Detailed Table of Contents

Section 2
Incompressible Flow Aerodynamics

Here, the authors explain the Magnus effect. The ball is deflected in the same direction as the rotation. The most common exposure and welcome statement of the Magnus effect is that a spinning object creates a vortex of fluid swirling around it. On the side where the movement of the vortex is in the same direction as the direction of the flow to which the object is exposed, the speed will increase. On the opposite side, where the directions are opposite, the speed will decrease. It is explained here, according to Bernoulli's principle, that the pressure is lower on the side with the greatest velocity, and therefore, there is an unbalanced force orthogonal to the flow of the fluid.

One of the most important design impacts in aircraft is an airfoil. This airfoil is also considered the cross-section of the wing, so the flow characteristics of air depend on the airfoil's shape. There are some airfoil theories we can consider for the design consideration while designing the wing. The airfoil theories describe the nature of fluid flow over the wing based on the angle of attack for variable speed conditions. This chapter deals with airfoil terminology and the theories of airfoil like the methodology of conformal transformation, Cauchy-Riemann relations, complex potential, Kutta-Joukowski, thin airfoil theory, and their applications. The theories mentioned above explain transformation and its applications, Kutta condition, Kelvin's circulation theorem, starting vortex creation of airfoil for variable speed considered for aircraft, and the advantages of the optimization.

Wing design is a very complicated and intricate issue. It is not feasible to cover everything in this chapter; however, it is possible to discuss some of the essential ideas that underpin design for high lift and low drag. Lift may be increased in four ways for fixed air characteristics and free-stream speed: increased wing area, increased angle of attack, increased camber, increased circulation through the use of high-momentum fluid. One of the most important applications of potential flow theory was the study of lifting surfaces such as aircraft wings, since the boundary conditions on a complex geometry can significantly complicate any attempt to tackle the problem via analytical techniques, which involves some simplification assumptions in order to arrive at a solution. These assumptions will be related to the concept of three-dimensional thin wing issues in this chapter.

Viscosity is a property that expresses the internal drag of a fluid to motion; impact of viscosity states the statics and flows. Statics means whenever fluids at zero velocity have no relative movements between layers of fluid and thus du/dy = 0. At the time there is no shear stress and viscosity of the fluid is free. Fluid viscosity plays a major role on the fluid floating in it. The authors focused on solids and fluids and the no slip condition, momentum transfer through molecular motion, shear stress and viscosity, Couette flow, and Poiseuille flow. Here the authors made a discussion the Newtonian viscous flow, and the statement of Newton's law of viscosity was examined. The discussion has been extended up to viscosity and the effect of their temperature and impact of increasing in temperature has been explained along with surface tension.

This chapter describes the basic concepts of aerodynamics, evolution of lift and drag, types of drag, reduction of wing tip vortices, non-planar wing concepts for increased aerodynamic efficiency, various methods for determination of aerodynamic forces of an airplane, classification of wind tunnels, blower balance tunnels, and a case study report on aerodynamic force measurement of the non-planar wing systems. To increase the aerodynamic efficiency of the monoplane configuration, the 'C-wing' configuration is presented in this chapter. The aim is to prove, at all angles of attack, C-wing produces a higher (L/D) ratio than straight wing for the same wetted surface area. The aerodynamic characteristics of three different wing models with NACA-64215 aerofoil such as straight wing, C-wing, and inverted C-wing at different angles of attack and low Reynolds number are shown. The inverted C-wing created more lift but produced more vibration, which may lead to lesser structural integrity.

hypersonic flight vehicle, the characteristics of the hypersonic atmosphere should be understood clearly. This comes as a major design constraint. A brief overview of the hypersonic vehicle design and high-speed flow characteristics will be presented in this chapter. A few characteristics of hypersonic flow and the ideal design conditions will also be reviewed.

