Graduate Research in Engineering and Technology (GRET)

Volume 1
Issue 5 *Reviews on Modern Technologies for Aircraft and Aero-Engines.*

Article 8

April 2022

A REVIEW ON BASE DRAG REDUCTION METHODS

Kishore Kumar S Mr.
Gitam University, kanna.aero@gmail.com

Srinivas Pendyala Dr.
Gitam (Deemed to be University), spendyal@gitam.edu

Y D Dwivedi Mr
GITAM University, yddwivedi@gmail.com

Recommended Citation

S, Kishore Kumar Mr.; Pendyala, Srinivas Dr.; and Dwivedi, Y D Mr (2022) "A REVIEW ON BASE DRAG REDUCTION METHODS," *Graduate Research in Engineering and Technology (GRET)*: Vol. 1: Iss. 5, Article 8.
DOI: 10.47893/GRET.2022.1073
Available at: https://www.interscience.in/gret/vol1/iss5/8

BASE DRAG REDUCTION METHODS

S KISHORE KUMAR [1], SRINIVAS PENDYALA [2], Y D DWIVEDI [3]
[1,2]Department of Mechanical Engineering, Gitam School of Technology, Hyderabad 500043, India
[3]Department of Aeronautical Engineering, Institute of Aeronautical Engineering, Hyderabad 500043, India.
Email:ksandan@gitam.edu, spendyal@gitam.edu ,yddwivedi@gmail.com

---

Abstract

In this project-based learning we are going to know about how the base drag are acting on an object to automobile and aircrafts and it deals with the equations of the drag in fluid dynamics and aerodynamics,and we come to know that what steps are we going to follow how to overcome the drag in vehicles. Reducing base drag of two dimensional and axisymmetric bodies having a blunt base. These methods include splitter plates, both thin and thick, splitter wedges, base bleed. boat-tailing and various types of serrated trailing edges. These methods include splitter plates, both thin and thick, splitter wedges, base bleed, boat-tailing and various types of serrated trailing edges. The effectiveness of the various devices in reducing base drag is shown and compared. In some cases, their influence on the lift of an airfoil is also indicated. Axisymmetric bodies, e.g., missiles, often have a blunt base. The corresponding base drag usually is an appreciable part of the total drag and, if the base drag is not reduced by suitablemeans, can remarkably reduce the overall performance of the system. Under most circumstances it is desirable to have as small a base drag as possible.

Keywords: CFD (computational fluid dyanmics), wind tunnel visualization, axisymmetric bodies,

1. Introduction

Fuel efficiency of vehicle is one of the main interests in the automotive industry due to it being as one the main factors that affect the operating cost of transportation. Aside from engine efficiency and performance, drag reduction is one of the main contributors to vehicle’s fuel efficiency. The aerodynamic performance has been taken into account in car designs but for heavy road vehicles such as tractor-trailer, major design change to achieve high aerodynamic performance is impossible. This is mainly because it will affect cargo capacity since, to achieve a high aerodynamic performance, tapered shape and slander body is required for the vehicles to minimize the pressure drag. Axisymmetric bodies, e.g. missiles, often have a blunt base. The corresponding base drag usually is an appreciable part of the total drag and, if the base drag is not reduced by suitable means, can remarkably reduce the overall performance of the system. Under most circumstances it is desirable to have as small a base drag as possible. Therefore, during the last 25 years many methods for reducing base drag have been developed. Often the periodic lift forces inducing vibrations also decrease if the base drag is reduced. One of the most important aerodynamic performance characteristics for the projectile's shell is the total drag. The total drag for projectiles can be divided into three components: (i) pressure drag (excluding the base), (ii) viscous (skin friction) drag, and (iii) base drag [1]. The base drag is a major contributor to the total drag, particularly at transonic speeds. Thus, the determination and minimization of base drag is essential in minimizing the total drag of projectiles. The breakdown of the total drag into various components is important in the preliminary design stage of a shell. This Information can aid the designer to find potential areas for drag reduction and achieve a desired increase in range and/or terminal velocity of projectiles since they are affected by the projectile drag.

