IARJSET

ISSN (O) 2393-8021, ISSN (P) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

ISO 3297:2007 Certified ⋇ Impact Factor 7.105 ⋇ Vol. 9, Issue 6, June 2022

DOI: 10.17148/IARJSET.2022.9607

DESIGN AND ANALYSIS OF AEROSPIKE BELL NOZZLE TO IMPROVE THRUST IN ROCKET ENGINE

A J Sriganapathy[1], Makudesh Arjun Kumar. A[2], Diwakar.R[3], Sathish kumar.M[4], Rahul.B[5]

[1]Assistant Professor, Department of Aeronautical Engineering, Mahendra Institute of Engineering And Technology, Thiruchengode, Namakkal, Tamil Nadu 637503

[2,3,4,5] Student of Aeronautical Engineering, Mahendra Institute of Engineering And Technology, Thiruchengode, Namakkal, Tamil Nadu 637503

Abstract: The present computational Analysis deal with improvement of momentum distribution of the high temperature exhaust in Rocket Engine through change in profile of Aero-spike nozzle. Due to change of profile, we can improve thrust and mass flow rate of exhaust gas. This project deals with the Design and Analysis of Aero-spike Nozzle by adding the bell configuration at the end of the Nozzle and also by providing the dimples at the spike of the nozzle to produce higher thrust than the normal aerospike nozzle. The design of the nozzle is carried out with the design software CATIA V5 and the analysis of the Nozzle is done in ANSYS FLUENT 19.2 software. The presence of Dimples at the spike of the nozzle increase the turbulence and enables the flow to be attached to the spike thereby reducing the flow separation of the nozzle. This Profile modification of the Nozzle produces considerable increase in thrust compared to the conventional Aerospike nozzle. The comparison has been done with respect to mass flow rate , velocity , streamline and temperature between unmodified profile and modified profile.

1. INTRODUCTION

The Present Computational Analysis deal with improvement of momentum distribution of the high- temperature exhaust in Rocket Engine through change in profile of Aero-spike nozzle. Due to change of profile we can improve thrust and mass flow rate of exhaust gas. This analysis is done by using Fluent (ANSYS FLUENT (19.2)) on modified profile of Aero-spike nozzle. The Comparison has been done with respect to mass flow rate, velocity streamline and vector between unmodified and modified profile. This Profile modification has done on basic Aero-spike nozzle and this CAD Model designed by CATIA V5. It has been analyzed that due to change profile in exist nozzle area. The new has improved mass flow rate of exhaust gas and also reduced non –uniform exhaust flow[1-2].

Flight research of an Aero-spike rocket nozzle was conducted using high power solid rockets. The lower Aero-spike chamber pressures and thrusts were likely to be caused by a larger actual Aero-spike nozzle throat area than the designed throat area. The design work on an Aero-spike nozzle design and associated testing hardware has been completed in preparation for a series of cold-flow tests on a truncated Aero-spike nozzle. The system will allow the evaluation of aerodynamic thrust vectoring and thrust augmentation through truncated Aero-spike base bleed. This series of tests will facilitate calibration of analytical prediction tools which include computational fluid dynamics results [3-4].

An Aero-spike nozzle model is selected for evaluating multidisciplinary optimization procedure. A multidisciplinary analytic model of a linear Aero-spike rocket nozzle has been developed; this model includes predictions of nozzle thrust, nozzle weight, and effective vehicle gross-lift off weight. The contour of the Aero-spike nozzle has been designed for maximum thrust at one design condition. The Aero-spike geometry (length, base height, surface contour) and the structural (dimensions like tube radii and thickness) design parameters are computed to satisfy a structural constraint (displacement, stress and buckling). The Design optimization formulation has been implemented with a goal of minimizing gross-lift off weight. For this thrust and nozzle wall pressure calculations were made using CFD and were linked to structural FEA for determining nozzle weight and structural integrity. Calculations for specific impulse and engine thrust to weight ratio are executed to determine optimum vehicle lift off weight [5].

The Multidisciplinary analysis was integrated with an optimization procedure that allowed investigation of Multidisciplinary feasible strategy. This MDO result are compared with the separate aerodynamics & structural optimized design which shows the comparatively improvement over individual results. The Rocket dyne-developed F-1 engine is the most powerful single-nozzle liquid-fuelled rocket engine ever flown. The RD-170 produces 11% more

IARJSET

ISSN (O) 2393-8021, ISSN (P) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

ISO 3297:2007 Certified ⁜ Impact Factor 7.105 ⁜ Vol. 9, Issue 6, June 2022

DOI: 10.17148/IARJSET.2022.9607

and the RD-171produces 20% greater thrust using a cluster of four combustion chambers and four nozzles. The M-1 rocket engine was designed to have more thrust; however, it was only tested at the component level. The F-1 was a liquid-fuelled rocket motor, burning RP-1 (kerosene) as fuel, and using liquid oxygen (LOX) as the oxidizer[6].Aero-spike nozzles are being considered in the development of the Single Stage to Orbit launching vehicles because of their prominent features and altitude compensating characteristics. The annular nozzle, also sometimes known as the plug or "altitude- compensating" nozzle, is the least employed of those discussed due to its greater complexity.
The term "annular" refers to the fact that combustion occurs along a ring, or annulus, around the base of the nozzle. "Plug" refers to the center body that blocks the flow from what would be the center portion of a traditional nozzle. As any fluid dynamists recognizes, the significant disadvantage of the "flat" plug is that a turbulent wake forms aft of the base at high altitudes resulting in high base drag and reduced efficiency "Altitude-compensating" is sometimes used to describe these nozzles since that is their primary advantage, a quality that will be further explored later. However, this problem can be greatly alleviated in an improved version of the truncated spike that introduces a "base bleed,”. This paper presents the aerodynamic features performance characteristics[7-10].Here we develop a computer code which uses the Method of Characteristics and the Stream Function to define the traditional Aero-spike nozzle contour for isentropic, inviscid, irrotational supersonic flows of any working fluid for any user-defined exit Mach number. The contour obtained is compared to theoretical isentropic area ratios for the selected fluid and desired exit Mach number. The accuracy of the nozzle to produce the desired exit Mach number is also checked. The flow field of the nozzle created by the code is independently checked with the commercial Computational Fluid Dynamics (CFD) code ANSYS FLUENT (19.2)- FLUENT. ANSYS FLUENT (19.2)- FLUENT predictions are used to verify the isentropic flow assumption and that the working fluid reached the user-defined desired exit Mach number.Good agreement in area ratio and exit Mach number is going to be achieved, verifying that the code is accurate. The code developed proves to be a useful tool in creating annular nozzle contour for isentropic, irrotational, inviscid flow. The program exhibits increasing accuracy in the exit Mach number and exit area ratio as the incremental Prandtl- Meyer expansion angle decreases. This accuracy increase is independent of fluid or desired exit Mach number. The exit Mach number of the nozzles calculated with the program shows good agreement with the ANSYS FLUENT (19.2)-FLUENT simulated exit Mach numbers [11].

