Simulation of Radome Moving at Supersonic Speed

[1]Charan M S, [1]Pavan G K, [1]Raghavendra G D, [1]Shyam S, [2]Dharamendara M
[1]U G students & [2] Assistant Professor
School of Mechanical Engineering, REVA University, Bengaluru

Abstract— Aerospace vehicles at supersonic speeds usually have Mach number in range of 1.2–5. These High-speed flights produce severe thermal and pressure load conditions on the radome. Radome design involves multi-disciplinary process. The shape should be designed for minimum aerodynamic drag and least aberrations. The radome must be able to withstand the aerodynamic heating and pressure loads during the flight conditions at the same time the design should not affect the performance of the electronic module such as Seeker and Radio Proximity Fuse. In aerospace applications, radomes often double as a nose cone and thus have a significant impact on the aerodynamics of the aircraft. High velocity and temperature acting on the radome during flight are analyzed using Computational Fluid Dynamics (CFD). This study deals with computational analysis various nose cone profiles. Flow phenomena observed in numerical simulations for different nose cone profiles are highlighted, critical design aspects and performance characteristics of the selected nose cone are presented. Variation of Drag Coefficient (C_d) for various radome shapes are compared.

Index Terms— Aerodynamics, Computational Fluid Dynamics, Drag Coefficient (C_d) Mach number, Nose Cone, Numerical Stimulation, Radome, Supersonic speed.

1 Introduction

1.1 Overview

The shape and design of a radome can dramatically influence the aerodynamic properties of a missile and the electronic performance of the radars and other equipment's stationed inside the radome. There are several equations that can be used to estimate some of the derivatives, but not all of them. These equations are just estimations and can be magnitudes off. Wind tunnel tests are a method that results in derivatives that are highly accurate. The problems with wind tunnel tests are that it is very expensive and can be very time consuming. Also, the wind tunnel models are scaled down to fit in the tunnel, and this can have a dramatic change on the results, since the results do not always scale up as easily. Air will flow over a smaller body differently than a larger body, due to the changes in Reynolds number and other flow characteristics. Using an experienced wind tunnel expert and a highly accurate tunnel can minimize these problems, but will be very expensive. Over the past couple of decades computer simulation has become much more prevalent. Computational Fluid Dynamic software is much more accurate than it once was and is becoming more user's friendly, but it still requires an expert to create a 3-D full aircraft CFD model. The mesh generation for a model can be difficult and requires a great deal of experience. This software is expensive to purchase, but can be used over and over again. Also, many different test cases can be run to determine flying qualities in various situations. Moreover, these programs are easily available to student like us where as wind tunnel tests are not available everywhere. There are also several different programs that are readily available that can produce high fidelity results, and some of these programs can be purchased at a reasonable price. FLUENT is a very high fidelity CFD program, but requires a large amount of experience and time. The main goal of this research was to use high fidelity CFD programs to conduct aerodynamic analysis as well as to test the validity of these engineering level programs.

1.2 Purpose of Radome

A radome is a structural, weatherproof enclosure that protects a radar antenna. The radome is constructed of material that minimally attenuates the electromagnetic signal transmitted or received by the antenna, effectively transparent to radio waves. Radomes protect antennas from structural damage due to wind, precipitation, and bird strikes. In aerospace applications, Radomes often double as a nose cone and thus have a significant impact on the aerodynamics of the aircraft. While Radomes should be designed not to affect the performance of the underlying antennas, they also must satisfy structural and aerodynamic requirements.

Radomes can be constructed in several shapes — spherical, geodesic, planar, conical, bicone, tangent ogive, elliptical etc. —depending on the particular application, using various construction materials such as fiberglass, PTFE-coated fabric, and others.

- *Charan M S, Pavan G K, Raghavendra G D, Shyam S are currently pursuing bachelor's degree in Mechanical Engineering in REVA University, Bengaluru, India.*
- *Dharamendara M, Professor, School of Mechanical Engineering, REVA University, Bengaluru, India*
E-Mail: charansomashekar@yahoo.com rgdgrdrgd@gmail.com pgkrgk51@gmail.com shyams9823@gmail.com dharmendram@reva.edu.in

1.3 Design Considerations

The ideal radome shape for radar transmission is a hemisphere, but the most aerodynamically efficient cross section for hypersonic flight is a slender body. Existing radome contours represent a compromise between these two extremes. Maximizing radome volume to accommodate more electronic hardware is another goal in radome shape design. Radome contours are usually cones, ogives, or combinations thereof. The key radome design requirements are summarized below:

- It must convey the energy with minimum loss.
- It must have minimum aerodynamic drag.
- It must have satisfactory physical properties, such as sufficient strength, resistance to thermal shock (from rapid aerodynamic heating), resistance to rain erosion at high speeds, and minimum water absorption.

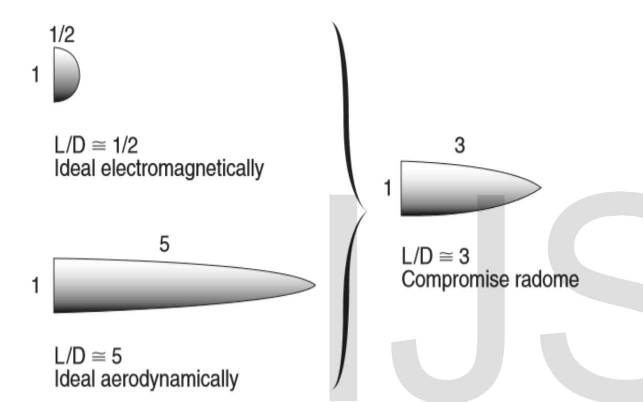


Fig 1. Illustrates Three Conceivable Radome Shapes.