2. FLOW CHART OF DRAG REDUCTION

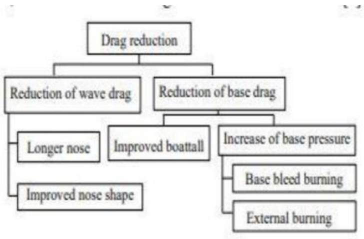

Figure 1: Various drag

2.1. COMPUTATIONAL APPROACH

The (2-D) axis symmetric body projectiles configurations are considered due to the physical complexity of the entire process. The base flow field is described with the Reynolds Averaged Navier-Stokes (RANS) computational fluid dynamics software (CFD). Using three turbulence models k- SST (The shear stress transport) (2 equations) transition k-kl- (3 equations) and Reynolds Stress Model RSM (5 equations), are used. Those models were tested and compared with the semi-empirical model (ADK0) using aerodynamic prediction based on theories. In addition, those results were compared with experimental results obtained by 3D radar in the case of standard projectile.

2.2. Hypothesis and boundary conditions

The simulations, they have performed considering steady state boundary conditions for different flight conditions. The atmospheric conditions, considered as stagnation conditions. This way, different flight conditions cases were simulated to obtain the body drag coefficient at different Mach numbers, and different mass flux injection (case of projectile with burning). The fluid considered in the simulations was air and propellant combustion gases. For both gases, the ideal gas assumption was used. The constant values were assumed for heat capacities. The Sutherland law for variable dynamic viscosity was used due to the high temperature ranges encountered in the problem studied. The projectiles were assumed to fly under zero angle of attack. All the walls were considered adiabatic. The flow field was considered compressible and the far field conditions were imposed at the external boundary, where the flight Mach number, pressure and temperature (stagnation values) they were introduced. The entire domain was initialized with these far field conditions. The mesh number of the grid is almost 300000 cells (600 cells were used on the projectile, a 100 of them were used at the base). Those grid density was determined after testedseveral grid density types cells. Regarding the propellant combustion, the process was modelled as an injecting of gas mass flow rate at a fixed temperature through the orifice. The temperature and mass flow rate values were obtained from the propellant combustion data. The relative chamber pressure was estimated from the static experimental combustion tests (we used an average value of 6000 Pa). The thermodynamic parameters and the composition of combustion products are introduced. They were obtained with the help of the thermo chemical calculation (TERMO code), which is developed based on with an elevation angle equal to 0 = 14,22°.

3. REVIEW ON EXPERIMENTAL SET UP

3.1. Tunnels and visualization technique.

The tests were conducted in a vertical water tunnel running under the influence of gravity at an upstream velocity of U1 = 7 cm/s corresponding to a Reynolds number Re1 = U1*H1/V1 = 103. The test section is 140. 140mm2 Schlieren photographs were recorded along two orthogonal directions normal to the upstream velocity. The tests in subsonic compressible flow were carried out in a wind tunnel whose test section was 200 mm high and 150mm wide. The Mach number was M2=0.66 and the U2 H2 /V2 = 3.2 * 105 Reynolds number Flow visualizations were performed with a CranzSchardin multiple sparks system. This technique of 24 sparks for ultra-high-speed visualization can be used for shadowgraph or schlieren observations. Windows mounted in sidewalls, floor and ceiling allowed simultaneous visualizations along two orthogonal directions normal to the upstream velocity.

Figure 2: wind tunnel test

3.2. MODELS

As the aim of the investigation in the water tunnel is to obtain some insight into the unsteadiness and the three-dimensional character of the base flow, the model must be as large as possible in order to improve the quality of the visualizations. It is essentially a thick, sharp-nosed flat plate spanning the tunnel, to which is attached a segmented trailing edge similar to that studied by Pollock. But only one space period is present along the span (Fig. 1).

The rear of the body may be characterized by a central simple blunt base inserted between two downstream extending blocks including spanwise cavities. The insulated model is equipped with flush mounted thin copper plates located close to each sharp edge of the base on either side. Each plate can be heated by an electrical resistance, fed by some kind of external energy regulator. The wind tunnel model was geometrically similar to the former, but with a lower aspect ratio. It was 18 mm thick and was equipped with 37 pressure taps located at the base and near the separation lines. Three transducers located at the rear of the body measured the unsteady pressure acting on the different sides of the model. The downstream extending blocks wereinterchangeable, so that a two dimension.

