Aerodynamic Pressure Drag Utilization to Impart Thrust to the body

Nadish Saini[1], Shivam Maitreya[1], Udit Dureja[1], and Amit Chauhan[1] +

[1]Department of Mechanical Engineering, UIET, Panjab University,
Chandigarh-160014, India

Abstract: Any body moving relative to fluid experiences a pressure drag. In present work, analysis has been carried out to study this pressure drag on a general simplified body and also the velocity profile in flow field using ANSYS (version 12) software. An attempt has been made to utilize the pressure difference at the front and rear face of the body and convert it into useful thrust. The same has been achieved by incorporating a nozzle like passage in the design of the body. Analysis has been performed at sonic and near sonic velocity and the result obtained confirms that considerable thrust can be generated from the pressure difference which was otherwise adverse to the motion of the body. The estimated thrust generated at sonic velocity is about 353.2 N-s/Kg. Also introducing such passage helps to soothe the eddies which were being formed in the wake region.

Keywords: Pressure drag, thrust, computational fluid dynamics, converging passage.

1. Introduction

When a body moves relative to a fluid it experiences a pressure drag because of the phenomena of boundary layer separation. To elucidate, a body with a cylindrical cross section gets its flow separated at the ends of diameter perpendicular to low resulting in the formation of wake region. This wake region experiences eddies whose intensity increases with the increase of Reynolds's number. These eddies result in huge losses. Also the wake region so formed is a site of low pressure. The pressure at the front of cylinder is quite high as compared to this wake region, as a result there exists a pressure difference at front and rear face of the cylinder, or of any body for that matter. Owing to this pressure difference, the body experiences a drag known as pressure or form drag and the body must do some work to overcome this pressure drag. If the air from the front and sides of a body is channeled and connected through a nozzle to the rear of body into the wake region where pressure is low, the pressure drag could be converted, which was earlier detrimental to the motion of the body, into kinetic energy of the out coming fluid. Owing to the Impulse Momentum theorem, the same principle which governs rocket propulsion, useful thrust can be generated. The equation governing the exit velocity of nozzle if the inlet velocity is zero is given as [1]:

$$V_e = \frac{T \times R}{M} \times \frac{2 \times k}{k-1} \times \left[1 - \left(\frac{P_e}{P} \right)^{\frac{k-1}{k}} \right] \tag{1}$$

Where, V_e= Exhaust velocity at nozzle exit, m/s; T= absolute temperature of inlet gas (K); R= Universal gas law constant = 8314.5 J/(kmol·K); M= the gas molecular mass, kg/kmol; k= c_p / c_v = isentropic expansion factor; c_p = specific heat of the gas at constant pressure; c_v = specific heat of the gas at constant volume; P_e= absolute pressure of exhaust gas at nozzle exit, Pa; P = absolute pressure of inlet gas, Pa.

Computational Fluid Dynamics (CFD), in one or another form, is based on the fundamental governing equations of fluid dynamics- the continuity, momentum, and energy equations. The fundamental basis of almost all CFD problems is the Navier–Stokes equations, which define any single-phase fluid flow. The continuity and momentum equations are given as [2]:-

$$\frac{\partial \rho}{\partial t} + \nabla(\rho \times u) = 0 \tag{2}$$

+   Corresponding author.
    E-mail address: chauhan98@rediffmail.com.

Where, ρ is fluid density, t is time, and $\mathbf{u}$ is the flow velocity vector field.

$$\rho \times \left(\frac{\partial v}{\partial t} + v \times \nabla v \right) = -\nabla p + \nabla \mathbb{T} + f \tag{3}$$

Where, v is the flow velocity, ρ is the fluid density, p is the pressure, $\mathbb{T}$ is the (deviatoric) stress tensor, and f represents body forces (per unit volume) acting on the fluid and ∇ is the del operator. This is the statement of conservation of momentum in a fluid and it is an application of Newton's second law to a continuum, in fact this equation is applicable to any non-relativistic continuum and is known as the Cauchy momentum equation.