For minimum angular distortion, a hemispherical shape (or hyper hemispherical shape as in a ground-based radar) would be ideal electromagnetically (upper left), but the drag penalty is excessive. From an aerodynamic perspective, the lower left radome shape is preferable, but it tends to have significant angular distortion characteristics. The tangent-ogive shape (on the right) is a typical compromise design. Nevertheless, some missiles use much blunter dome designs despite the drag penalty. L/D, lift-to-drag ratio.

2 LITERATURE REVIEW

A number of numerical and experimental analysis were conducted between radome and air flow during flight and static conditions. Radome seek for more capable approaches to study the fluid dynamics and pressure loading under flight conditions.

[1] Lucas de Almeida Sabino "CFD Analysis of Drag Force for Different Nose Cone Design". In this paper, the authors analyzed by means of a computational procedure the influence of the shape of rocket nose cones varying the speed in each of them, with this it was possible to analyze the drag force generated, having as a working parameter in subsonic medium ranging from a Mach 0.05 to approximately 0.62, considering standard conditions for temperature and pressure.

[2] A. Yeshwanth.et.al "Nose Cone Design and Analysis of An Avion". The objective of this paper is to show minimum drag force on entire body 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 stream line structure to an aircraft.

[3] Martin Schmucker.et.al "WHIPOX All Oxide Ceramic Matrix Composites". The purpose of this paper is to determine the mechanical and thermal properties of WHIPOX. WHIPOX is the fiber-reinforced porous matrix CMC. The composites consist of aluminons-silicate fibers and mullite. Depending on the matrix composition shear strength value about 10MPa, young's moduli of approximately 150GPa and thermal conductivity nearly 1W/mK. Since no extra coating is required to obtain these property makes WHIPOX inexpensive CMC material.

[4] Oliver M. Hohn.et.al "Experimental Investigations for The Thermal Qualification of High-Speed Missile Radomes". This paper presents the results of experimental investigations for the thermal qualification of radomes for high-speed missiles made of the oxide ceramic matrix composites WHIPOX with different types of thermal insulation Experiments were conducted in a blowdown wind tunnel for aerothermodynamic investigations at Mach 3 flight conditions and in an arc heated wind tunnel at flight-relevant heat loads of higher Mach numbers derived from sample trajectories.

[5] Ben Stewart.et.al "CFD Analysis of The HyCause Nosecone". This paper presents the results of Computational Fluid Dynamic calculations performed to determine the drag and heating characteristics of three nosecone geometries at hypersonic flight conditions. Also included are calculations of a triple-ramp two-dimensional scramjet intake and combustor.

[6] A Sanjay Varma, et.al. "Cfd Analysis of Various Nose Profiles". The purpose of this paper is to propose a solution for performance improvement using various missiles nose profiles. von Karman ogive nose profile give higher critical Mach number and minimum pressure coefficient which is desirable for the subsonic flows as stated in problem International Journal of Pure and Applied Mathematics Special Issue definition.

[7] Levi .C Wade "Nose Cone Design of An Aerial Drop Vehicle". The purpose of this paper is to develop a nose cone using Kazak Composites. After analysis of selected nose cones, the rubber nose concept conforms with the requirements for structural integrity, weight, functionality, and cost.

[8] Hemateja et.al "Influence of Nose Radius Blunt Cones on Drag in Supersonic and Hypersonic Flows". This paper mainly focusses on reducing drag force and heat generation on nose cone while a craft moving at high speeds and how both are co related to each and this paper gives approximately optimal results and still research has to be done.

[9] G. A. Crowell Sr. "The Descriptive Geometry of Nose Cone". This paper has a necessary collection of equations describing the various nose cone shapes. The also brought together some nomenclature and nose cone information as these equations were very useful to design different nose cone shapes according to the required characteristics.

[10] LECTURE 7: Turbulence Modelling, Introduction to Ansys Fluent, 2014. This lecture teaches to use the Reynolds number to determine whether the flow is turbulent. To select the turbulence model. To choose which approach to use for modeling flow near walls. To specify turbulence boundary conditions at inlet.

This thesis presents an assessment of possible approaches flow and pressure case in commercial CFD and with an estimation to demonstrate the real physical phenomena.

3 Problem Statement

The objective of this paper is to show the flow over different shapes of the nose cone /radome. 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 stream line structure to an aircraft. So, in our paper, we are going to discuss about basic nose cone structures used now days and how can we improve efficiency of an aircraft by providing optimum geometry to the nose cone. The present problem is analyzed using CFD software. Flow phenomena observed in numerical simulations different nose cone profiles.

