ANALYSIS OF FLOW VARIATION OVER ELLIPTICAL NOSE CONE AT DIFFERENT ANGLE OF ATTACK

Ankit Kumar Mishra, Amandeep Singh

Department of Aeropsace Enginnering, Lovely Professional University
Phagwara, India

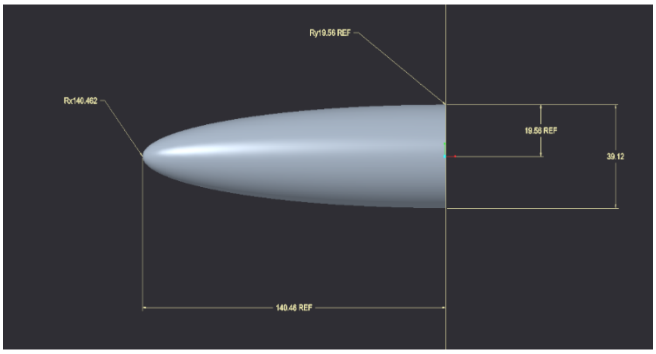
ABSTRACT

The object moving at high speeds encounter forces which tend to decelerate the objects. This resistance in the medium is termed as drag which is one of the major concerns while designing high speed aircrafts. Another key factor which influences the design is the air drag. The main challenges faced by aerospace industries is to design the shape of nose cone of flying object that travels at high speeds with optimum values of air drag. This study deals with computational analysis on elliptical nose cone profiles of a commercial aircraft. The effect of nose shape on the drag is studied at subsonic flow with Mach 0.3.The scope of this paper is to develop some prototype profiles with outstanding aerodynamic qualities and low cost for use in construction projects for aircrafts increasing their efficiency and effect on target. The motivation for such a work is caused by lack of data on aerodynamics for profile of elliptical nose cone and especially improved aerodynamic qualities that can be used in designing aircrafts. Flow phenomena observed in numerical simulations for different AOA for elliptical nose cone profile are highlighted, critical design aspects and performance characteristics of selected nose cone are presented.
Keywords: Angle of Attack (AOA), Mach Number, Nose cone, Drag, CFD

1. INTRODUCTION

In sub sonic conditions nose cone shape plays an important role reducing drag force on entire body and not allowing flow separation which are adverse effects on efficiency of an aircraft. It also accounts on the efficiency of an aircraft that can be increased by producing least drag on aircraft [1]. Providing optimum shape to nose is not a difficult when compared to other designs but little care must be taken in account. We also discussed in paper about different Angle of Attack on elliptical cone design and experiences different values of drag and lift. The mostly used nose profiles now-a-days are majorly used are conical, parabolic ogive and elliptical as per commercial purpose [2]. For the analysis, we have chosen elliptical shape as elliptical shape induced less drag compare to other models and is done by using Creo 4.0 software and analyzed using computational fluid dynamics for different Angle of Attack under sub sonic conditions. This work totally based on CFD analysis but used in real world which brings a difference in efficiency of aircraft and not only this flow separation also plays an important role in aircrafts which can be controlled by usage of optimum nose cone [3].

(IJAER) 2020, Vol. No. 20, Issue No. III, September **e-ISSN: 2231-5152, p-ISSN: 2454-1796**



2. PROBLEM STATEMENT

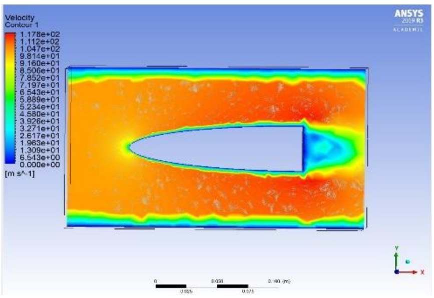
The objective of this paper is to show the variation in drag force on entire body that can be achieved by the shape of the nose of aircraft. For a space vehicle like an aircraft the shape of the nose cone has a significant effect on the drag of the vehicle. So, to increase overall efficiency, we need to give an optimum shape to nose cone which can reduce drag force and provide a streamline structure to an aircraft. Now a days the main problem faced by commercial aircrafts are not having optimum shape or geometry that chooses the aircraft. So, in this paper, we will discuss about now basic nose cone structures used days and how can we improve efficiency of an aircraft by providing optimum geometry to the nose cone.