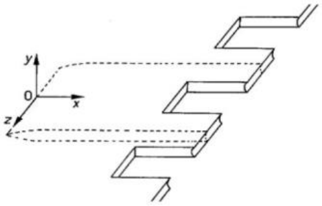

Figure 3: Determination of the model span for one period of a segmented base blunt base configuration could be examined

4. Perspective of flight drag reduction

From the beginning of manned flight, the iteration of lift, weight, drag, and thrust have been the balancing factors involved in so-called aeronautical engineering. Lift greater than weight is needed to get up, thrust greater than drag is needed to go anywhere. These fundamentals are still as true as ever. The list of variables involved in successful aeronautical engineering has grown significantly to include speed, cost, comfort, aesthetics, pollution, noise, etc., with perhaps the most significant currentinterest in fuel economy. There will surely be other tradeoffs to be faced, but never will we be able to ignore lift, weight, drag, and thrust. In this conference, we wi” deliberately focus attention on drag reduction. Drag is the basic parameter affecting the ability of aircraft to go somewhere efficiently. A hot gas balloon can get up and stay up reasonably well, with essentially no consideration for drag. It goes when the wind blows, at no more than the speed of wind. But as soon as you decide to makeit go faster than the wind, or in another direction, its drag becomes very important. In the earlydays, airplanes were a lot like the free balloon– getting up and staying up was difficult enough without worrying much about going somewhere efficiently. The structures guys were hard pressed to make

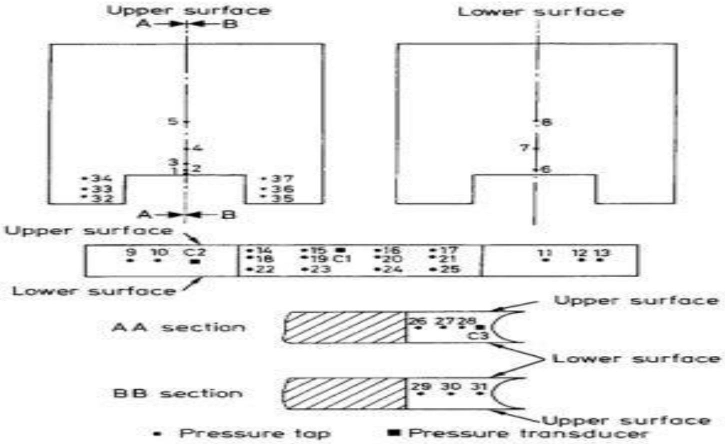

Fig. 2. Sketch of the model used in wind tunnel

5. Profile Drag

The resistance of an object moving through air is pretty clearly a function of the cross-sectional area, the wetted area, the shape of the object, and the friction caused by the scrubbing of the air over the object. Here again, the wing, although streamlined in appearance, contributes 20-40 percent of the total airplane parasite drag.

6. Propulsion Drag

Another form of drag many of us have gotten used to is that associated with internal flows around engines and accessories. To be sure, the matter is an inter-disciplinary problem involving interfaces with the engine, propeller, and airframe. Until the jet engine came along and caused more aerodynamicists to concern them-selves with internal flows, the matter of engine-nacelle drag was largely an empirical or experimental matter. The NACA’s experimental work on cowlings in the 20’s and 30’s provided data for use with radial and in-line engines used extensively during the 1940’s

7. REVIEW OF AUTOMOBILE BASE DRAG

In automobile drag reduction the mostly needed for the ground transport vehicle this will increase the economical pressure increasing which leads to the fuel efficiency, to avoid this problem we need to emphasize its multidisciplinary nature through the use of integrated product teams. These commercial trends, together with the immense volume of design, manufacturing and maintenance data inherent to complex modern equipment’s, demand for a heavily computerized environment. Multidisciplinary Design and Optimizations (MDO) envisions a parametric description format of input data, which will generate, for a specific set of values of the parameters, a new vehicle description that in turn is used to generate input for Computer Aided Engineering, including Computational Fluid Dynamics (CFD). The aerodynamic design plays a crucial role in the development phase of new automotive configurations and, due to its intrinsic complexity, the designer needs as much aids as possible to strengthen his/her choices and discard unsuitable solutions. In this context, the possibility of evaluating performances of different configurations is of utmost importance; however, difficulties arise due to the high number of geometrical parameters involved, which are necessary for defining each configuration. A systematic analysis taking into account the effects of all these parameters is very difficult, given the complexity related to both aerodynamic load evaluation and the assessment of mechanics, stylist, commercial and others requirements. A direct numerical optimization technique may be satisfactorily employed to find one’s way through this complex survey. Indeed, with this kind of aid the designer has a great flexibility in the choice of the design variables and the problem may be addressed systematically. The activity on the study of optimization has significantly increased over the last years, driven by advances in computational methods and improvement in computer performances, and this aspect has been particularly significant in the aerodynamic design, especially for land and flying vehicles. Several examples of this trend can be found, for instance, in. From the analysis of the existing bibliography, two different aspects can be highlighted. The first aspect is the need to improve the accuracy and the validity range of the results, to obtain a realistic representation of the aerodynamic flow; this implies using a sophisticated flow solver within the optimization procedures. The second aspect is the requirement to obtain the results in short time. These two aspects have the common targets to achieve an improvement in the configuration performances and to reduce the “time to market”. Clearly, they act in opposite directions, because high accuracy flow solutions imply long computing time. As a consequence, an important research activity on the optimization techniques are in progress, especially in the aeronautical field for a review.