3.1 Governing Equations

The governing equations with which the physical phenomenon of flow over the radome are as follows:

Navier Stokes Equation:

$$\rho\left(\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v}.\nabla\mathbf{v}\right) = -\nabla\mathbf{p} + \nabla.\mathbf{T} + \mathbf{f} \qquad (1)$$

Conservation of Mass Equation:

$$\frac{\partial\rho}{\partial t} + \partial\frac{\partial}{\partial x_i}(\rho x_i) = 0 \qquad (2)$$

Momentum Conservation Equation:

$$\frac{\partial}{\partial t}(\rho\,\vec{v}) + \nabla.(\rho\,\vec{v}\vec{v}) = -\nabla p + \nabla.(\bar{\bar{\tau}}) + \rho\,\vec{g} + \vec{F} \qquad (3)$$

Energy Equation:

$$\begin{Bmatrix}\text{Rate of change of}\\ \text{energy inside the}\\ \text{fluid element}\end{Bmatrix} = \begin{Bmatrix}\text{Net flux of}\\ \text{heat into}\\ \text{the element}\end{Bmatrix} + \begin{Bmatrix}\text{Rate of working done on}\\ \text{the element due to body}\\ \text{and surface forces}\end{Bmatrix}$$

$$\frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_k}(\rho \mathbf{u}_i \mathbf{H}) = \frac{\partial}{\partial x_i}(\mathbf{u}_j \tau_{jk}) + \frac{\partial q_k}{\partial x_k} \qquad (4)$$

Where,

v is the flow velocity,

ρ is the fluid density,

p is the pressure,

T is the stress tensor,

f represents body forces (per unit volume) acting on the fluid,

∇ is the del operator,

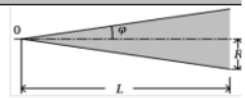
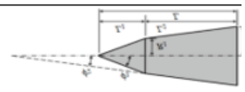
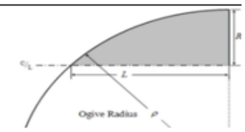
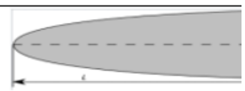
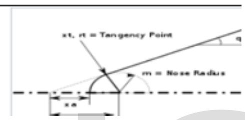
F -mass force per volume unit

i, j, k=1,2,3…

4 Methodology

In this paper, 5 nose cone shapes are designed using **Solid Works** software. The first step is to create a 2D model as per the equations mentioned in **Table 1** and convert into a 3D model for CFD testing. The commonly used tools to create a model in solid works 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. The above three steps are carried out in **ANSYS** using fluid **Fluent** CFD for designing and meshing. Simulations are carried out using **ANSYS CFX** a finite volume solver at with inlet conditions **Mach 3** for each nose by using fluid fluent model with convergence criteria.

TABLE 1-DIFFERENT SHAPES AND FORMULA

NAME	FORMULA	SHAPE
CONIC	$y = \frac{xR}{L}$	
BI-CONIC	$y = \frac{xR_1}{L_1}$ $: y = R_1 + \frac{(x-L_1)(R_2-R_1)}{L_2}$	
TANGENT OGIVE	$\rho = \frac{R^2+L^2}{2R}$ $y = \sqrt{\rho^2-(L-x)^2}+R-\rho$	
ELLIPTICAL	$y = R\sqrt{1-\frac{x^2}{L^2}}$	
BLUNT CONICAL	$x_t = \frac{L^2}{R}\sqrt{\frac{r_n^2}{R^2+L^2}}$ $y_t = \frac{x_t R}{L}$	

5 RESULTS

The results are extracted from CFD POST after the analysis from CFX solver as shown in below figures. These results give the contours of velocity distribution, temperature distribution, density distribution, pressure distribution over the radome for various nose shapes.

