International Journal of Research
Available at
https://edupediapublications.org/journals
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 05 Issue 19
August 2018

Investigation into Effect of Nozzle Divergent Angle in an Elaval Nozzle Design

B.S.P.Pavan, M.Tech., Swarnandhra Institute of Engineering and Technology, Seetharampuram

Dr. P. Sreenivasulu, M.Tech., Ph.D (Professor in Mechanical Engineering), Swarnandhra Institute of Engineering and Technology

Abstract- Nozzle is a device designed that control the rate of flow, speed, direction, mass, shape and pressure of the stream that exhaust from it. Nozzles have variety of shapes and sizes depending on the mission of the rocket. Nozzles is very important for the understanding the performance characteristics of rocket. The proper geometrical design of the nozzle, the exhaust of the propellant gases will be regulated in such a way that max effective rocket velocity can be reached. Convergent divergent nozzle is the most commonly used nozzle. Objective of this work is to determine the optimum Nozzle parameters such as Divergent angle for a rocket nozzle. CFD simulation has been made using ANSYS FLUENT with appropriate Boundary condition.

3-D model of nozzle is designed with Pro-E and UG-NX software. Optimum values have been determined on the basis of Flow parameters obtained in the CFD simulation. CFD is a branch of Fluid Mechanics which rely on numerical methods and algorithms to solve and analyze problem that involves fluid flow. Variation in parameters like velocity, static pressure, turbulence intensity and temperature are being analyzed. Objective of this research is to investigate best suited divergent angle. The phenomena of oblique shock are visualized and it was found that at 15^0,26.6^0 and 27^0 of divergent angle it is completely eliminated from 3 different nozzles. Also intensity of velocity is found to have an increasing trend with increment in divergent angle thereby obtaining an optimum divergent angle.

I. INTRODUCTION

Nozzle is a main component in Rocket is used to convert the Chemical-Thermal Energy generated in the Combustion Chamber into Kinetic energy. The Nozzle converts the low velocity, High Pressure, High Temperature gas in the thrust Chamber into High Velocity gas of lower Pressure and temperature. De Laval Nozzle found that the most efficient conversion occurred when the nozzle first narrowed, increasing the speed of the jet to the speed of sound. Computational Fluid Dynamics (CFD) is an engineering tool that assists experimentation .Its scope is not limited to fluid dynamics.CFD could be applied to any process which involves transport phenomena with it. To solve an engineering problem we can make use of various methods like the analytical method Experimental methods using prototypes. The analytical method is very complicated and difficult. The experimental methods are very costly .If any errors in the design were detected during the prototype testing, another prototype is to be made clarifying all the errors and again tested.

The modern research considering computational fluid dynamics is that it involves software testing and no prototype is need to build during designing stages and hence solution can be obtained faster and at less cost. Computational Fluid Dynamics become a popular tool for solving various problems and the physical aspects are governed by three aspects.

- Mass is conserved.
- Newton's Second law is observed.(Momentum is conserved)
- Energy is conserved.

These factors are expressed in terms of the equation which is either integrals or differential equations. CFD is the art of study of replacing theses integrals or differential equations in terms of discredited algebraic forms which in turn are solved to obtain a number for flow field's values at the discrete point in time or space. The final product of the study of CFD is a collection of the numbers in contrast to closed form analytic solution which is applicable in the practical solution.

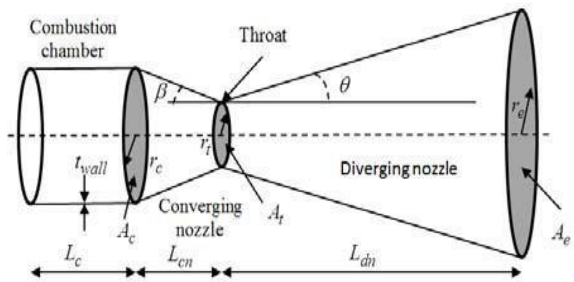


Fig.1: Convergent-divergent nozzle

The inlet Mach number is less than one, Convergent section accelerates it to sonic velocity at the throat and further accelerated to supersonic velocities by the diverging section.

In this project the designing and analysis of CD nozzle geometry is done in the CFD (Computational Fluid Dynamics software). Firstly the design of nozzle is made in UG-NX software and then the nozzle geometry is further analyzed in fluent software in order to analyze the flow inside the CD nozzle and to get the view of the behavior of fluid inside the convergent-divergent section of nozzle.

Project Objective

The objectives of this project are:

1. To obtain the optimized divergent angles in a different delaval nozzles using CFD.
2. To get the optimal divergent angle of anozzlebased on static pressure using CFD.
3. To get the optimal divergent angles of anozzlebased on velocity gradient using CFD.