3. METHODOLOGY

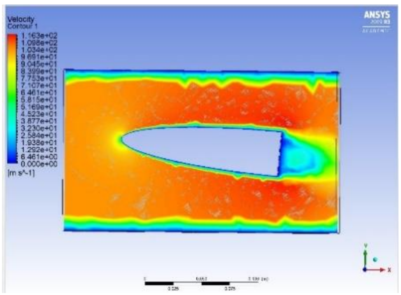
In this paper nose cone shape are designed using Creo 4.0 software. The first step is to create a 2D model as per the equations mentioned above and convert into a 3D model for CFD testing. The commonly used tools to create a model in Creo 4.0 parametric are- Extrude, extrude cut, Revolve, revolve cut Sweep, Swept cut, Fillet, Chamfer, Mirror. CFD Analysis is carried out in three steps i.e. (i) Preprocessing, geometry, – Designing, meshing, boundary conditions and numerical method, (ii) Processing – Solving fluid flow governing equations by numerical method till the convergence is reached and (iii) Post processing – extracting results in terms of graphs, contours which explains the physics of flow and required results [4]. The above three steps are carried out in ANSYS using fluid fluent CFD for designing and meshing with Hybrid grid that is prismatic layer around nose and unstructured grid. Simulations are carried out using ANSYS fluent a finite volume solver at with inlet conditions Mach 0.3 i.e. sonic condition [5].