2. MODELLING OF NOZZLES:

The Design parameters of the nozzles are shown in the table below

Design Parameters	Values
Throat area, A_t	1.853 in^2
Exit area, A_e	9.621 in^2
Exit area ratio, A_e/A_t	5.192
Rocket flow specific heat ratio, γ	1.194
Nozzle exit Mach number M_e	2.802
Rocket chamber pressure P_c	500 psia
Nozzle exit pressure P_e	15.34 psia

Table 1. Design parameters

Using the CATIA V5 software, constructed a solid Aero-spike nozzle and the isometric view of the nozzle are given: The 3D modelling of the Aero-spike nozzle is done by the Catia v5 by the following steps:

➢ start
➢ Mechanical design
➢ Part design
➢ Sketch drawn using tools such as profile, line, arc
➢ Select constraint and give the dimensions
➢ Exit the work bench
➢ Select the command – SHAFT
➢ O K

IARJSET
ISSN (O) 2393-8021, ISSN (P) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
ISO 3297:2007 Certified ⁜ Impact Factor 7.105 ⁜ Vol. 9, Issue 6, June 2022
DOI: 10.17148/IARJSET.2022.9607

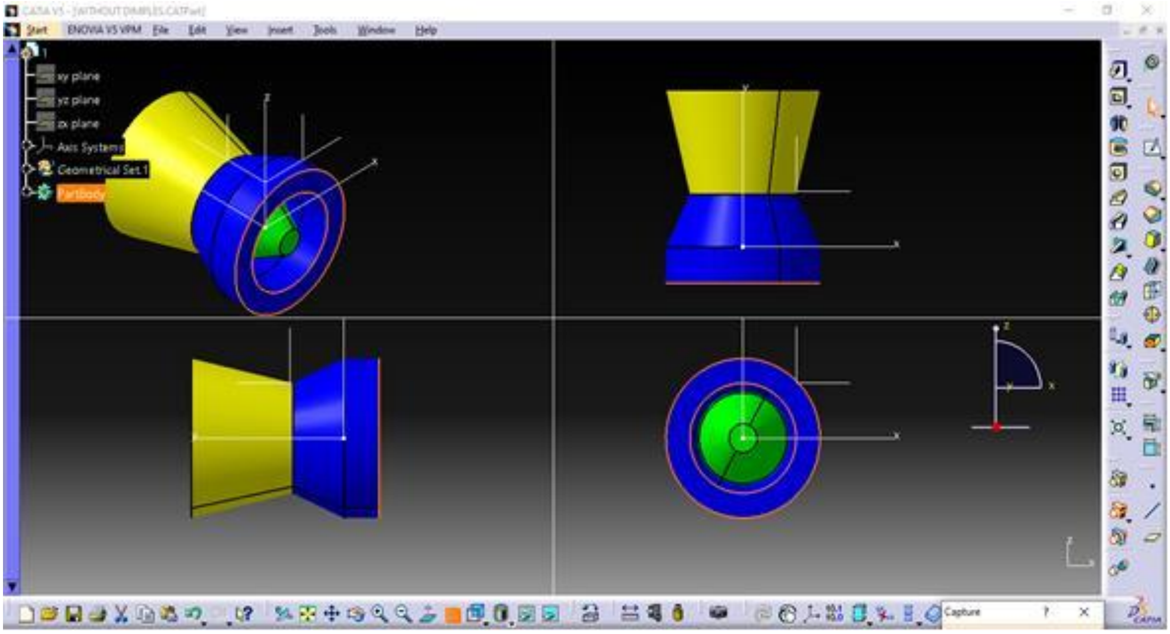


Figure 1 . 3D Model of Aero-spike bell Nozzle.

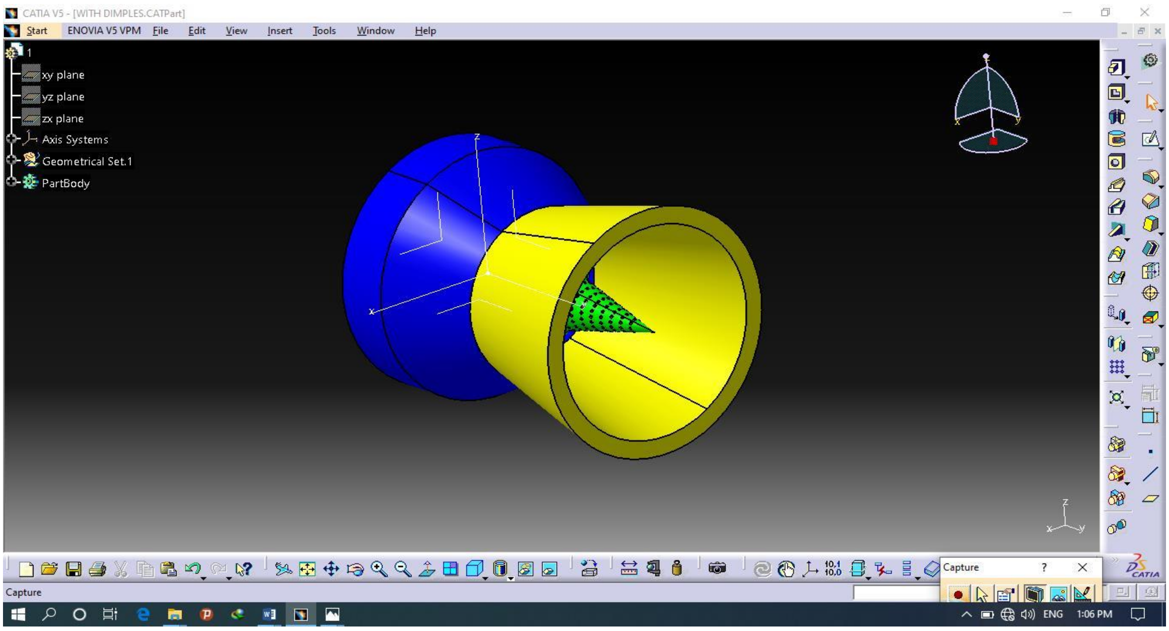


Figure 2 .3D Model of Aero-spike Bell Nozzle with Dimples.