II. LITERATURE SURVEY

A convergent-divergent nozzle is designed for attaining speeds that are greater than speed of sound. The design of this nozzle is obtained from the area-velocity relation (dA / dV) = -(A/V)(1-M^2) where M is the Mach number (which means the ratio of local speed of flow to the local speed of sound) A is area and V is velocity.

One important point is that to attain supersonic speeds there is a need to maintain favorable pressure ratios across the nozzle.

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 05 Issue 19
August 2018

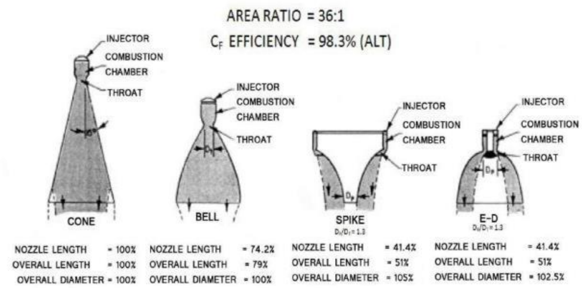


Fig.2: Rocket Nozzle Profiles

III. STUDY

The most popular altitude-compensating nozzle to date is the aerospike nozzle, the origin of which dates back to Rocket dyne in the 1950s. This type of nozzle was designed to allow for better overall performance than conventional nozzle designs.

Nozzle:

A nozzle is a mechanical device of varying cross section which controls the direction and characteristics of the fluid (Air or Water) flowing through it. They are used in rocket engines to expand and accelerate the combustion gases, from burning propellants, so that the exhaust gases exit the nozzle at supersonic or hypersonic velocities.

NEED FOR NEW DESIGN:

The revolution in aerospace propulsion was increased greatly during World War 2. Faster, bigger and more efficient aerospace vehicles were required which led to the birth of Space research organizations like NASA. Speaking about the future, advanced rocket propulsion systems will require exhaust nozzles that perform efficiently over a wide range of ambient operating conditions. Most nozzles either lack this altitude compensating effect or they are extremely difficult to manufacture. In present day, only bell nozzles are used for launch activities. But, these bell nozzles have a major drawback of decreasing in efficiency as altitude increases. This is due to a phenomenon causing loss of thrust in the nozzle at higher altitudes called as separation of the combustion gases.

Working Principle of nozzle:

The function of a rocket nozzle is to direct all gases, generated in the combustion chamber of the engine and accelerated by the throat, out of the nozzle. The key feature of the aerospike engine is that, as the launch vehicle ascends during its trajectory, the decreasing ambient pressure allows the effective nozzle area ratio of the engine to increase. An aerospike nozzle is often referred to as an altitude-compensating nozzle, because of its specific design capability of maintaining aerodynamic efficiency as altitude increases and thus throughout the entire trajectory. At the outer cowl lip, the gas expands to the atmospheric pressure immediately, and then causes serious expansion waves propagating inward at an angle through the gas stream.

Flow around Nozzle:

The main advantage to the annular aerospike nozzle design (both full length and truncated spike) is its altitude compensation ability below or at its design altitude. More specifically, the aerospike will not suffer from the same overexpansion losses a bell nozzle suffers and can operate near optimally, giving the highest possible performance at every altitude up to its design altitude. Above the design altitude, the aerospike behaves much like a conventional bell nozzle.

Advantages and Disadvantages of Using an Nozzle:

The aerospike nozzle has 90% overall better performance than the conventional bell shaped nozzle. The efficiency at low altitudes is much higher because the atmospheric pressure restricts the expansion of the exhaust gas. A vehicle using an aerospike nozzle also saves 25-30% more fuel at low altitudes. At high altitudes, the aerospike nozzle is able to expand the engine exhaust to a larger effective nozzle area ratio. An aerospike nozzle with an expansion ratio of

IV. RESEARCH METHODOLOGY

The methodology of this project starts with the past literature with relevant journals to identify the exact problem identification in the muffler area. Problem identification has been initiated with the past journals study to minimize the undesired effects and maximize the performance. So, based on the literature we have found out the back pressure as a key to improving the performance with their different operating pressures. Next level moves to frame an objective of this project based on the problem identification, the objective has been framed as to improve the performance by varying the angles. shows the various design parameters such as length, height, width, mean and with the help of these different design values the full model has been created by using COMSOL Multiphysics software.