2. Methodology

Anybody that moves relative to a fluid has to spend considerable amount of energy to overcome pressure drag. This project intends to utilize this pressure drag to provide a thrust to a body. It is based on a manifestation of the basic law of conservation of energy- changing pressure energy into kinetic energy. It will use the otherwise detrimental pressure drag to the advantage of a moving body by providing an extra thrust without affecting fuel consumption. This would revolutionize the aerodynamic modeling. The concept which this project intends to introduce has been proved by analyzing a simplified general body in air flow keeping in view the research of Jason et al. [3] which demonstrates the influence of geometry simplification of a vehicle model to perform simulations. The results of this project can be used as a benchmark for more complicated designs. A general aerodynamic body has been analyzed using the ANSYS (FLUENT) software. The software offers two models Large Eddy Simulation (LES) and Detached Eddy Simulation (DES), which can be implemented in the project. Both models depict real flows. The model used in this project is DES as the model gives reasonably correct results as shown by research of XU Chang-yue et al. [4]. This model attempts to treat near-wall regions in a Reynolds Averaged Navier Stokes (RANS)-like manner, and treat the rest of the flow in an LES-like manner. The model was originally formulated by replacing the distance function d in the Spalart-Allmaras (S-A) model with a modified distance function $\tilde{d}$ given as [5]:

$$\tilde{d} = min[d, C_{DES}\Delta] \tag{4}$$

Where, C_{DES} is a constant and Δ is the largest dimension of the grid cell in question. This modified distance function causes the model to behave as a RANS model in regions close to walls, and in a Smagorinsky-like manner away from the walls. The DES approach may be used with any turbulence model that has an appropriately defined turbulence length scale (distance in the S-A model) and is a sufficiently localized model.

The parameters used in this project are: realizable K –epsilon with delayed DES; the model constants are C_{DES}: 0.61, C2-Epsilon: 1.9, Turbulent Kinetic Energy (TKE) Prandtl Number: 1, Turbulent Dissipation Rate (TDR) Prandtl Number: 1.2, the boundary conditions are stationary wall with a specified shear stress of 290 pascal (X-component), turbulence intensity of 10% and hydraulic diameter of 4m, the direction of backflow is normal to boundary with radial equilibrium pressure distribution, the materials used are air as fluid and aluminium as the solid body.

3. Results and Discussions

3.1 Analysis of General Model

The figure 3.1 illustrates the general model selected for analysis. The figure shows a converging passage incorporated in the design. The cuboid in the figure shows the flow field and the body carved out of flow field is distinctly visible. The blue section in the figure shows inlet section i.e. the section through which air enters perpendicular to the surface. The front face of the model is the symmetry plane.