4. ANALYSIS OF NOZZLES

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flows. Computers are used to perform the calculations required to simulate the free-stream flow of the fluid, and the interaction of the fluid (liquids and gases) with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved, and are often required to solve the largest and most complex problems.
CFD is applied to a wide range of research and engineering problems in many fields of study and industries, including aerodynamics and Aero-spike analysis, weather simulation, natural science and environmental engineering, industrial system design and analysis, biological engineering, fluid flows and heat transfer, and engine and combustion analysis.

DOI: 10.17148/IARJSET.2022.9607

ANALYSIS OF THE MODEL USING ANSYS SOFTWARE:

computational fluid dynamics (CFD) simulation software is a comprehensive suite of products that allows you to predict, with confidence, the impact of fluid flows on your product — throughout design and manufacturing as well as during end use. ANSYS renowned fluid analysis tools include the widely used and well-validated ANSYS FLUENT (19.2) Fluent and ANSYS FLUENT (19.2) CFX, available separately or together in the ANSYS CFD bundle.
Because of the robustness and speed of its solvers, knowledge and experience of its development teams, and advanced modeling capabilities, ANSYS FLUENT (19.2) fluid dynamics solutions provides results you can trust Combining Fluent or CFX with the full- featured ANSYS CFD-Post fluid flow post-processing tool allows you to perform advanced quantitative analysis or create high-quality visualizations and animations.
ANSYS fluid dynamics solutions are fully integrated into the ANSYS Workbench platform. This environment delivers high productivity and easy-to-use workflows. Workbench integrates all your fluid workflow needs (pre-processing, simulation, and post-processing) as well as Multiphysics functionality (fluid–structure interaction, electromagnetic/fluid coupling).
Here the analysis of the nozzle is carried out using the ANSYS FLUENT (19.2)

STEPS INVOLVED IN ANALYSIS OF THE MODEL USING ANSYS SOFTWARE ARE

- Pre-processing.
- Defining the geometry.
- Meshing the Model.
- Set-up the boundary conditions.
- Post- Processing (Obtaining the solution and result).

PRE-PROCESSING

A preprocessor is a program that processes its input data to produce output that is used as input to another program.

DEFINING THE GEOMETRY:

It involves in defining the model that is going to be analyzed either by creating the geometry or by importing the required model which is already created in the 3d software
like (Auto-cad, Catia, and Creo etc.)

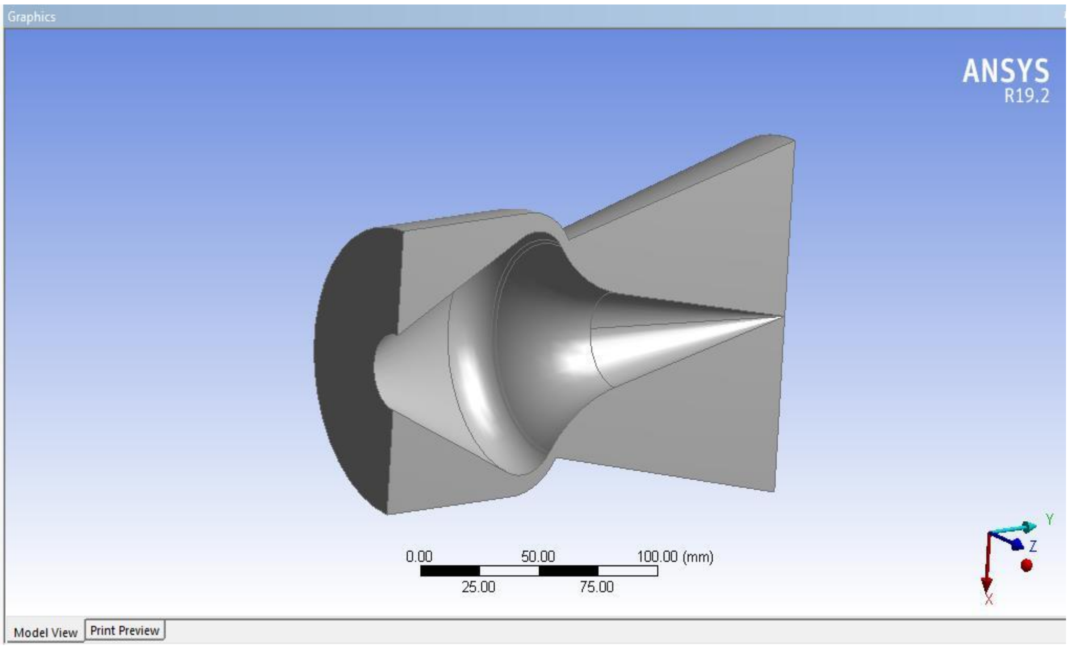


Figure 3 . Geometry of the aerospike bell nozzle without dimples.

IARJSET

ISSN (O) 2393-8021, ISSN (P) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

ISO 3297:2007 Certified ⁜ Impact Factor 7.105 ⁜ Vol. 9, Issue 6, June 2022

DOI: 10.17148/IARJSET.2022.9607

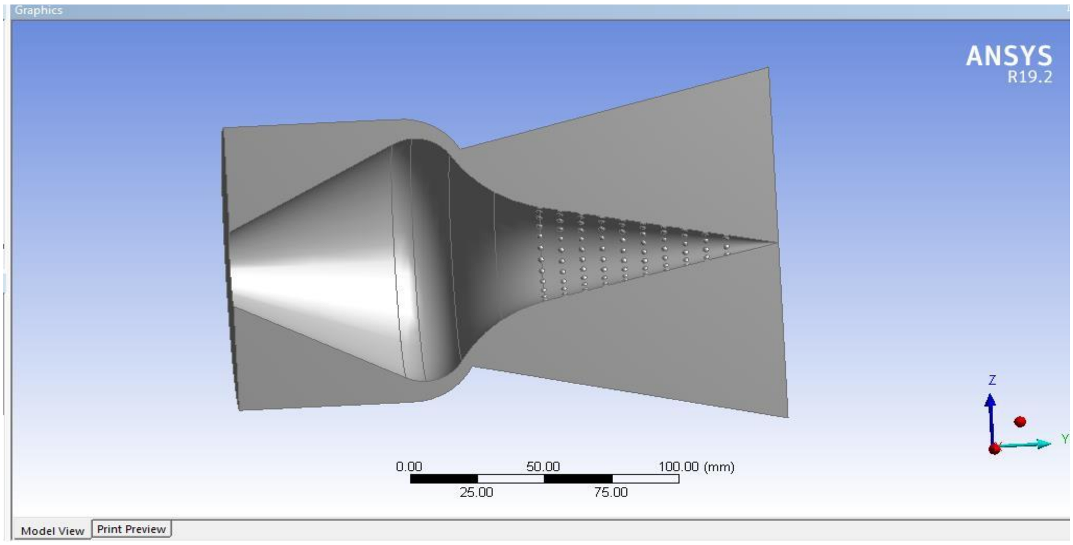


Figure 4. Geometry of the Aero-spike bell nozzle with dimples

MESHING THE MODEL

Meshing is an integral part of the engineering simulation process where complex geometries are divided into simple elements that can be used as discrete local approximations of the larger domain. The mesh influences the accuracy, convergence and speed of the simulation.