7.1. AN EXAMPLE OF APPLICATION TO AUTOMOTIVE PROBLEMS

A preliminary application of the optimization approach was developed. The project, named HIPEROAD (High Performance Road vehicle Optimized Aerodynamic Design), had the object to make the optimization loop in large part automatic, in order to reduce the time needed to evaluate and improve the sketch design provided by the stylists. The complete procedure is described in the pa- per. the most significant aspects of the procedure are presented, and the capability of the methodology is shown by means of a specific application.

DESCRIPTION OF THE METHODOLOGY

In the analysis through direct numerical optimization, an aerodynamic code is coupled with an optimization routine, giving rise to an iterative procedure which is able to automatically manage the values of the design variables – typically concerning geometry modifications – by minimizing a given scalar quantity (the objective function). This approach is extremely flexible, and capable of meeting multidisciplinary requirements. In flow chart of the optimization loop is shown; the main components will be shortly described in the following sections. EXAMPLES OF APPLICATION - The described procedure of optimization was used in the development of the Ferrari 360 Modena. The basic geometry analyzed is shown in the above figure. As a first example, a run carried out with the vertical load as cost function is considered. The car could be modified on the upper surface, with a centered longitudinal control line and one control point, placed on the car side, at a position corresponding to the rear axle: 5 degrees of freedom were then used. A maximum number of 100 iterations were considered, and, in a first run, no constraints were considered. The results show a significant increase in the

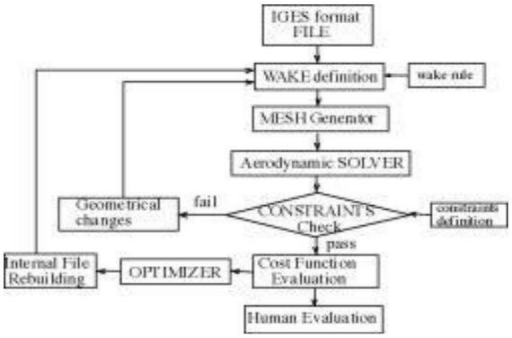

Figure 5: The optimization loops

Figure 6: The basic geometry for the analysis of the 360 Modena

download: at a speed of 280 Km/h, after 100 iterations there is an increase in the download of about 1200 N. Unfortunately, this solution cannot be considered a suitable one: indeed, the modification to the geometry, which is not constrained, is found to be not acceptable. As could be foreseen, it is evident that, to have a suitable solution, the constraints (in particular the geometric one) must be activated. A calculation on the same case, with a geometric constraint defined by a maximum displacement of the car surface of 5 cm, was then carried out. The final shape (after 100 iterations) showed a very small modification in the car shape, and a final shape that is acceptable from the esthetic point of view. The results in terms of download, for both the described analyses, are shown in Figure is possible to note that up to iteration 40 the behavior is not affected by the constraints. After this iteration the geometric constraint is violated, and the increases in download became less important, being smaller the modifications allowed to the geometry. In any case we can observe an increase in the download, after 100 iterations, of about 1000 N at 280 Km/h.

