Proceedings of the 3rd IASME/WSEAS Int. Conf. on FLUID DYNAMICS & AERODYNAMICS, Corfu, Greece, August 20-22, 2005 (pp80-83)

Supersonic Flow over Blunt Body with a Decelerator

KHALED ALHUSSAN*

* Research Assistant Professor, Space Research Institute, King Abdulaziz City for Science and Technology, P.O. Box 6086 Riyadh 11442, KSA, alhussan@kacst.edu.sa

Abstract: The work to be presented herein is a Computational Fluid Dynamics investigation of the complex fluid mechanisms that occur over blunt body with and without a decelerator, specifically with regard to the total aerodynamic drag. Drag is needed to decelerate the body. The aim of this research is to design deceleration devices for a blunt body. In this paper a qualitative analysis of the flow structure over a blunt body and blunt body with a decelerator was shown.

The results will show, that the adding a deceleration device will change the flow structure behind the body especially with regard to the pressure drag and wake. Results of contour plots of Mach number for different angles of attack and different speeds will demonstrate that the aerodynamic forces and the velocity are changed when the deceleration device is integrated with the blunt body.

Key- Words: Detached Shock Waves, Decelerator, Compressible flow, Aerodynamic Drag, CFD.

1 Introduction

The work to be presented herein is a Computational Fluid Dynamics investigation of the complex fluid mechanisms that occur over blunt body with and without a decelerator, specifically with regard to the total aerodynamic drag. Drag is needed to decelerate the body. The aim of this research is to design deceleration devices for a blunt body. In this paper a qualitative analysis of the flow structure over a blunt body and blunt body with a decelerator was shown.

A drag over moving body consists of two components: pressure drag and friction drag. Drag is due to the effect of viscosity. Pressure drag is a result of the eddying motions that are generated in the fluid due to the movement of the body. Pressure drag is related to the cross-sectional area of the body and it is associated with the formation of a wake it is also important for separated flows. Frictional drag is a result of the friction between the fluid and the surfaces over which it is flowing. Frictional drag is related to the surface area exposed to the flow and it is associated with the development of boundary layers it is also important for attached flows [1-2].

For a blunt body, pressure drag is the dominant source of drag, but for streamlined body friction drag is the dominant source of air resistance. In some applications of aerodynamics, a deceleration of a moving body is required therefore the prediction and controlling of the drag is essential [3]. The deceleration devices such as air bag or fins can be added to the body to increase the aerodynamics drag. For a supersonic speed, a flow around blunt body is complicated due to the detached shock wave, flow separation, boundary layer and their interactions. When a decelerator is integrated with the blunt body the flow is subject to sever change of aerodynamic forces and velocity [4-5].

A number of important conclusions follow from the current research. First, study of the actual flow configuration over a blunt body with a decelerator offers some insight into the complex flow phenomena. Second, adding the decelerator will increase the separation that will result in an increase of total drag.

2 Computational Fluid Dynamics Analysis

The governing equations are a set of coupled nonlinear, partial differential equations. In order to formulate or approximate a valid solution for these equations they must be solved using computational fluid dynamics techniques. To solve the equations numerically they must be discretized. That is, the continuous control volume equations must be applied to each discrete control volume that is formed by the computational grid. The integral equations are replaced with a set of linear algebraic equations solved at a discrete set of points.

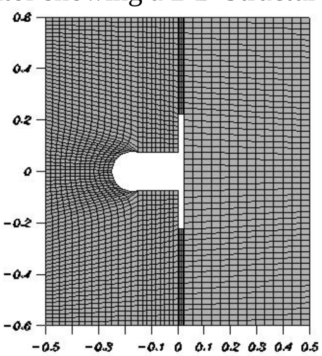
CFD-FASTRAN is used in the current research to model the flow over the blunt body with a decelerator. The CFD code is an integrated software system capable of solving diverse and complex multidimensional fluid flow problems. The fluid flow solver provides solutions for incompressible or compressible, steady-state or transient, laminar or turbulent flow in complex geometries. The code uses block-structured, non-orthogonal grids to discretize the domain.

It should be possible to model the detached shock waves, flow separation, the interaction of the shock waves, and expansion fans around the blunt body with the decelerator using the CFD analysis [6-9] .

A numerical analysis must start with breaking the computational domain into discrete sub-domains, which is the grid generation process. A grid must be provided in terms of the spatial coordinates of grid nodes distributed throughout the computational domain. At each node in the domain, the numerical analysis will determine values for all dependent variables such as pressure and velocity components.

Creating the grid is the first step in calculating a flow. Two-dimensional Navier Stokes equations are solved using fully implicit scheme with K-epsilon turbulence model. The grid is refined near the surfaces and in front of the body in order to model the large gradient.