Humans longing to fly higher and quicker have prompted the improvement of hypersonic vehicles. Typically, hypersonic streams are described by high temperature fields and a thin layer of shock close to the object wall or the body surface. To ease the reduction of thermal loads, a blunt nose is forced in a hypersonic vehicle which is more imperative. In any case, increase in the wave drag is one of the quick outcomes of a constrained bluntness. Consequently, investigation in the hypersonic field is constantly fixated on the wave drag decrease. The flow features around the blunt body get changed because of the attachment of spike in front of the vehicle. This chapter aims to give a detailed review of a hypersonic vehicle that involves an aerospike design in front of the blunt body, which tends to reduce the drag at the forebody. Views of various researchers are investigated, and efforts are taken to summarize the reported results on how the drag has been reduced using aerospike technique.

Nature helps a lot to create and recreate new concepts to enhance our existence. Using nature-inspired means to address new engineering problems gives a better solution that is very quick, easy, and environmentally friendly. The goal of this chapter is to explore the overall performance increase by bioinspired design elements. Simulations of computational fluid dynamics are utilized to compare existing designs to experimental data in various flow regimes.

Flow separation and wind gusts affect the aerodynamic performance of low-Reynolds number flyers with a chord-based Reynolds number of 105 or below. Active flow control provides information into fluid dynamics as well as potential vehicle performance enhancements. Aircraft engines, as well as stationary flow devices, exhibit undesirable flow states that have a significant impact on noise output and aerodynamic loss. Vibrations and aerodynamic performance are likely to suffer as a result of flow and boundary layer separations. Flow management can be an effective approach for lowering noise levels, increasing efficiency and thereby lowering fuel usage. Various forms of flow can be controlled using both passive and active flow control strategies. The active alteration of aerodynamic flows utilizing small time-dependent actuators in well-chosen places is defined as adaptive flow control. Micro-adaptive flow control has a significant overall system benefit in a variety of large-scale applications, which are presented.

The chapter addresses an extensive numerical investigation for the optimization of static pressure recovery in the dump diffuser of annular combustor for modern aero-gas turbine engines. The modelling and simulation processes are accomplished through the commercial ANSYS fluent module with SST k-epsilon (k-ε) turbulence model with the atmospheric conditions prevailing at 9 km altitude. The effect of velocity changes on the flow recirculation and vortex mixing phenomena is also addressed with novel dome-shaped optimization at the velocity of 25 m/s. The dome geometry can also be considered as a flame tube head, and the influence of vortex pattern and pressure distributions caused by the different dome shapes are analyzed using CFD. The dome-shaped optimization strategy addressed herein could initiate more potential studies on dump diffusers with hot and cold flow conditions to enhance the overall combustion efficiency in the near future.

Chapter 13
Study of Drag Reduction on a Hypersonic Vehicle Using Aerospike

Naren Shankar R.
Vel Tech Rangarajan Dr. Sagunthala R&D Institute of Science and Technology, India

Irish Angelin S.
Vel Tech Rangarajan Dr. Sagunthala R&D Institute of Science and Technology, India

Vitalii Pertsevyi
Dnipro National University of Railway Transport, Ukraine

ABSTRACT

Humans longing to fly higher and quicker have prompted the improvement of hypersonic vehicles. Typically, hypersonic streams are described by high temperature fields and a thin layer of shock close to the object wall or the body surface. To ease the reduction of thermal loads, a blunt nose is forced in a hypersonic vehicle which is more imperative. In any case, increase in the wave drag is one of the quick outcomes of a constrained bluntness. Consequently, investigation in the hypersonic field is constantly fixated on the wave drag decrease. The flow features around the blunt body get changed because of the attachment of spike in front of the vehicle. This chapter aims to give a detailed review of a hypersonic vehicle that involves an aerospike design in front of the blunt body, which tends to reduce the drag at the forebody. Views of various researchers are investigated, and efforts are taken to summarize the reported results on how the drag has been reduced using aerospike technique.

