com/News_Detail_Aerodynamics_of_jcb_dieselmex_car_designed_entirely_with_cfd_code_fluent_9408.asp#axzz2H86gSGFW

[16] 'Aerodynamic Development of Buckeye Bullet Electric LSR', Carrington Bork, Department of Mechanical Engineering, Ohio State University.

[17] http://www.carsbase.com/photo/photo_full.php?id=45469

[18] www.buckeyebullet.com/BB3.html

[19] 'Fundamentals of Vehicle Dynamics', Thomas D. Gillespie, Society of Automotive Engineers, 400 Commonwealth drive, Warrendale, PA 15096-0001.

[20] CFD tutorials, Dr. Carl Gilkeson, University of Leeds.

[21] <http://www.grc.nasa.gov/WWW/k-12/airplane/nseqs.html>

<https://assignbuster.com/aerodynamic-development-of-land-speed-record-car-engineering-essay/>

[22] `An Introduction to CFD', H K Versteeg and Malasekara, 2nd edition.