Fig.1 Schematic view of a blunt body with a decelerator showing a 2-D Structured Mesh



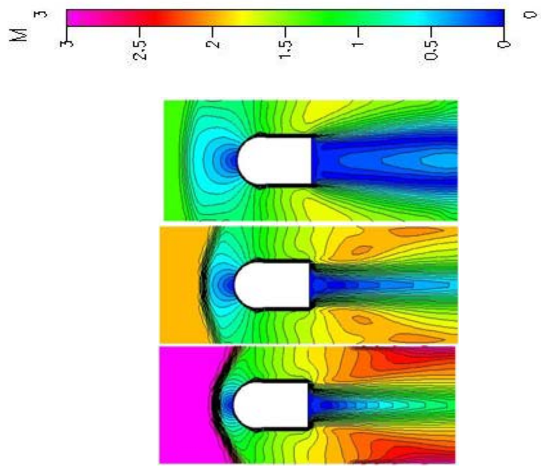
A computational model that illustrates the physics of flow over a blunt body with a decelerator was developed. Through this computational analysis, a better interpretation of this physical phenomenon can be achieved. The results from the numerical analysis will be used to develop a design methodology so as to predict optimal performance.

3 Results and Discussion

Figure 1 shows a two dimensional structured mesh. Figure 1 also shows a schematic view of the blunt body with a decelerator. The analysis was carried out based on the flow over the blunt body with out a decelerator for a range of Mach number from 1.4 to 3.0 and also for different angles of attack starting from zero to 15 degrees.

Figure 2 shows a contour plot of Mach number for flow over a blunt body without a decelerator at zero angle of attack. The Mach number is changed from 1.4 to 3.0. One can see the detached shock waves in figure 2. One can also notice from this figure that the shape and the strength of the detached shock waves and flow separation region changing with increasing Mach number.

Fig.2 Contour plot of Mach number, flow over blunt body without decelerator, Zero angle of attack



Contour plot of Mach number for flow over a blunt body without a decelerator at 15-degree angle of attack is shown in figure 3. The Mach number is changed from 1.4 to 3.0. One can see the detached shock waves in figure 3. One can also notice from this figure that the shape and the strength of the detached shock waves and flow separation region changing with increasing Mach number.

Figure 4 shows a contour plot of Mach number for flow over a blunt body with a decelerator at zero angle of attack. The Mach number is changed from 1.4 to 3.0. One can see the detached shock waves in figure 2. One can also notice from this figure that the shape and the strength of the detached shock waves and flow separation region changing with increasing Mach number. It is observed, from figure 4 that adding a decelerator can lead to significant increase in flow separation behind the body.

Contour plot of Mach number for flow over a blunt body with a decelerator at 15-degree angle of attack is shown in figure 5. The Mach number is changed from 1.4 to 3.0. One can see the detached shock waves in figure 5. One can also notice from this figure that the shape and the strength of the detached shock waves and flow separation region changing with increasing Mach number. It is observed, from figure 5 that adding a decelerator can lead to significant increase in flow separation behind the body.

Fig.3 Contour plot of Mach number, flow over blunt body without decelerator, 15-degree angle of attack

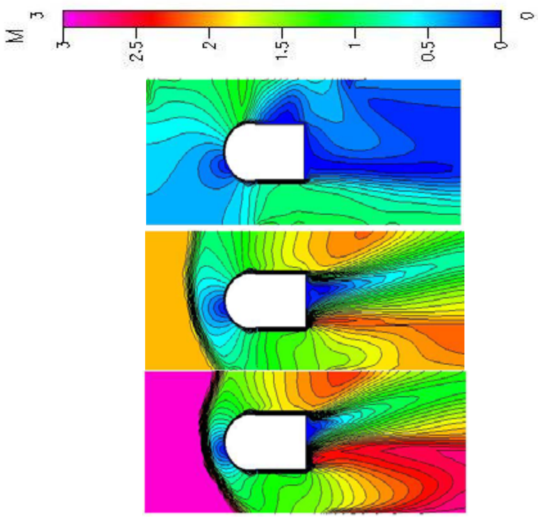


Fig.4 Contour plot of Mach number, flow over blunt body with decelerator, Zero angle of attack