INTRODUCTION

The speed of an aircraft is usually denoted in Mach Number, M. Based on the speed regime the aircraft can be classified into different types (i) 0 < M < 0.8 subsonic, (ii) 0.8 < M< 1.2 transonic, (iii) 1.2 < M < 4 supersonic, (iv) M ^{3} 5 hypersonic. If M = 1 it is called as sonic condition. In hypersonic flows, the

DOI: 10.4018/978-1-6684-4230-2.ch013

flow velocity is much greater than the sound velocity, which is the velocity of propagation of small disturbances. Hypersonic vehicles in general fly at higher altitudes where the density and Reynolds numbers are considered to be low and where the boundary layer formation is found to be thick. Reducing the drag and aerodynamic heating are the two main factors in critically designing a hypersonic vehicle. Reducing the drag in the hypersonic vehicle is necessary as it helps in saving the fuel, extending the range, simplifying the propulsion unit and increasing the payload to take-off the gross weight ratio. A hypersonic vehicle creates a bow shock in front of the blunt body which leads to a higher surface pressure load and aerodynamic drag. This surface pressure can be reduced if the conical shock wave is created rather than a bow shock wave. This can be achieved by using a spike in front of the blunt body. This chapter aims to review various papers related on drag reduction techniques by using aerospike passive method. The objects moving at high speed experience forces that will in general tend to slow objects down. This confrontation withinside the considered volume is named such as a drag that is a solitary significant concern throughout the time of planning in designing a high speed vehicle. Hypersonic vehicles are those that travel at Mach above 5. Usually, missiles, rockets, reentry vehicles travel at hypersonic speed. It is often planned to reduce surface heat flux by designing hypersonic space vehicles with blunt noses. By any means, this detached shock increases the wave drag, making space flights more expensive. Thus, the spaceship configuration should consider the best balance between wave drag and surface heat flux. Subsequently, in the future, there will be more development and improvement of the hypersonic drag reduction technique as it becomes a significant research area within the hypersonic field. A number of strategies have been put forth in this area.

The streamlined aerodynamic layout of hypersonic vehicles must be completed considering the intricacies associated with the hypersonic flow. Typically, hypersonic flows are described by high temperature field and a thin layer of shock close to the object wall or the body surface. Such shock and viscous dissemination came about raised temperature close to the space vehicle surface which led to inordinate warming of vehicles flying at speeds above Mach 5, which are considered as hypersonic speed regimes.

The nose area of a space vehicle which is considered as the most serious place of space vehicle that undergoes most noteworthy heat flux, blunt nosed plan of the equivalent design has been recognized as a powerful strategy to drive the shock layer away from the body, to lessen the surface heat flux. (John D Anderson, 1989). In the meantime, the aerodynamic heating appearing in the spot of stagnation changes conversely which is equal to the blunt nose radius root, possessing a huge blunt nose area is worthwhile all things considered. Besides this design measure improves the drag reduction. For space missions increase in drag makes higher fuel utilization and lesser payload limit. In this manner, the expense of space missions increments radically. Along these lines, the decrease of wave drag has stayed as a significant field of exploration in the field of hypersonics. Wave drag is triggered by the development of shock waves around the vehicle in supersonic flight and hypersonic flight. Also, to reduce fuel usage and to expand the payload limit of the space vehicle, decreased heat load or lower drag is required for a hypersonic flight or mission.

To ease the reduction of thermal loads, a blunt nose is forced in a hypersonic vehicle which is more imperative. In any case, improving the wave drag is a quick outcome of constrained blunt body. Consequently, investigation in hypersonic field is constantly fixated on the wave drag reduction. In order for hypersonic vehicles to succeed, reducing drag is a vital component. Extreme aerodynamic heating and aerodynamic forces are imposed at the blunt-nosed surface, owed to the drastic bow shock to be found at the forefront of the nose when a vehicle travels at hypersonic speed.