RESERCH APPROACH:

Rapid progress in aerospace technologies requires a constant interplay between analysis, ground testing, and flight testing activities. All three of these activities are equally important. Testing, both on the ground and in flight, keeps analysis well-grounded and relevant, while analysis provides the foundation and explains flight and ground test results [3]. In this project I shall draw the drawings of nozzle shape in CFD and then we will got the different results in terms of fluid analysis, but here we shall concentrate on pressure only for further work.

METHODOLOGY:

Input parameters for this work are basically Nozzle geometry (contours), weight, thickness of the contours, various constrains like Area ratio, Length of Nozzle. And output is nozzle profile. Generally after the study related to this topic gives the basic shape of nozzle which is parabolic. And we will do study on parabolic contour nozzle by taking the parabola is the basic shape and finding all the results related to it. After that will change the contour shape in different manner's to comparing study between it. Those different nozzle contour shapes are cubical contour profile, semi cubical parabolic profile. After that finding best results from these.

V. DESIGN METHODOLOGY

After getting familiarized with the concepts of the nozzle, let us now get into detail of the design procedure. Therefore this chapter gives a main focus on the design procedure of the different kinds of nozzles. This chapter relates to the application of the above mentioned thermodynamic relations and the parameters required to design nozzle. It mainly consists of designing of a Conical and Contour nozzle.

Design of complete Nozzle:

Supersonic nozzles are generally specified in terms of the cross sectional area of finaluniform flow A and the final mach number M. The nozzle-throat area is obtained by the 1D flow equation, the shortest nozzles that may be designed by the method of reported are those without a straight-walled section. The straightening part immediately follows with the expanding part. The purpose of method of characteristics is to illustrate the design of a supersonic nozzle by the method of computation with the weak waves.

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 05 Issue 19
August 2018

Modeling:

The Geometry of the nozzle was created using Pro-E

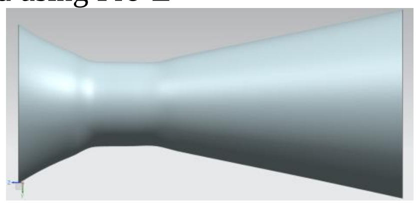


Fig.3: Design Model-1

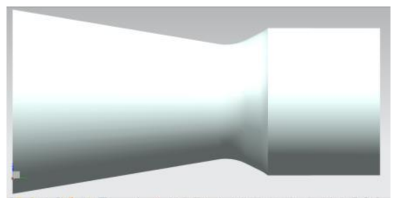


Fig.4: Design Model-2

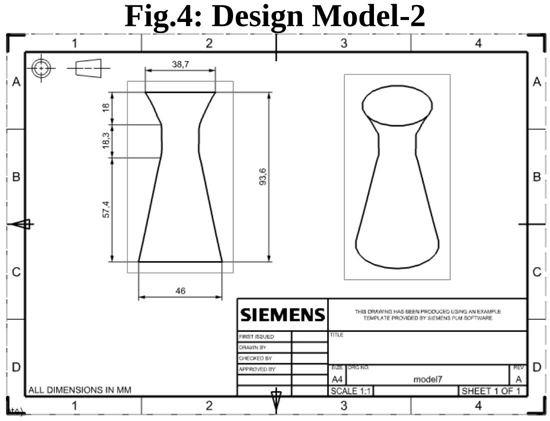


Fig.5: Drafting-1

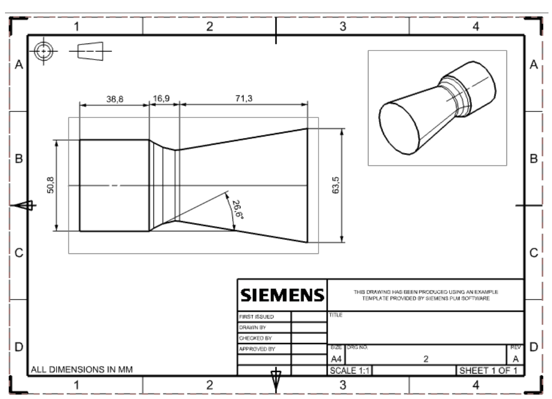


Fig.6:Drafting-2

Meshing Pattern

Mesh generation also called as a „grid generation" is an important part of CFD. Meshing for CFD is pivotal in achieving realistic renderings and physical simulations. Realistic rendering and high precision simulations depend on the quality of meshing for CFD.