3.2 Mesh Generation

The figure 3.2 shows the mesh generated for flow field with passage by ANSYS Meshing software for analysis.
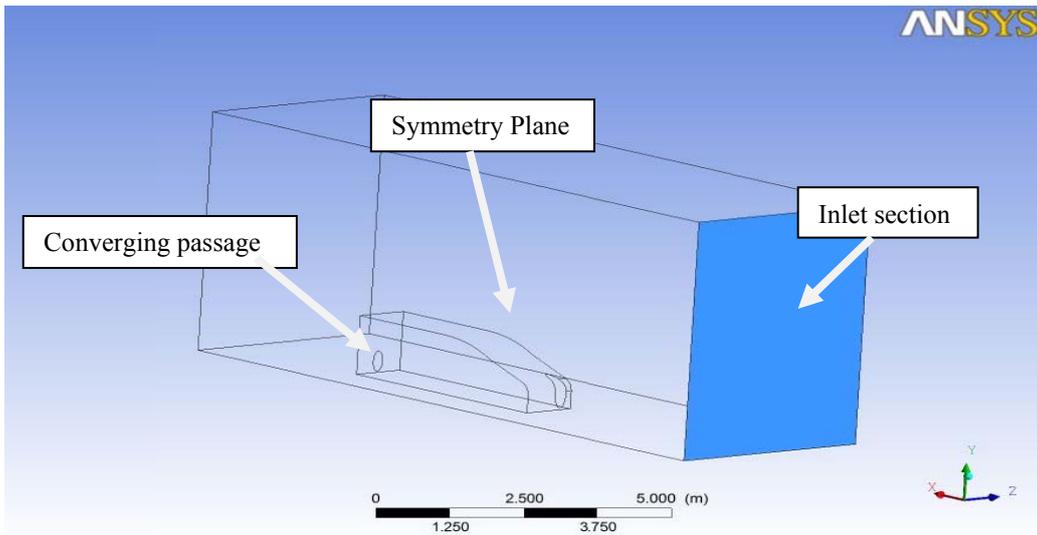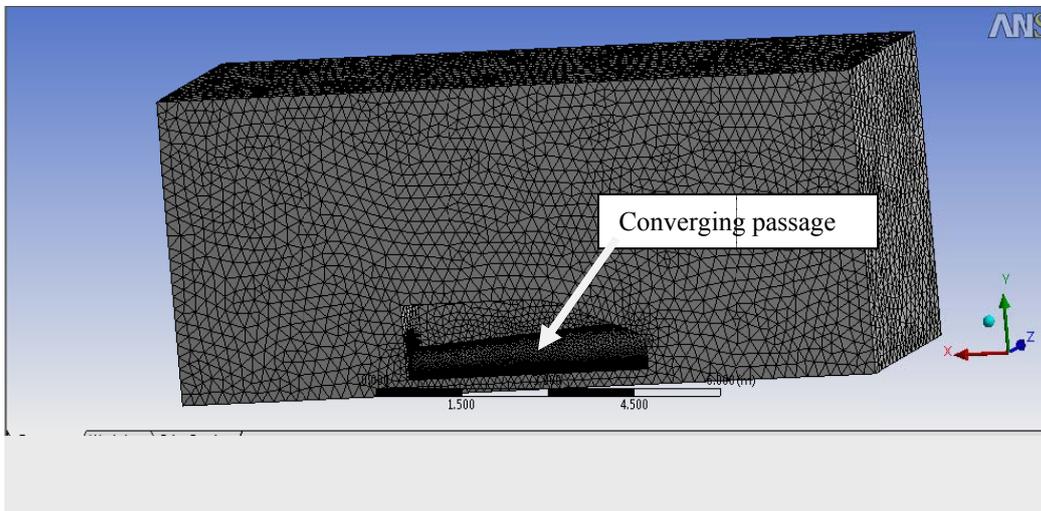
Fig. 3.1:General body under analysis-converging passage
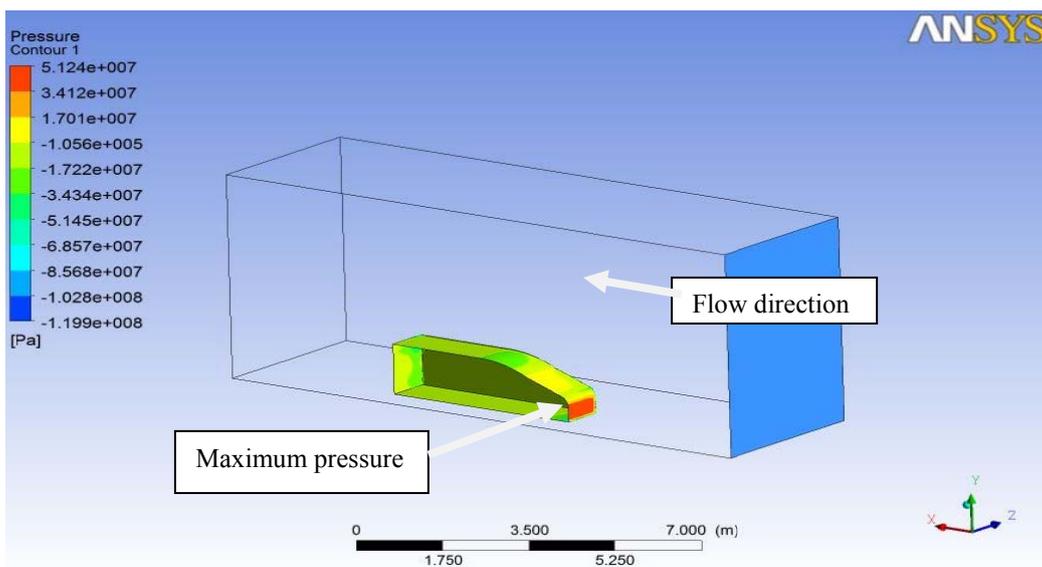

Fig. 3.2: Mesh for flow with passage.