4. RESULTS AND DISCUSSION

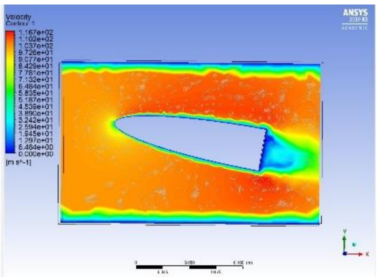
4.1) Velocity Distribution



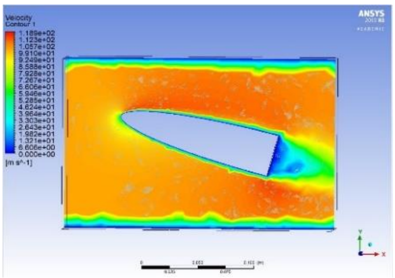
AOA 0 deg



AOA 5 deg



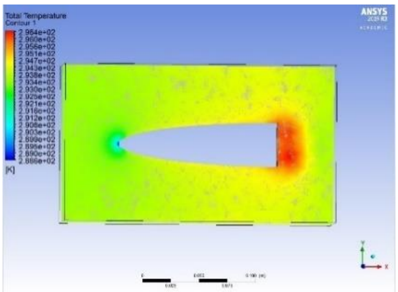
AOA 10 deg



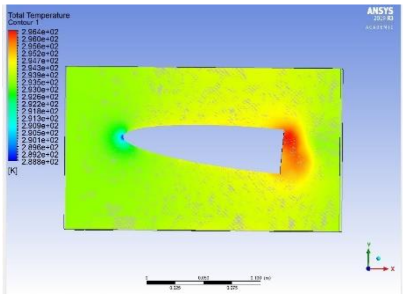
AOA 15 deg

Fig 4.1

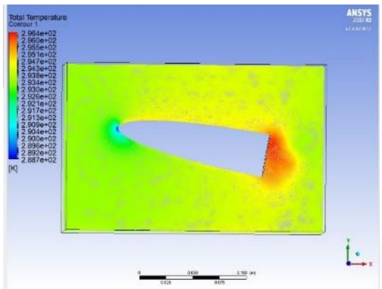
4.2) Total temperature distribution



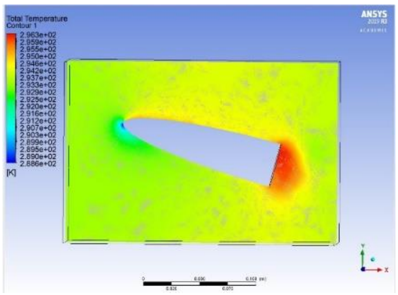
AOA 0 deg



AOA 5 deg



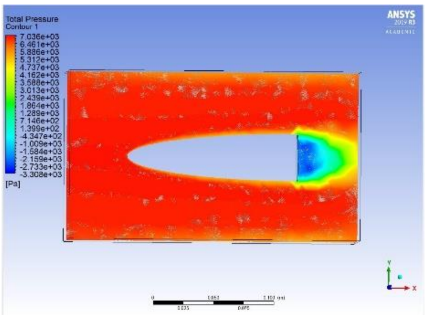
AOA 10 deg



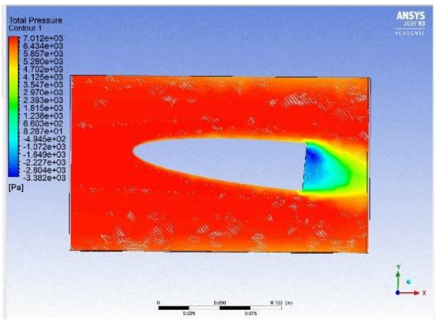
AOA 15 deg

Fig 4.2

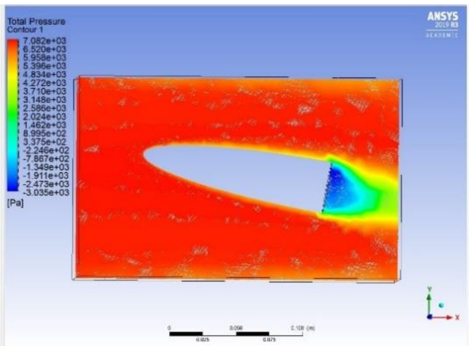
4.3) Total pressure distribution



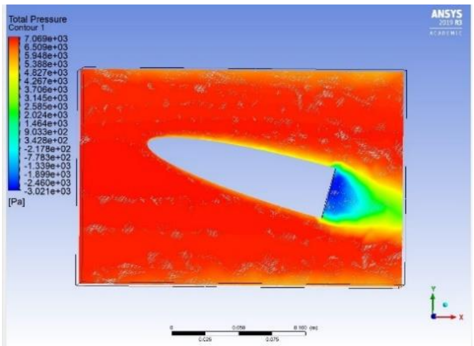
AOA 0 deg



AOA 5 deg



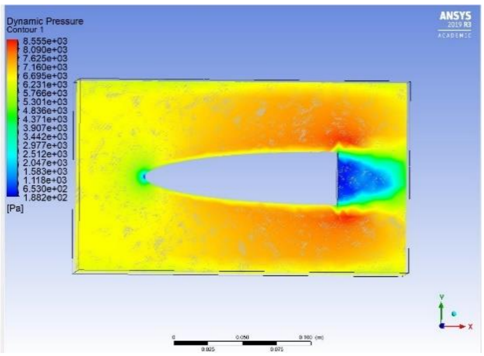
AOA 10 deg



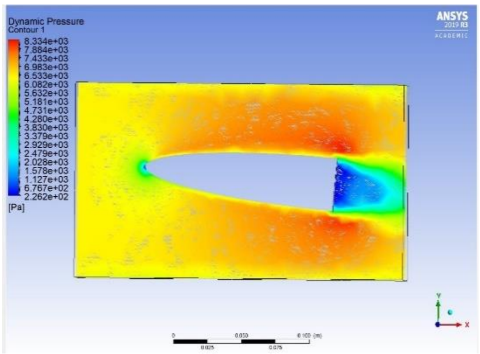
AOA 15 deg

Fig 4.3

4.4) Dynamic pressure distribution

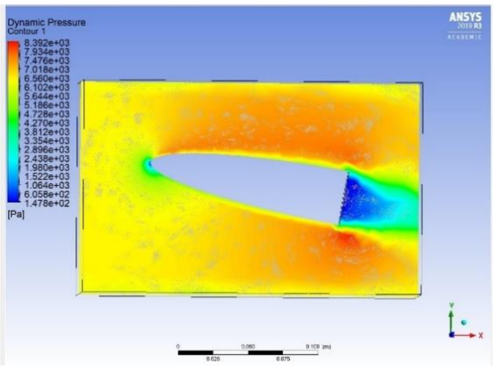


AOA 0 deg

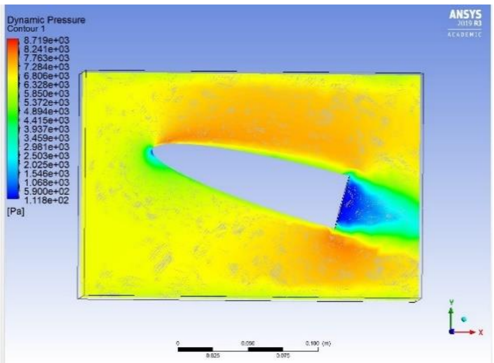


AOA 5 deg

(IJAER) 2020, Vol. No. 20, Issue No. III, September **e-ISSN: 2231-5152, p-ISSN: 2454-1796**



AOA 10 deg



AOA 15 deg

Fig 4.4

S NO.	AOA	Lift (N)	Drag (N)	Cl	Cd	Cp
1.	0	0.093	3.430	0.0015	0.0027	-14.591
2.	5	1.628	3.574	0.0013	0.0028	-16.494
3.	10	2.794	3.892	0.0022	0.0031	-15.135
4.	15	3.779	4.347	0.0031	0.0035	-14.373

Table I

The above results show the variation of total temperature, total Pressure etc. at Mach 0.3 over different Angle of Attack (AOA). As from the table plotted above, we see the variations in values over several parameters deciding the flow variation in nose cone. The maximum value of pressure coefficient at 0 deg. AOA obtained from graph is **1.002** and **0.972, 0.987, 1.006** at 5 deg, 10 deg and 15 deg, respectively. The table also represents the value of Lift, Drag and their coefficients. The maximum value of Lift and Drag are obtained at 15 deg and are- **3.779, 4.347,** respectively.