5.1 Velocity Distribution Contours

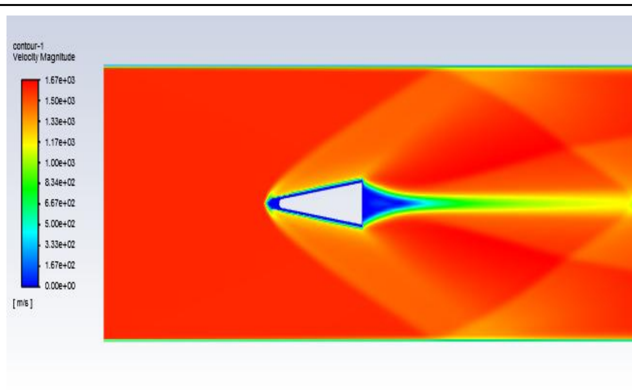


Fig 2. Velocity Distribution of Blunt Conical Shape Radome

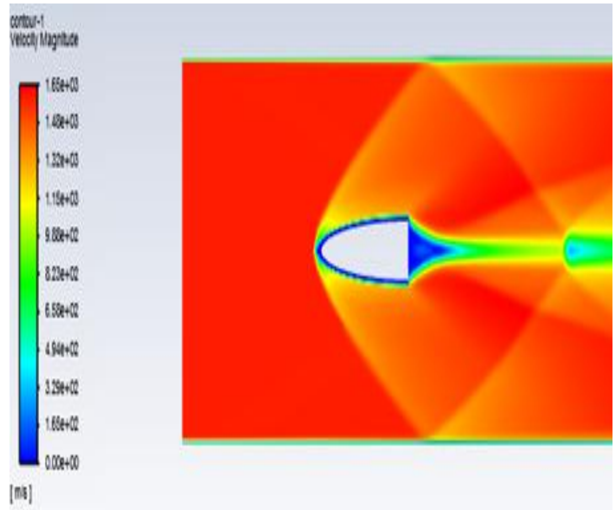


Fig 3. Velocity Distribution of Elliptical Shape Radome

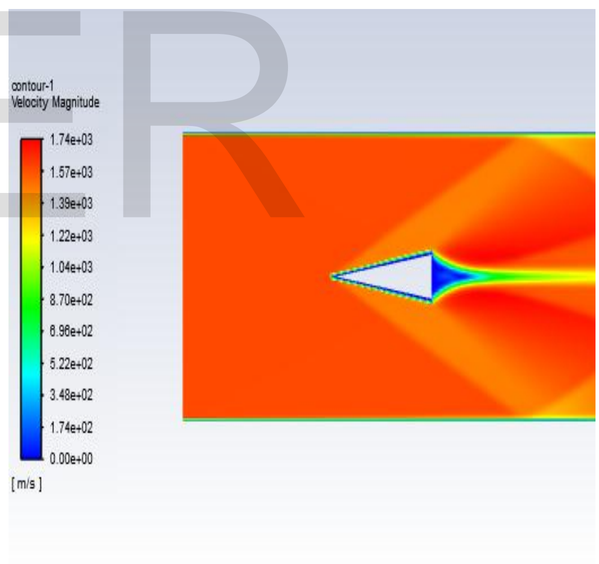


Fig 4. Velocity Distribution of Conical Shape Radome

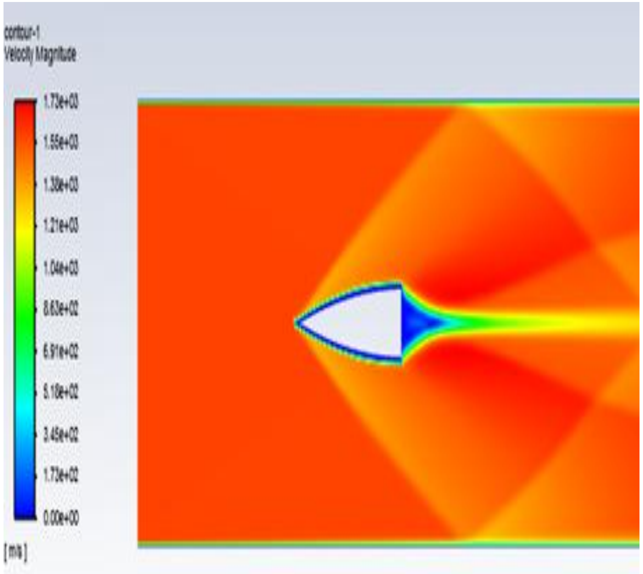


Fig.5 Velocity Distribution of Tangent Ogive Shape Radome

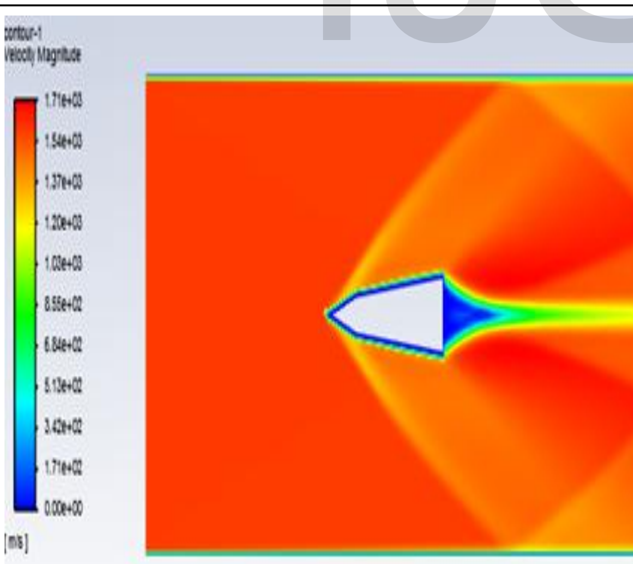


Fig.6. Velocity Distribution of Bicone Shape Radome