Fig. 3.3: Pressure contours on the aerodynamic body without passage at 300 m/s.

3.3 Pressure contours on the model

The figure 3.3 illustrates the pressure contours on the aerodynamic body. It shows that maximum pressure is experienced at the front face of the body and that the pressure at the back face of the body is significantly lower than that at the front face. The figure 3.4 depicts the pressure contours on the body with the converging passage incorporated in it. There is a gradual decrease in the pressure through the passage from the inlet section to the outlet section, which can be compared from the legend. This large difference in the pressures at inlet and outlet section is converted into kinetic energy by the use of the converging passage incorporated in the body.
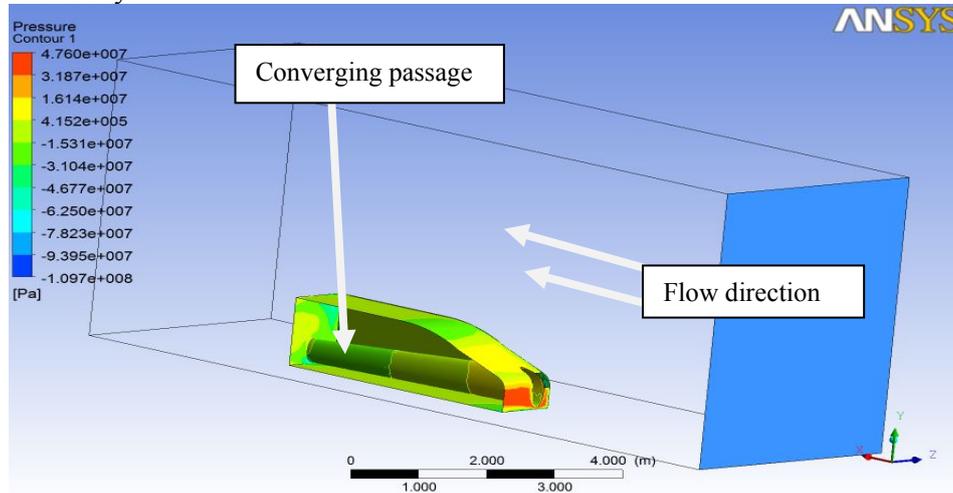

Fig. 3.4: Pressure contours on the aerodynamic body with passage at 300m/s.

3.4 Analysis at 300 m/s

The figure 3.5 depicts the velocity vector distribution in the flow field when the body does not have the converging passage. The dark blue region at the rear side of the body represents the wake region.The figure 3.6 represents the velocity vector distribution in the flow field when the converging passage has been introduced in the body. As it is visible in the figure the red region at the rear end of the body depicts the high velocity coming out of the outlet section of the converging passage (having smaller cross-section). This high velocity is highly coherent and tends to straighten out eddies that were being formed otherwise. The high velocity also creates a forward thrust for the body as is given by the impulse momentum theorem. Thus for unit mass flow rate thrust produced can be estimated from the legend:-

Thrust per unit mass flow rate = $V_2 - V_1$ = (4.518e+002) – (1.129e+002)
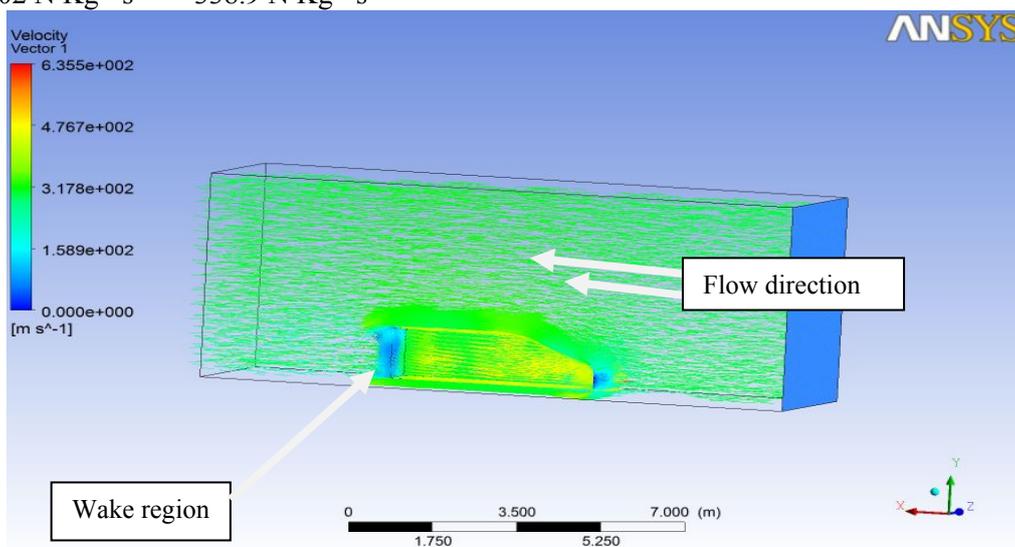=3.389e+002 N Kg^{-1} s   =338.9 N Kg^{-1} s