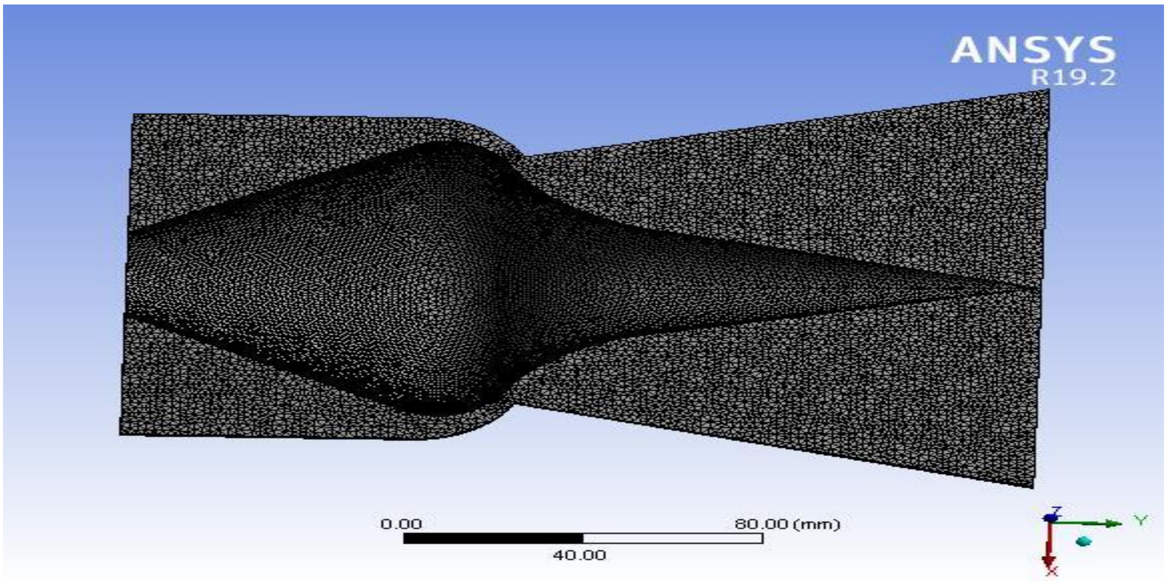


Figure 5. Meshing of the Aero-spike bell nozzle without dimples

Method	Tetrahedrons	Max Element Size	Default (3.0 mm)
Algorithm	Patch Independent	Nodes	871388
Elements	599765		

IARJSET ISSN (O) 2393-8021, ISSN (P) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

ISO 3297:2007 Certified ⁜ Impact Factor 7.105 ⁜ Vol. 9, Issue 6, June 2022

DOI: 10.17148/IARJSET.2022.9607

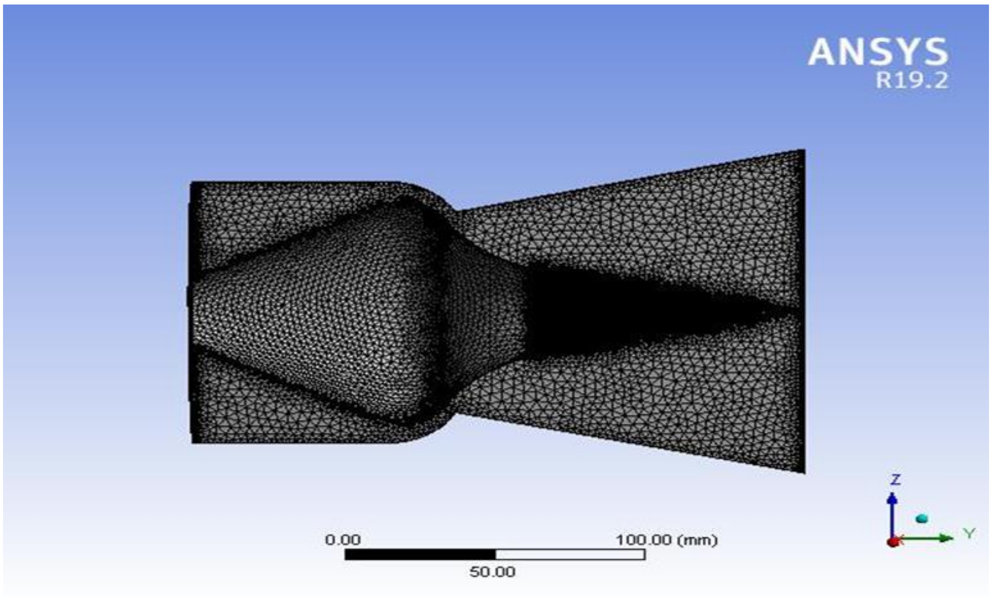


Figure 6. Meshing of the Aero-spike bell nozzle with dimples.

SET-UP THE BOUNDARY CONDITIONS:

Boundary conditions consist of flow inlets and exit boundaries, wall, repeating, and pole boundaries, and internal face boundaries. All the various types of boundary conditions are discussed in the sections that follow.

Inlet velocity: Entry at subsonic speed (M=0.8) Pressure at the inlet: 500 psia

Temperature at the inlet: 1500K

POST-PROCESSING

Post-processing gives users complete insight into their fluid dynamics simulation results. ANSYS CFD-POST, the common post-processor for all ANSYS fluid dynamics products, gives users everything they need to visualize and analyze their results.

SOLUTION

Pressure variation of the nozzle

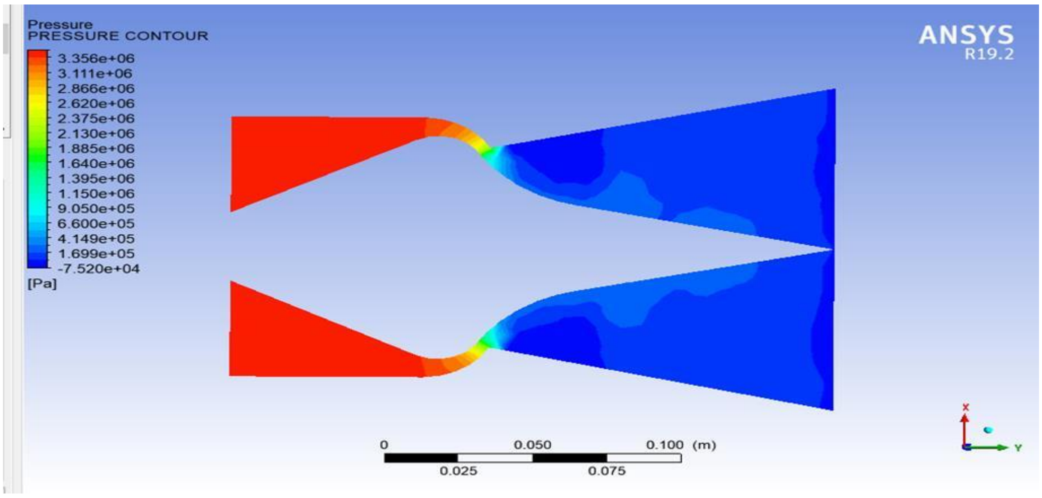


Figure 7 .Pressure plot of Aero-spike Bell nozzle without Dimples.