5.2 Pressure Distribution Contours

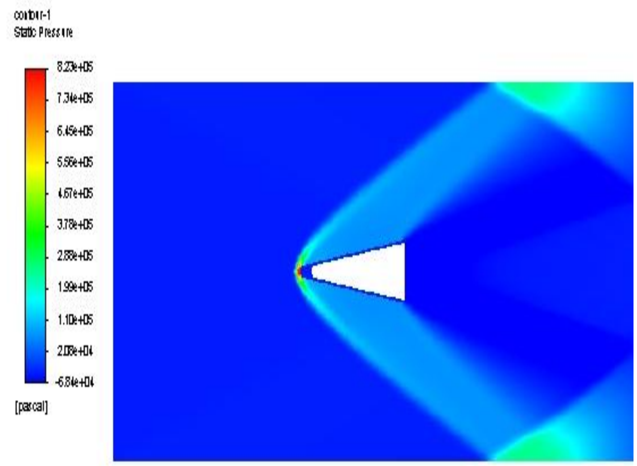


Fig.7 Pressure Distribution of Blunt conical Shape Radome

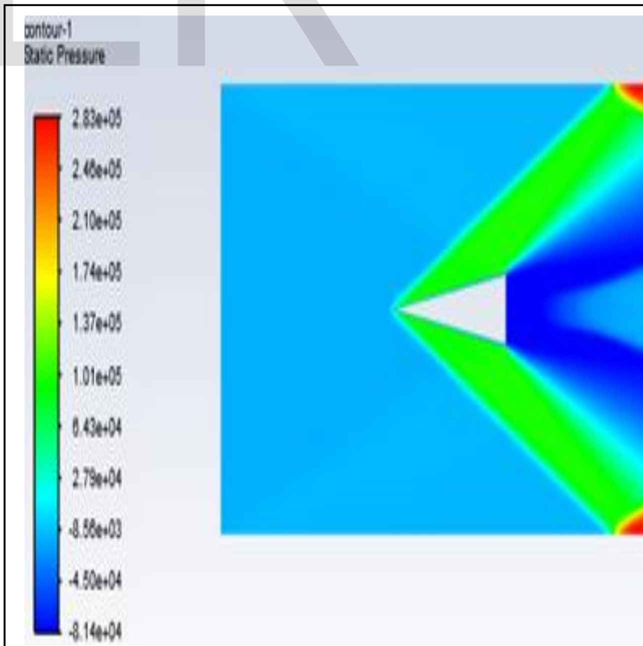


Fig 8. Pressure Distribution of Conical Shape Radome

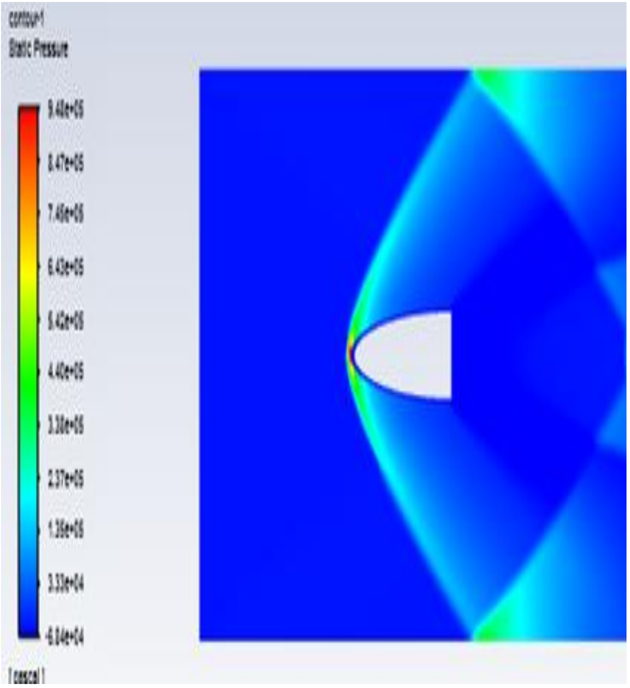
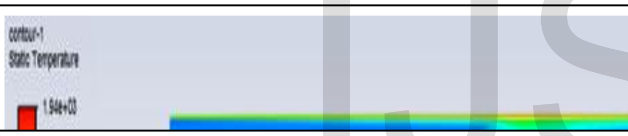


Fig.9 Pressure Distribution of Elliptical Shape Radome



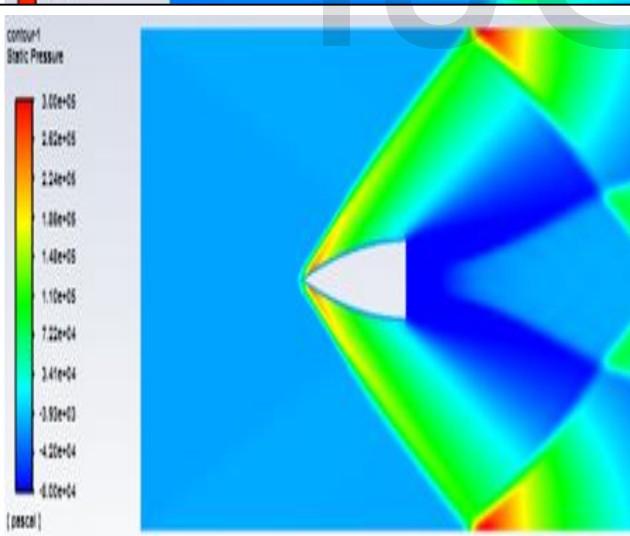


Fig .10 Pressure Distribution of Tangent Ogive Shape Radome

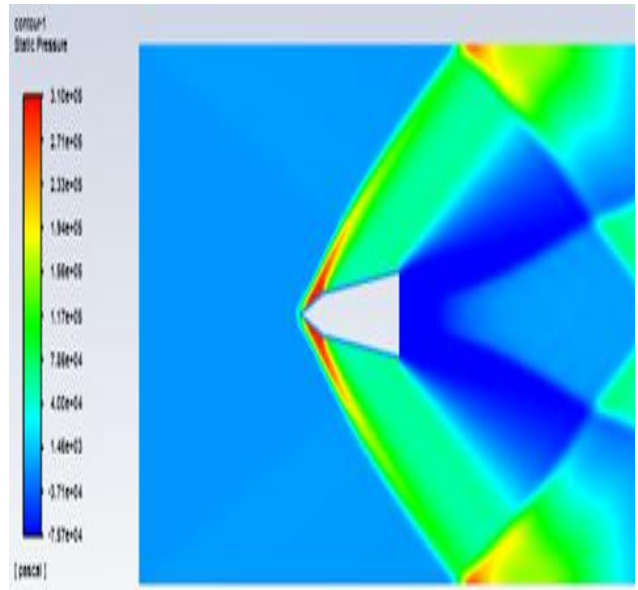
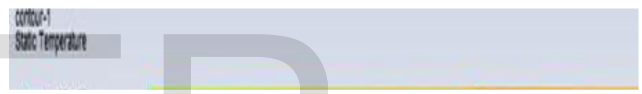