Fig. 3.5: Velocity vector distribution in the flow field without passage at 300 m/s.
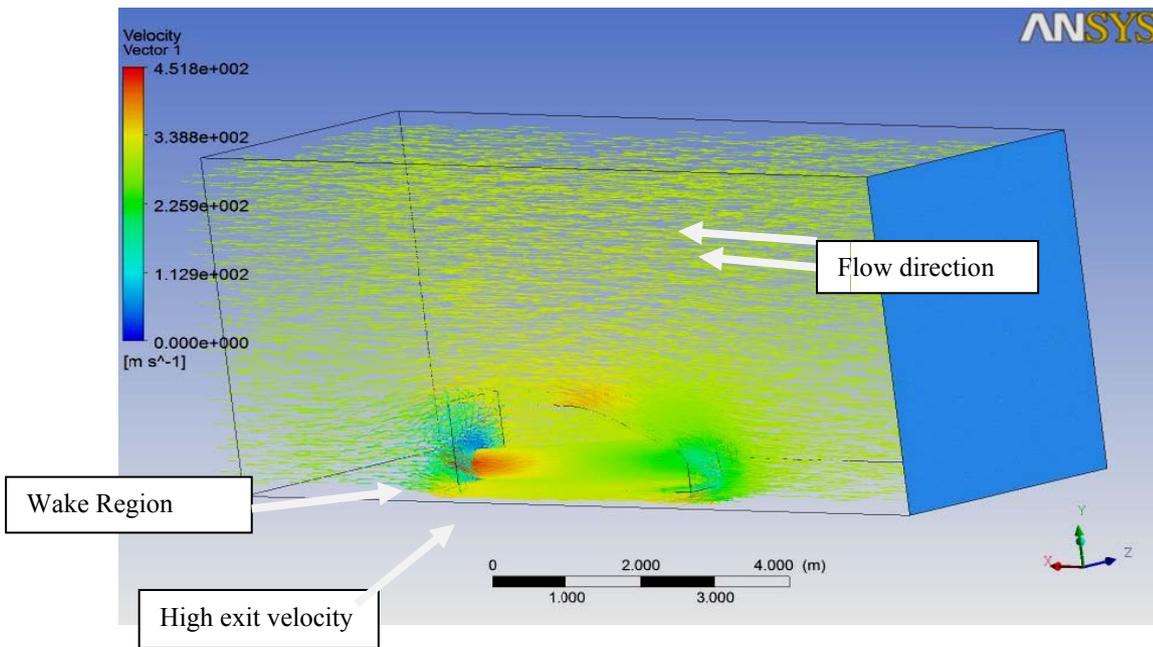
Fig. 3.6: Velocity vector distribution in the flow field with passage at 300 m/s.

3.5 Analysis at Sonic velocity (330 m/s)

When the body was analyzed at sonic velocity of air the results obtained were also positive. The air coming out of the exit section of the converging passage was at high velocity as is shown by the red region in the figure 3.7 at the exit of the passage.