Minimum pressure in psi	Maximum pressure in psi
- 10.90683504 101010.980683504	24.64190523

International Advanced Research Journal in Science, Engineering and Technology
ISO 3297:2007 Certified ⁝⁝ Impact Factor 7.105 ⁝⁝ Vol. 9, Issue 6, June 2022
DOI: 10.17148/IARJSET.2022.9607

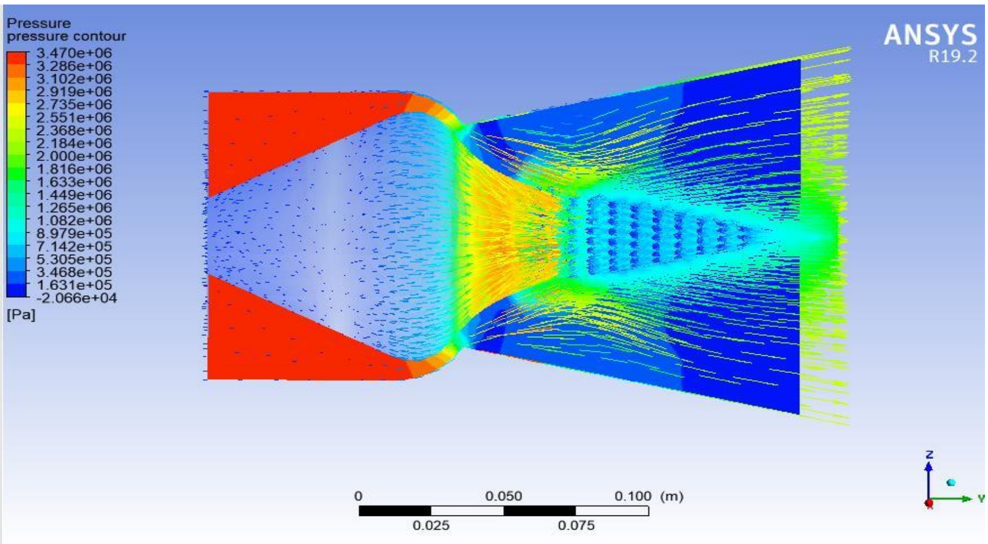


Figure 8 . Pressure plot of Aero-spike Bell nozzle with Dimples.

Minimum Pressure in psi	Maximum Pressure in psi
2.996478882	23.65564887

Temperature variation of the nozzle

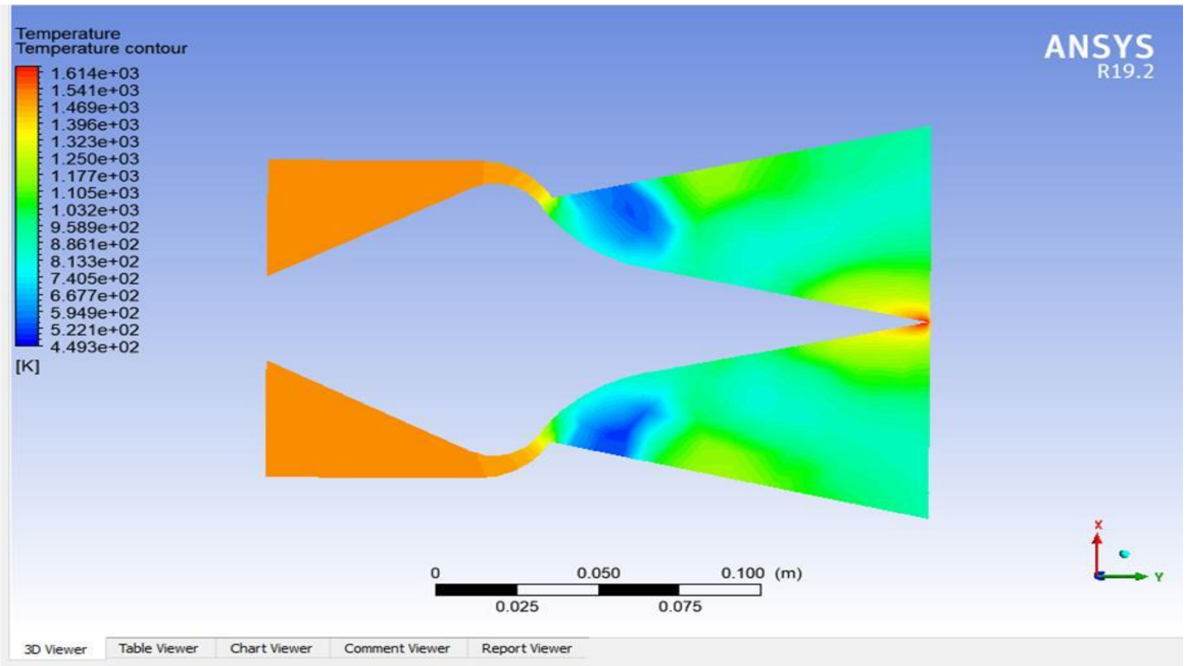


Figure 9 .Temperature plot of Aero-spike Bell nozzle without Dimples

Minimum Temperature in K	Maximum Temperature in K
886.1	958.9

IARJSET

ISSN (O) 2393-8021, ISSN (P) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