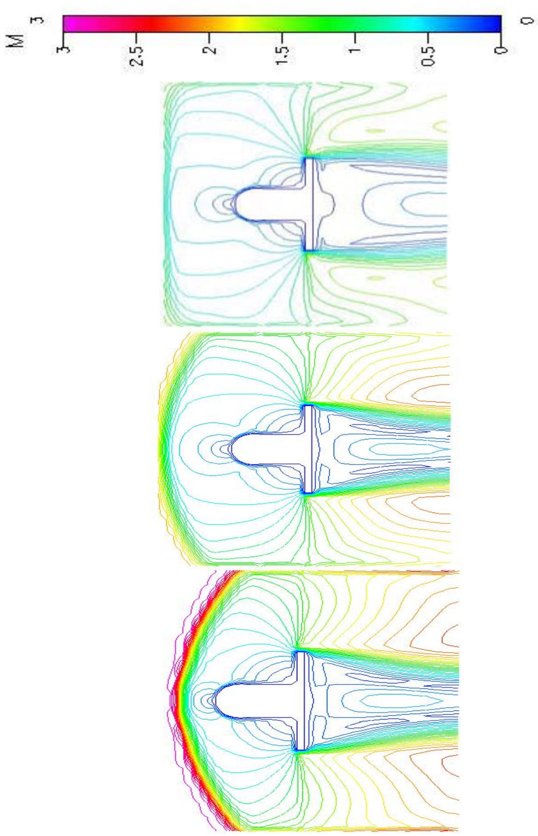
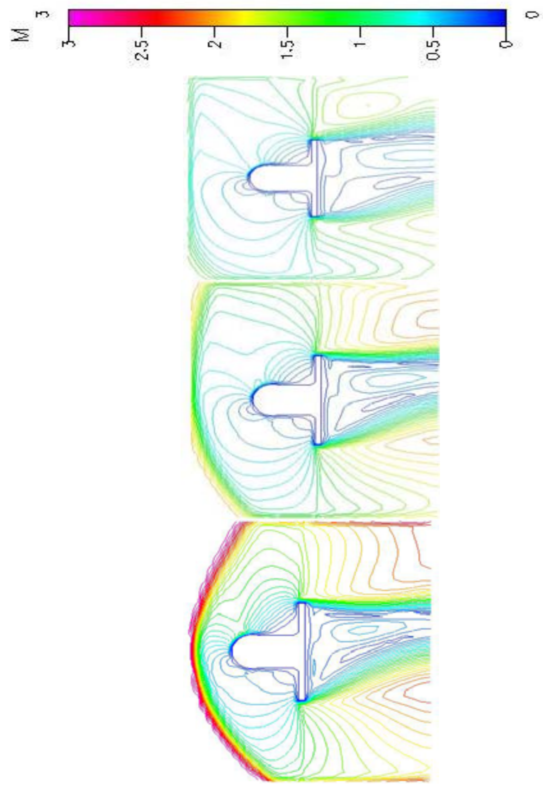


Fig.3 Contour plot of Mach number, flow over blunt body with decelerator, 15-degree angle of attack



4 Conclusion

In this paper, a numerical analysis was conducted to study the flow structure over a blunt body with and without decelerator. A qualitative analysis of the flow structure over a blunt body and blunt body with a decelerator was shown.

The results will show, that the adding a deceleration device will change the flow structure behind the body especially with regard to the pressure drag and wake. Results of contour plots of Mach number for different angles of attack and different speeds will demonstrate that the aerodynamic force and the velocity are changed when the deceleration device is integrated with the blunt body.

5 Acknowledge

The author acknowledges support of this research from King Abdul Aziz City for Science and Technology, Kingdom of Saudi Arabia.

References:

[1] Sahu, J. & Nietubicz, C.”Navier-Stockes Computations of Projectile Base Flow with

and without Mass Injection: AIAA Journal, Vol. 23, No. 9, pp.1348-1355, 1985.

[2] Bourdon, C. & Dutton, J. " Visualization of a Central Bleed Jet in an Axisymmetric Compressible Flow" Physics of Fluids, Vol. 15, No. 2 pp.499-510, 2003.

[3] Perkins, S. C.& Dillenius, M. " Supersonic Submunitions Aerodynamics during Dispense" Journal of Spacecraft and Rockets, Vol. 28, No.3 pp.276-283, 1991

[4] Lee, Y., Kim, H. & Raghunathan, S. "A Study of Base Drag Optimization using Mass Bleed" 15th Australian Fluid Mechanics Conference, Sydney, Australia, 2004.

[5] Lee, Y., Chang, S.Lee, J. & Kim, H.: A Study of the Aerodynamic Drag of a Blunt Body with an Air Bag" the 6th KSME-JSME Thermal and Fluids Engineering Conference, Jeju City, Korea, 2005.

[6] Alhussan, K., " Computational Analysis of High Speed Flow over a Double-Wedge for Air as Working Fluid", Proceedings of FEDSM2005 ASME Fluids Engineering Division Summer Meeting and Exhibition FEDSM2005-77441 June 19-23, 2005, Houston, TX, USA.

[7] Alhussan, K. ," Study the Structure of Three Dimensional Oblique Shock Waves over conical rotor-Vane surfaces", Proceedings of FEDSM2005 ASME Fluids Engineering Division Summer Meeting and Exhibition FEDSM2005-77440 June 19-23, 2005, Houston, TX, USA.

[8] Alhussan, K. ," Oblique Shock Waves Interaction in a Non-Steady Three Dimensional Rotating Flow", Proceedings of FEDSM2005 ASME Fluids Engineering Division Summer Meeting and Exhibition FEDSM2005-77442 June 19-23, 2005, Houston, TX, USA.

[9] Alhussan, K. "Application of Computational Fluid Dynamics in Discontinuous Unsteady Flow with Large Amplitude Changes; The shock Tube Problem" IASME Transaction Issue 1 Volume 2, pp 98-104, January 2005.