5. CONCLUSION

The overall presentation of this paper concludes about the variation of flow over nose cone at different AOA. From results, we conclude that the Lift and Drag values are increasing with AOA which results in formation of induced drag over the body surface which increases as well. It also proposes the solution to improve performance over different nose cone profiles. By referring to the results, the minimum drags coefficient values at 0.8, 1.0, 1.5 Mach number which is desirable for subsonic flows to improve efficiency of vehicles by reducing drag.

(IJAER) 2020, Vol. No. 20, Issue No. III, September **e-ISSN: 2231-5152, p-ISSN: 2454-1796**

6. REFERENCES

[1] A Sanjay Varma,, G Sai Satyanarayana, Sandeep "CFD ANALYSIS OF VARIOUS NOSE PROFILES", International Journal of Aerospace and Mechanical Engineering , ISSN (O): 2393-8609, Volume 3 – No.3, June 2016.

[2] Levi C. Wade, "NOSE CONE DESIGN OF AN AERIAL DROP VEHICLE", project no NAG 0802 thesis submitted in full fillment of degree in bachelor science AUGUST 2008.

[3] A. Hemateja, B. Ravi Teja, A. Dileep Kumar, Rakesh, "INFLUENCE OF NOSE RADIUS BLUNT CONES ON DRAG IN SUPERSONIC AND HYPERSONIC FLOWS" IOP Publishing ICMAEM-2017.

[4] P. Keerthana, B.G. Geetha, P. Kanmani, "CRUSTOSE USING SHAPE FEATURES AND COLOR HISTOGRAM WITH KNEAREST NEIGHBOUR CLASSIFIERS", International Journal of Innovations in Scientific and Engineering Research (IJISER), Vol.4, No.9, pp.199-203,2017Y.

[5] James T. clay, major Usaf, BLUNTNESS, "CONE ANGLE, AND MACH NUMBER EFFECTS ON THE STABILITY DERIVATIVES OF SLENDER CONES". Hypersonic Research Laboratory Project No. 7064, ARL 67-0185, September 1967.