ISO 3297:2007 Certified ⁜ Impact Factor 7.105 ⁜ Vol. 9, Issue 6, June 2022

DOI: 10.17148/IARJSET.2022.9607

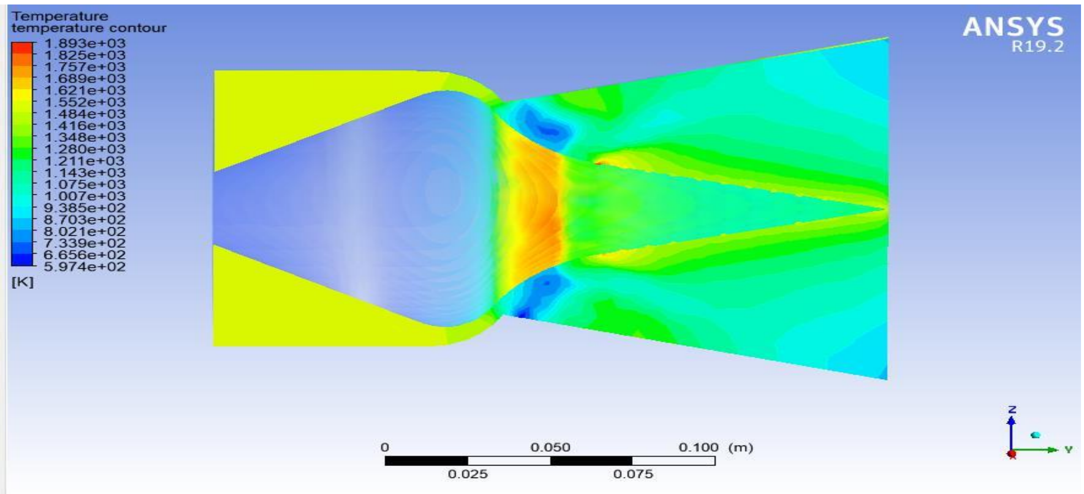


Figure 10.Temperature plot of Aero-spike Bell nozzle with Dimples.

Minimum Temperature in K	Maximum Temperature in K
1075	1143

Velocity variation of the nozzle

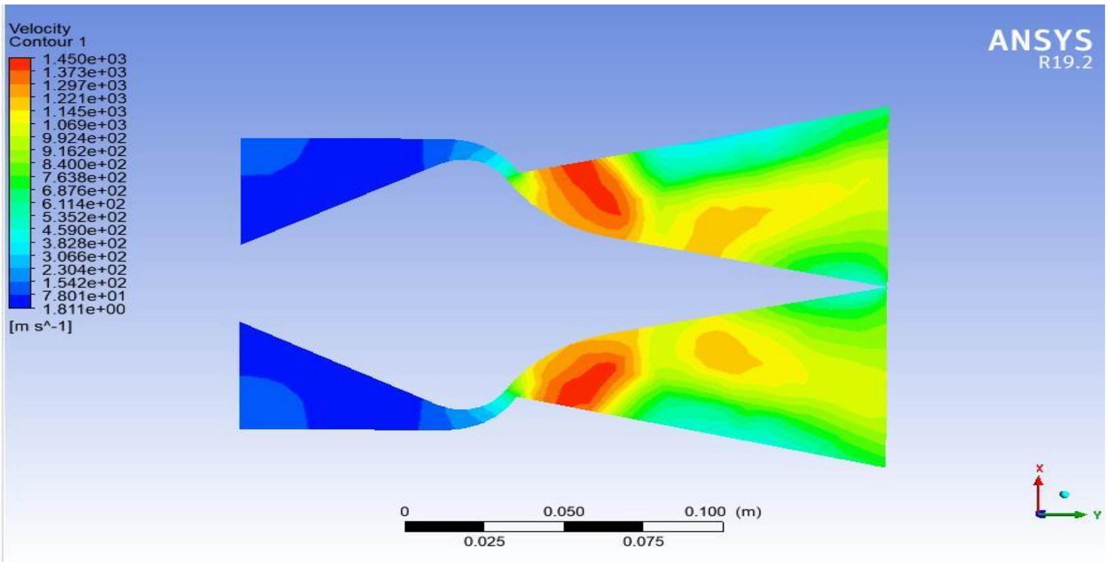


Figure 11.Velocity plot of Aero-spike Bell nozzle without Dimples

Minimum Velocity in m/s	Maximum Velocity in m/s
916.2	992.4

IARJSET
ISSN (O) 2393-8021, ISSN (P) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
ISO 3297:2007 Certified ⋇ Impact Factor 7.105 ⋇ Vol. 9, Issue 6, June 2022
DOI: 10.17148/IARJSET.2022.9607

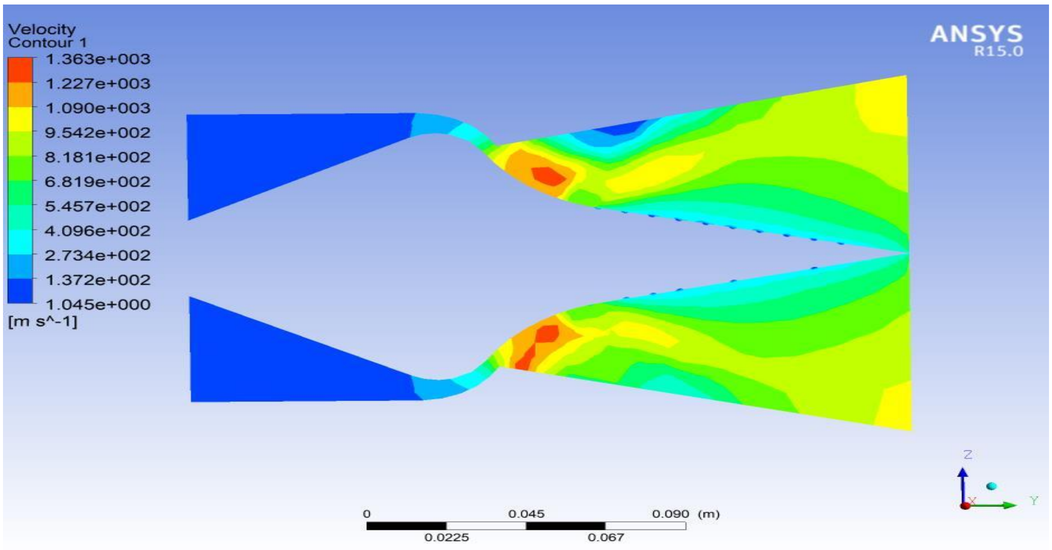
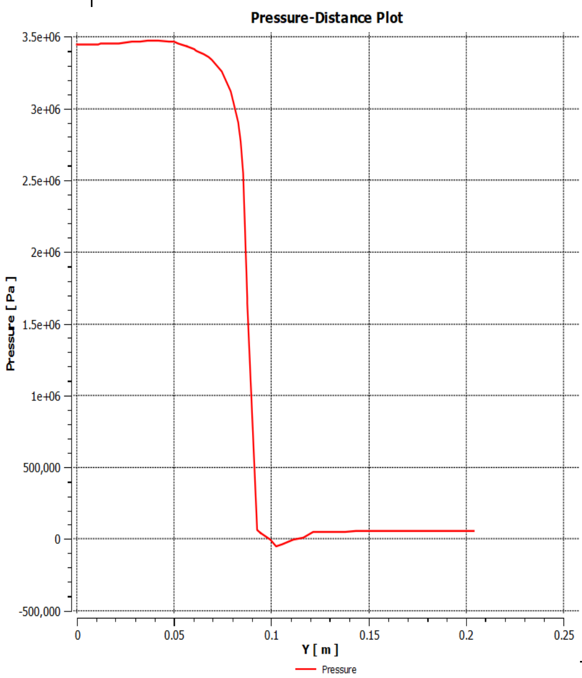