Fig.11 Pressure Distribution of Bi-Cone Conical Shape Radome



5.3 Temperature Distribution Contours

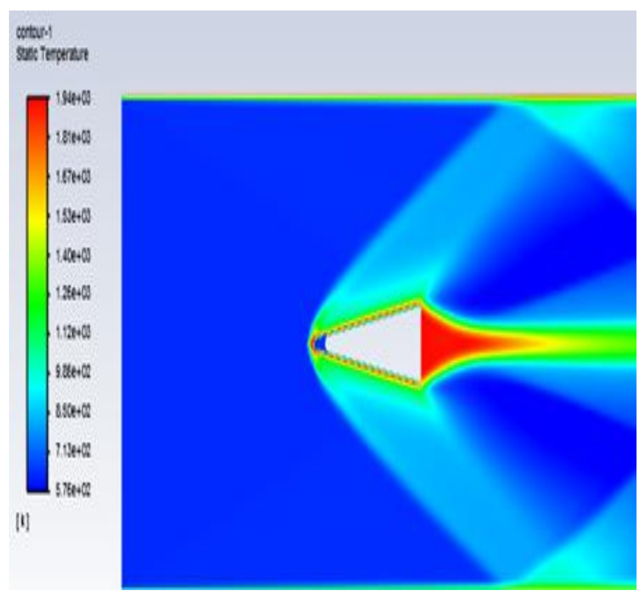


Fig.12 Temperature Distribution of Blunt Conical Shape Radome

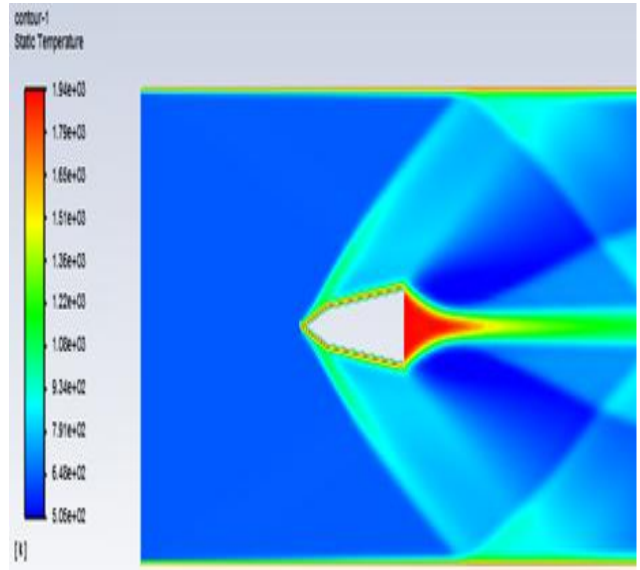


Fig.16 Temperature Distribution of Bi-Cone Conical Shape Radome

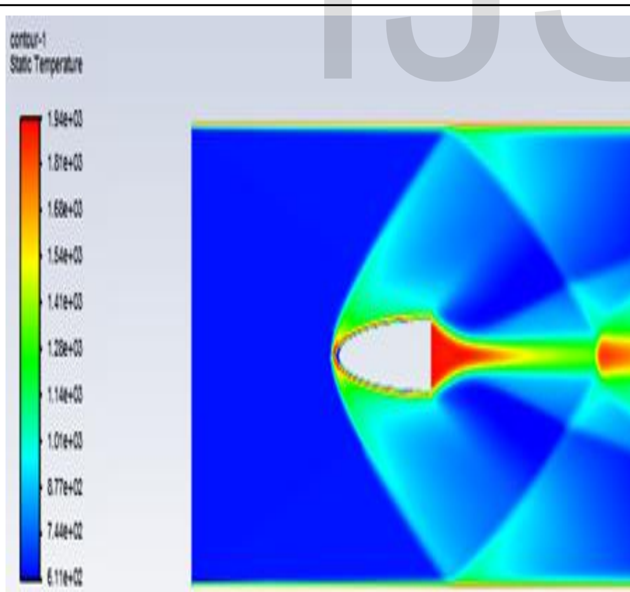


Fig.14 Temperature Distribution of Elliptical Shape Radome

5.4 Variation of Drag Coefficient (Cd) For Various Nose Cone Profiles

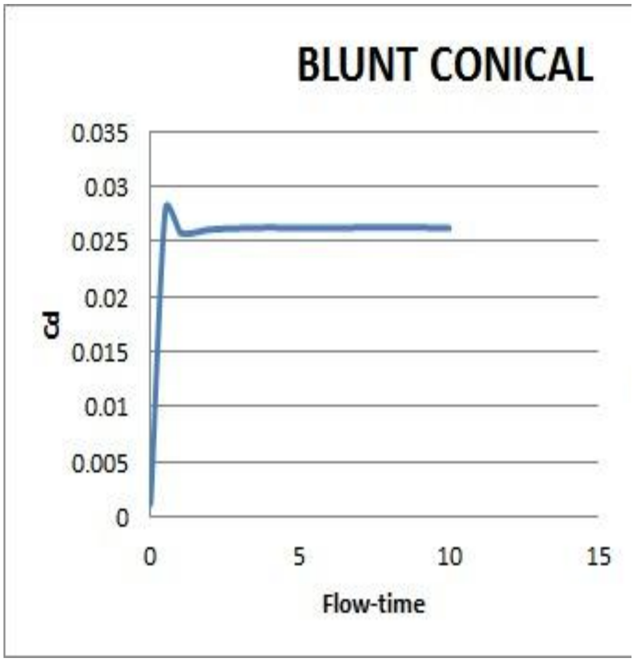


Fig.17 Variation of Cd Vs Flow Time for Blunt Conical Shaped Radome

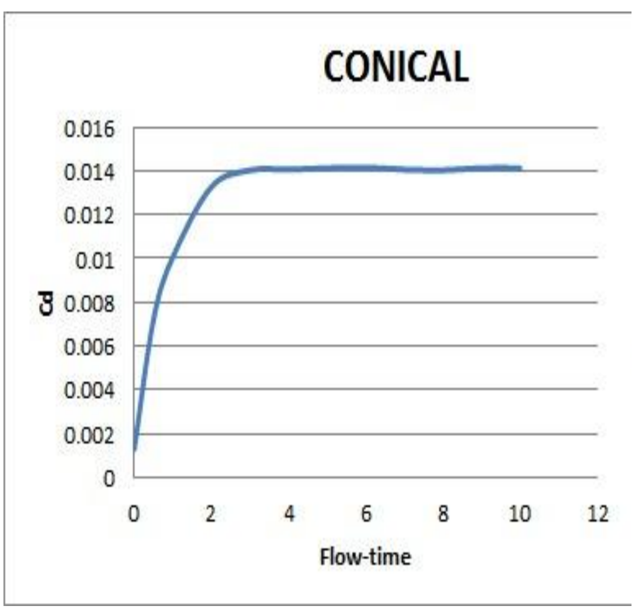


Fig.18 Variation of Cd Vs Flow Time for Conical Shaped Radome

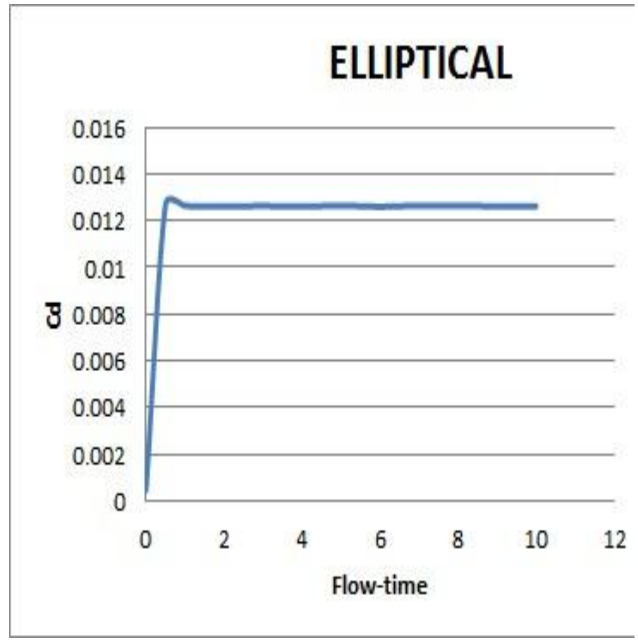


Fig.19 Variation of Cd Vs Flow Time for Elliptical Shaped Radome

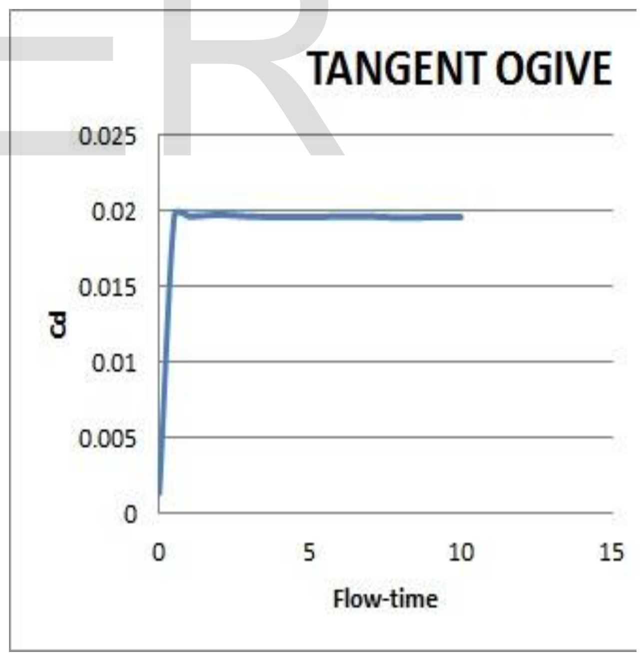


Fig.20 Variation of Cd Vs Flow Time for Tangent Ogive Shaped Radome

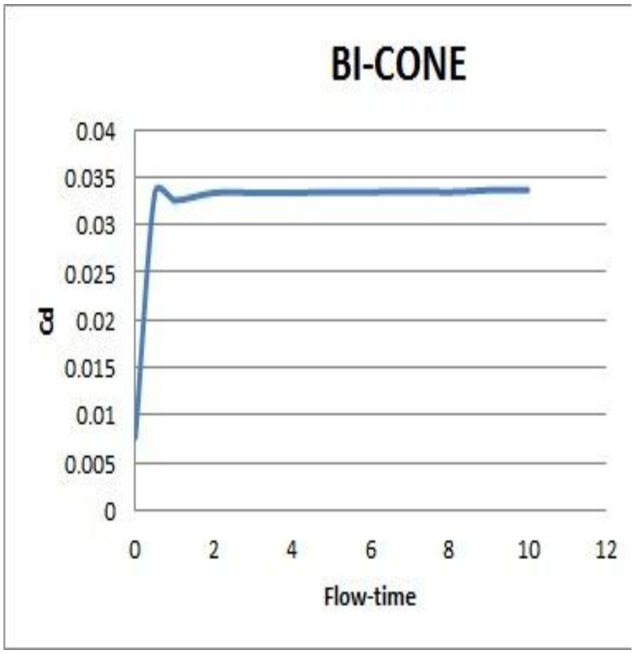


Fig. 21 Variation of Cd Vs Flow Time for Bi-Conical Shaped Radome

5.5 Comparison of Drag Coefficient (Cd) For Various Nose Cone Profiles

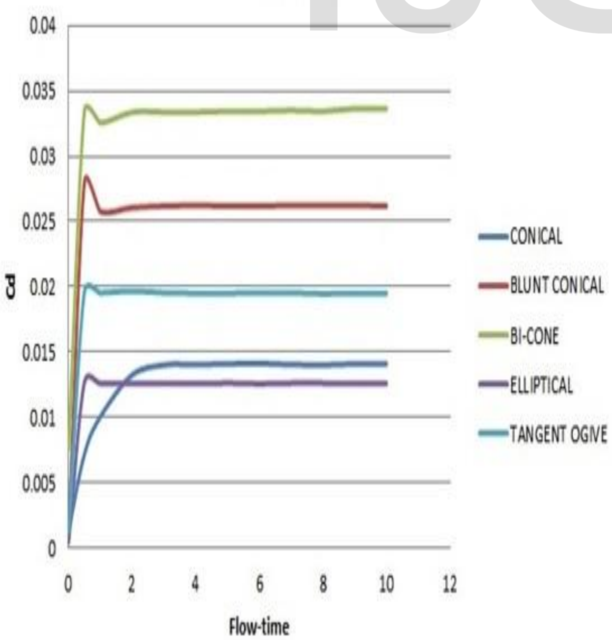


Fig. 22 Comparison of Cd for Various Radome Shapes

6 CONCLUSIONS

- The work included the pre-processing and simulation of Various radome shapes.
- Then the results are successfully computed for drag generated by the various nose cone shapes and the contours of pressure, velocity and temperature have been plotted.