8. CONCLUSION

By the review of all the journal article we concluded that there are different methods of base drag reduction, those articles are particularly worked on one method to know the drag from and how to overcome them. They conducted on the automobile, aircrafts, launch vehicle by Nasa and airfoil which are conducted and experimented in CFD analysis, an investigation was conducted in the Langley l6-foot transonic tunnel to determine the effects on base drag of recessing the base of an axisymmetric body. The conclusions reached from that investigation are as follows: Over the range of Mach numbers from 0.3 to 1.3, and for boattail angles of 0 to 10^0, recessing the base of an axisymmetric body with a truncated conical boattail causes an increase in base pressure and hence a decrease in base drag. The maximum increase in base pressure coefficient derived from recessing the base varied from 0.01 to 0.03 depending on the boattail. In general, for a given boattail angle and length, base pressure increases with increasing base concavity up to a particular amount of concavity, but, beyond this amount, further concaving the base has little or no effect. In the test range of boattail angles 0 to 10^0, the steeper boattail angles generally require a greater ratio of base concavity to base radius to derive the maximum base pressure benefits from recessing the base than do the shallower boattail argils. For boattail angles of 0 to 10^0, recessing the base has practically no effect on boattail pressures. The base drags of projectiles and of fuselages with blunt or cut-off trailing end are a distinct function of the forebody drag From the drag characteristics of sheet-metal joints it is possible to calculate the drag originating from the blunt or cut-off trailing edge of airfoil sections They have used Fluent software to estimate the best base configuration that would minimize the total drag of a 155mm artillery shell.In addition to the basic configuration, six modified configurations were studied These are: boattail only, base blade only, base cavity only, boattail with base bleed, boattail with base cavity and a combination of the three. They have found that having a boattail with angle 9.5 degrees minimizesthe drag coefficient by about 50 by about 12While, by using a boattail with angle 15 degrees it was possible to minimize the total drag by 55 degrees in transonic and supersonic regimes.

References:

1. ROSHKO, A., On the drag and shedding frequency of two-dimensional bluff bodies. NACA TN 3169 (1954).

2. ROSHKO, A., On the wake and drag of bluff bodies. J. Aero. Sci. 22, 124-132 (1955).

3. HOERNER, S. F., Fluid–Dynamic Drag. Published by the author (1965).

4. TANNER, M., Totwasserbeeinflussung bei Keilstr6- mungen. Deutsche Luft- und Raumfahrt Forschungsbericht 64-39 (1964).

5. TANNER, M., The wake pressure behind wedges as influenced by splitter plates and suction. AGARD CP-4, pp. 887-909 (1966).

6. NASH, J. F., QUINCEY, V. G. and CALL1NAN, J., Experiments on two-dimensional base flow at subsonic and transonic speeds. Aeronautical Research Council R. and M. 3427 (1963).

7. BEARMAN, P. W., Investigation of the flow behind a two-dimensional model with a blunt trailing edge and fitted with splitter plates. J. Fluid Mech. 21,241-255 (1965).

8. POLLOCK, N., Some effects of base geometry on two-dimensional base drag at subsonic and transonic speeds. Note ARL/Aero. 316, Aeron. Res. Laboratories (Australia) (1969).

9. TANNER, M., Dreikomponenten- und Basisdruckmessungen an einem Rechteckfliigel mit stumpfer Hinterkante mit verschiedenen Trennplatten im Totwasser. Deutsche Luft- und Raumfahrt Forschungsbericht 73-61 (1973).

10. TANNER, M., New investigations for reducing the base drag of wings with a blunt trailing edge. AGARD CP-124 pp. 12-1 to 12-9 (1973).

11. TANNER, M., EinfluB von Trennkeilen auf den Auftrieb und Widerstand eines Rechteckfli.igels mit stumpfer Hinterkante. Deutsche Luft- und Raumfahrt Forschungsbericht 74-61 (1974).

12. TANI, I., Experimental investigation of flow separation over a step. Grenzschichtforschung IU-TAMSymposium. Freiburg/Br. 1957, pp. 377-386, Springer, 1958.

13. TANNER, M., Auftriebs- und Widerstandsmessungen an einem Rechteckfliigel mit Trennkeil im Totwasser im Machzahlbereich von Ma = 0,5 bis 1,2 Z. Flugwiss. 22, 416-426 (1974).

14. BEARMAN, P. W., Investigation into the effect of base bleed on the flow behind a two-dimensional model with a blunt trailing edge. AGARD CP-4, Part 2, pp. 485-507 (1966).

15. BEARMAN, P. W., The effect of base bleed on the flow behind a two-dimensional model with a blunt trailing edge. Aero. Quart. 18, 207-214 (1967).

16. WOOD, C. J., The effect of base bleed on a periodic wake. J. Roy. Aero. Soc. 68, 477–482 (1964).