Figure 12. Velocity plot of Aero-spike Bell nozzle with Dimples

Minimum Velocity in m/s	Maximum Velocity in m/s
932.7	1004

5. RESULT & DISCUSSIONS



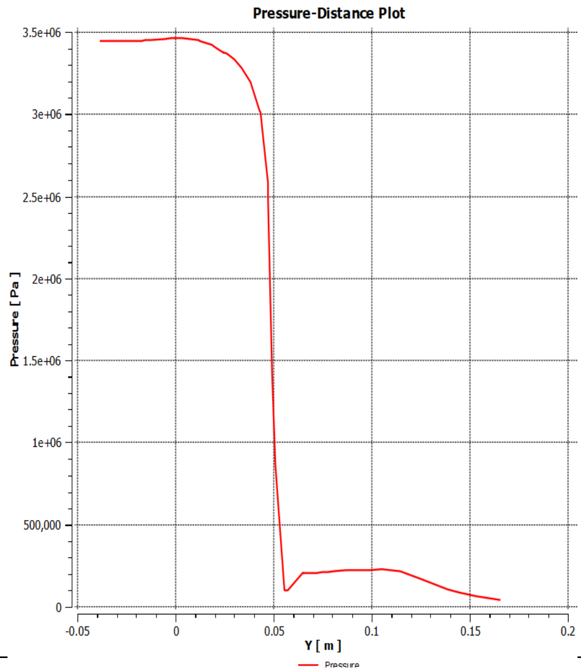


Figure 13: Comparison of pressure of Aero-spike bell nozzle of with and without dimples

The maximum pressure at the exit portion of the Aero-spike bell nozzle without dimples is 24.64190523psi, and the minimum pressure is -10.90683504psi. By Providing the dimples on the spike of the nozzle, a tiny pocket of turbulence or fluid disturbance is created on the surface. This causes the flow to be attached to the surface of the spike, thereby reducing the flow separation in the nozzle. As a result, the pressure at the nozzle outlet is low compared to the nozzle

IARJSET ISSN (O) 2393-8021, ISSN (P) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

ISO 3297:2007 Certified ⁙ Impact Factor 7.105 ⁙ Vol. 9, Issue 6, June 2022

DOI: 10.17148/IARJSET.2022.9607

without dimples. Because of dimples the maximum pressure in the exit portion is 23.65564887psi, and the minimum pressure is -2.996478882psi. Above graph shows that the pressure at the nozzle of with dimples and without dimples vary in small amount resulting in markable amount of pressure.

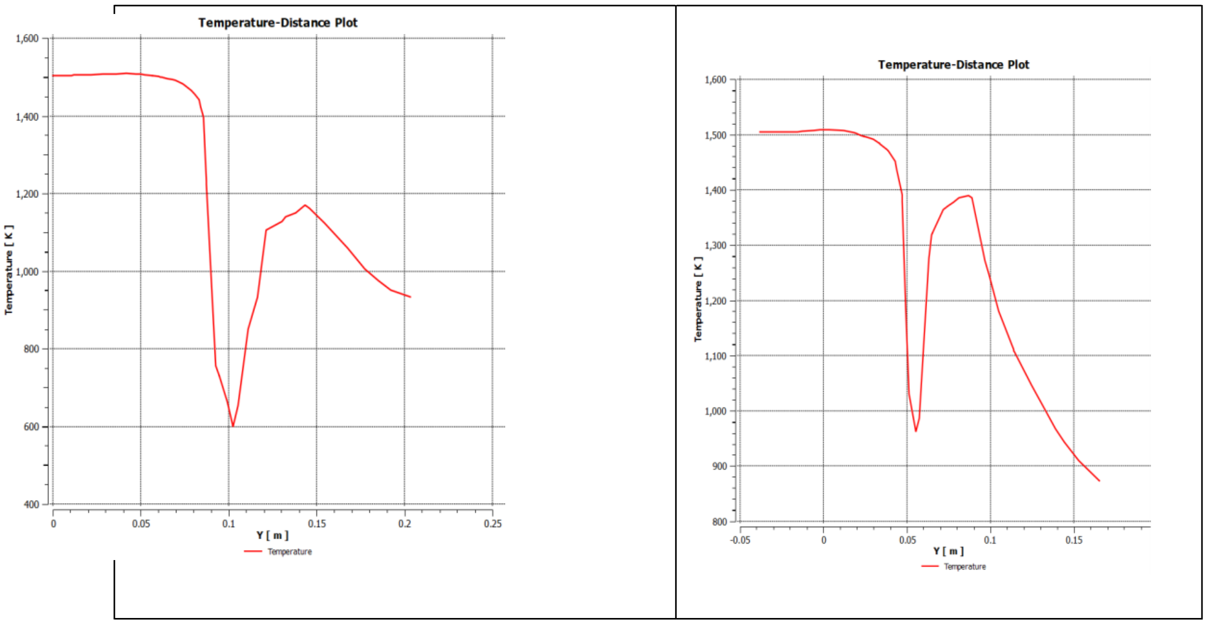


Figure 14: Comparison of temperature of Aero-spike bell nozzle of with and without dimple

The maximum temperature at the exit portion of aerospike bell nozzle without dimples in the is 958.9K and minimum temperature is 886.1K. And the maximum temperature of aerospike nozzle with dimples is 1143K and the minimum temperature is
1075K. This is because the dimples create the turbulence at the region of spike portion, resulting the temperature increases in the spike of the nozzle compared to Aero-spike bell nozzle without dimples. The above graph shows that the temperature at the nozzle of with dimples and without dimples varies greatly resulting in high amount of temperature difference