- The proposed objectives of the project have been successfully achieved. Contours of static pressure, velocity magnitude, static temperature have been studied and analyzed. They are found coherent with the expected phenomena.
- The Variation of Drag Coefficient (Cd) vs Flow time for various radome shapes are plotted and compared.
- From the comparison it is evident that Elliptical Shaped radome produces the least drag coefficient.
- A low drag coefficient implies that the streamline shape of the Elliptical shaped radome's body is such as to enable it to move easily through the surrounding air medium with the minimum of resistance. Thus, increasing its aerodynamic efficiency.

7 SCOPE FOR FUTURE WORK

- CFD findings are indicative only, therefore experimental study is more reliable.
- Hence experimental evaluation of the above results using wind tunnel tests is a possibility.
- Also, comparison of other nose cone shapes like secant ogive, Parabolic, Power Series, Haack Series Type and Von-Karman shapes is a possibility.
- With modern technology advancing, future work would see a different approach to gaining accurate results, an idea would be to use a 3D scanner to gauge a true likeness of the object shape and its properties and put it under CFD analysis.

REFERENCES

- [1] Lucas de Almeida Sabino Carvalho and Geovanio Claudino Cavalcante Filho, "CFD Analysis of Drag Force for Different Nose Cone Design", Department of Mechanical Engineering – University of Federal do Rio Grande do Norte, Av. October 2019
- [2] A. Yeshwanth and PV. Senthil, "Nose Cone Design and Analysis of An Avion", Department of Mechanical Engineering, SSE, Saveetha University, Chennai, India, Volume 119 No. 12 2018, 15581-15589.

- [3] Martin Schmucker and Hartmut Schneider, "WHIPOX All Oxide Ceramic Matrix Composites", German Aerospace Center (DLR) Institute of Materials Research Linder Hone, D-51147 Koln. pp:423-435.
- [4] Oliver M. Hohn, Burkard Esser, Josef Klevanski, Ali Gülhan, Resul Fener, Britta Panthen and Martin Frieb, "Experimental Investigations for The Thermal Qualification of High-Speed Missile Radomes", 8TH European conference for aeronautics and space sciences (EUCASS). July 2019, pp:1-11.
- [5] Ben Stewart, Russell Boyce, Andrew Neely and Judy Odam, "CFD Analysis of The Hycause Nosecone", University of New South Wales, Australian Defence Force Academy, Canberra, 2600, Australia.
- [6] 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.
- [7] Levi Wade, "Nose Cone Design of An Aerial Drop Vehicle", Project no NAG 0802 thesis submitted in fulfillment of degree in bachelor science AUGUST 2008
- [8] 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.
- [9] G. A. Crowell Sr., "The Descriptive Geometry of Nose Cones", pp. 1-6, 1996.
- [10] Lecture 7: Turbulence Modelling, introduction to ANSYS Fluent, 2014