The thrust obtained in this case may be estimated as:

Thrust per unit mass flow rate = $V_2 - V_1$ = (5.032e+002) – (1.500e+002)

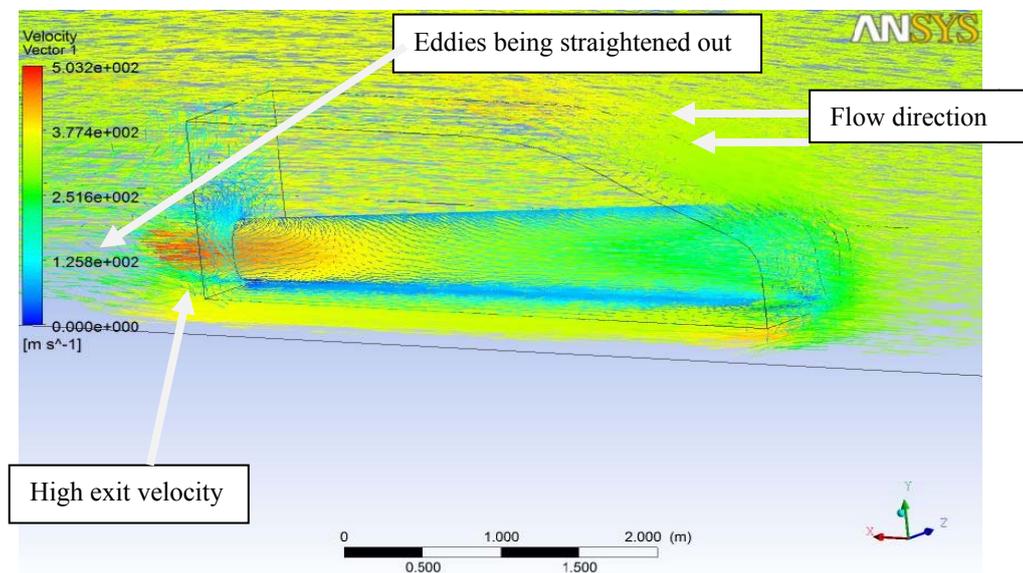$\quad$ =3.532e+002 N Kg^{-1} s=353.2 N Kg^{-1} s

Fig. 3.7: Velocity vector distribution at 330 m/s (magnified view).

4. Conclusion

Just by bringing about certain changes in design of the body and channeling the ambient air to pass through the body, it was possible to impart an additional thrust to it by converting the available pressure difference into kinetic energy of air. The velocity of fluid at exit section is quite high and also highly coherent. This will tend to straighten out eddies in the wake region and help reduce wake losses besides thrust generation as shown in figure 3.7. If such designs are introduced into the body of a rocket such that the passage channels

the ambient air into the exhaust of the rocket, from where all the exhaust gasses of propulsion exit the rocket, not only it would create an additional thrust for the rocket but it would also increase the mass flow rate of the exhaust gasses which would in turn further increase the thrust for the rocket according to impulse momentum theorem. In super cars if such a design is implemented and the datum of the exit section of the converging passage is kept at a height higher than the datum of the inlet section of the passage, then the air would be forced to move in an upward direction. This would in turn create a downward traction force according to the Newton's Law that every action has an equal and opposite reaction. On analyzing the general aerodynamic body with and without the converging passage, positive results have been obtained at both sonic velocity and near sonic velocity in terms of thrust generation which is evident from the increased velocity at the outlet section.

References

[1] Theodore Stevens and Henry M. Hobart, Steam Turbine Engineering (New York, N.Y., The MacMillan Co., 1906), pages 24-27.

[2] Computational Fluid Dynamics- The Basics with Applications by John D. Anderson Jr. Tata McGraw Hill Publishers – 1995 (page 76)

[3] Jason M. Ortega, Tim Dunn, Rose McCallen, and Kambiz Salari, Computational Simulation of a Heavy Vehicle Trailer Wake, Preprint UCRL-JC-152549, submitted to the United Engineering Foundation, Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains Conference, Monterey, CA, December 2-6, 2002.

[4] XU Chang-yue, CHEN Li-wei, LU Xi-yun, Large-eddy and detached-eddy simulations of the separated flow around a circular cylinder, Journal of Hydrodynamics, Ser.B, 2007,19(5):559-563.

[5] Spalart, P. R., Jou, W.-H., Stretlets, M., and Allmaras, S. R. (1997), "Comments on the Feasibility of LES for Wings and on the Hybrid RANS/LES Approach", Advances in DNS/LES, Proceedings of the First AFOSR International Conference on DNS/LES.