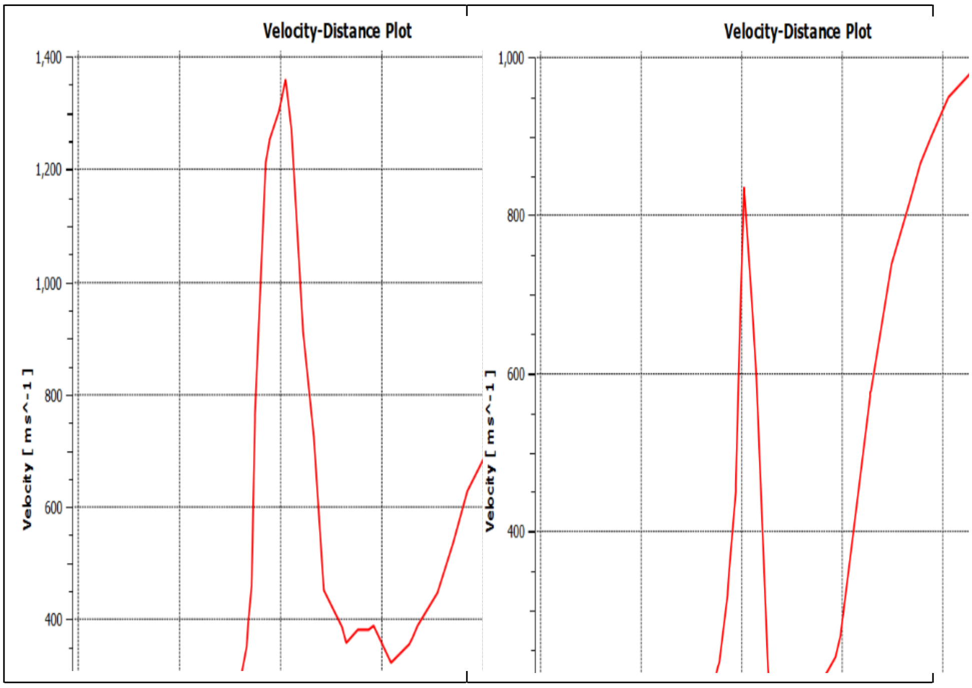


Figure 15: Comparison of velocity of Aero-spike bell nozzle of with and without dimples

DOI: 10.17148/IARJSET.2022.9607

The maximum velocity at the outlet of the Aero-spike bell nozzle without Dimples is 992.4m/s and the minimum velocity is 916.2m/s. The maximum velocity at the exit portion of Aerospike bell nozzle with Dimples is 1004m/s and the minimum velocity is 932.7m/s. This phenomenon is based on the Bernoulli's principle. According to the Bernoulli's principle, the decrease in the fluid's potential energy or pressure of the fluid occurs simultaneously with an increase of the speed of the fluid (V This graph indicates that the velocity at the nozzle of without dimples increase at the throat section but the velocity towards the exit decreases gradually while the velocity at the nozzle of with dimples shows a gradual increase at the throat section but the velocity towards the exit shows increased velocity compared to the nozzle without dimples.

6. CONCLUSION

Design of Aero-spike bell nozzle of with and without dimples has been done successfully by using design software CATIA V5. The analysis of Aero-spike bell nozzle of with and without dimples is analysed using ANSYS FLUENT (19.2) software.

The presence of dimples at the end of the Aero-spike bell nozzle may considerably increase the exit velocity by increasing the turbulence and enables the flow to be attached to the spike of the nozzle. This implies that the velocity produced at the outlet of the aerospike bell nozzle of with dimples is 1004m/s which is higher than the velocity produced at the outlet of the aerospike bell nozzle of without dimples (992.4m/s).

The phenomena is achieved by providing dimples at the exit portion of the Aero-spike bell nozzle, therefore the thrust produced is considerably higher than the normal Aero-spike bell nozzle.

REFERENCES

1. Aerospike nozzle contour design and its performance validation Chang-Hui Wang∗, Yu Liu, Li-Zi Qin403 Teaching and Research Section, Beijing University of Aeronautics and Astronautics, 100083 Beijing, People's Republic of China.
2. Multidisciplinary Approach To Linear Aerospike Nozzle Optimization J. J. Korte*, A.O. Salast, H.J. Dunn?, and N.M. Alexandrov P NASA Langley Research Center, Hampton, Virginia 23681.
3. Development of a Reusable Aerospike Nozzle for Hybrid Rocket Motors Patrick Lemieux1 California Polytechnic State University, San Luis Obispo, CA, 93407.
4. Flight Research of an Aerospike Nozzle Using High Power Solid Rockets Trong T. Bui* and James E. Murray NASA Dryden Flight Research Center, Edwards, CA, 93523 Charles E. Rogers Air Force Flight Test Center, Edwards Air Force Base, CA, 93523 Scott Bartel§ blacksky Corporation, Carlsbad, CA, 92008 and Anthony Cesaroni** and Mike Dennett Cesaroni Technology/Cesaroni Aerospace, Sarasota, FL, 34243.
5. Numerical Analysis Of Aero-Spike Nozzle For Spike Length Optimization Sanoob S N1, Prince Mg2 & Sundar B3 1Research Scholar, MES College of Engineering, Kuttipuram, Malappuram, Kerala, India 2Assistant Professor, MES College of Engineering, Kuttipuram, Malappuram, Kerala, India 3Deputy Head, VSSC, ISRO, Kerala, India.
6. Aerospike Engines for Nanosat and Small Launch Vehicles (NLV/SLV) Eric Besnard* California State University, Long Beach, CA, 90840 and John Garvey† Garvey Spacecraft Corporation, Huntington Beach, CA 92649.
7. Thrust Force Analysis Of Spike Bell Nozzle V.Gopala Deva Kowsik1, Chris Joseph2, M.P.Arun Justin3, R.Sainath4, D.Thanikaivel Murugan5, S.Ilakkiya6 1to4: UG Students, 5&6: Assistant Professor Department of Aeronautical Engineering, Jeppiaar Engineering College, Chennai – 600119.
8. Hybrid propulsion: an overview of the Onera activities JérômeAnthoine and Michel Prévost ONERA – The French Aerospace Lab 31410 Mauzac, France Recent Advances In Hybrid Propulsion1brian Cantwell,∗ArifKarabeyoglu, & David Altman StanfordUniversity, Stanford,California 94305,USAand Space Propulsion Group, Incorporated ,Sunnyvale, California 94085,USA.
9. Numerical Modeling of Pressure drop due to Single - phase Flow of Water and Two - phase Flow of Air - water Mixtures through Thick Orifices- Manmatha K. Roul , Sukanta K. Dash. "Numerical Modeling of Pressure drop due to Single - phase Flow of Water and Two - phase Flow of Air - water Mixtures through Thick Orifices". International Journal of Engineering Trends and Technology (IJETT).
10. Design Optimization and Analysis of Rocket Structure for Aerospace Applications- Anoop Thankachen, Santosh kumar"Design Optimization and Analysis of Rocket Structure for Aerospace Applications", International Journal of Engineering Trends and Technology (IJETT), V24(6),286-291 June 2015. ISSN:2231-5381.