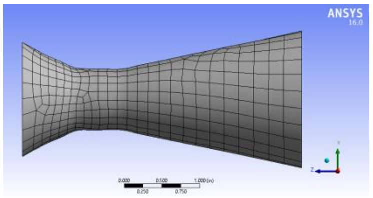


Fig.7: Mesh Model-1

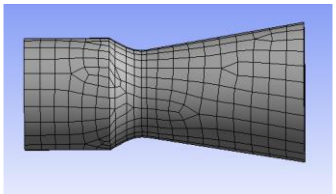


Fig.8: Mesh Model-2

Boundary Conditions

Specification of the boundary zones has to be done in WORKBENCH only, as there is no possibility to specify the boundary zones in FLUENT. Therefore proper care has to be taken while defining the boundary conditions in WORKBENCH. With all the zones defined properly the mesh is exported to the solver. The solver used in this problem is ANSYS FLUENT. The exported mesh file is read in Fluent for solving the problem.

No	Input for model-1 and model-2	Value
1	Inlet pressure	1.6 bar
2	Guagepressure(norm al to boundary)	15.89 bar
3	Outlet pressure	0.0001 bar
4	Tem perature	350 k
5	Divergent Angle for model-1 and model-2	27 and 26.6 degrees

No	Input	Value
1	Inlet Width	1.000 (m)
2	Throat Width	0.304 (m)
3	Exit Width	0.861 (m)
4	Throat Radius Curvature	0.228 (m)
5	Convergent Length	0.640 (m)
6	Convergent Angle	30°
7	Divergent Angle	5°, 15° Respectively
8	Total Pressure	45 (bar)
9	Total Temperature	3400 (k)
10	Mass Flow Rate	826 kg/s

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 05 Issue 19
August 2018

No.	Steps	Process
1	Problem statement	Information about the flow and working parameters
2	Mathematical model	Generate nozzle geometry
3	Mesh generation	Nodes/cells, time instants
4	Space discretization	Coupled ODE/DAE systems
5	Time discretization	Algebraic system Ax=b
6	Iterative solver	Discrete function values
7	CFD software	Implementation, debugging
8	Simulation run	Parameters, stopping criteria
9	Post processing	Visualization, analysis of data
10	Verification	Model validation / adjustment
11	Saving case and data	Save all the obtain data
12	Comparing	Comparing the outcome values with real practical values

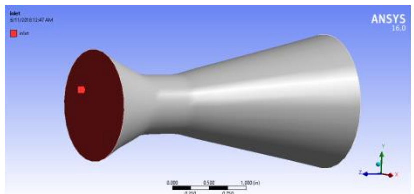


Fig.9: Design-1(Inlet)

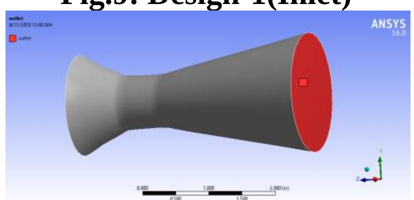


Fig.10: Design-1(Outlet)

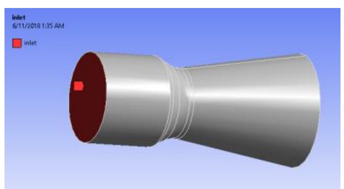


Fig.11: Design-2(Inlet)

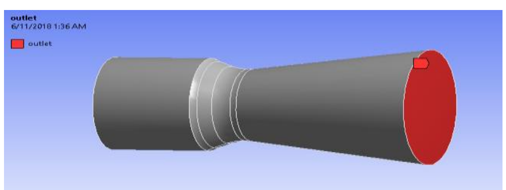


Fig.12: Design-2(Outlet)

Solving:

FLUENT analysis is carried out on nozzle at different meshing conditions.

VI. RESULTS AND DISCUSSIONS

MODEL-1

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 05 Issue 19
August 2018

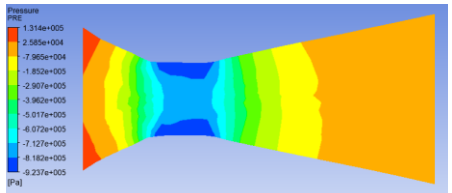


Fig.13: Pressure of Model-1

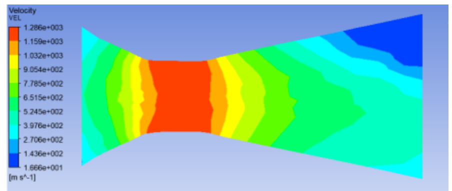


Fig.14: Velocity of Model-1

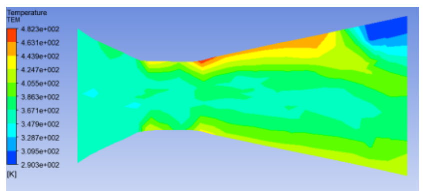


Fig.15: Temperature of Model-1

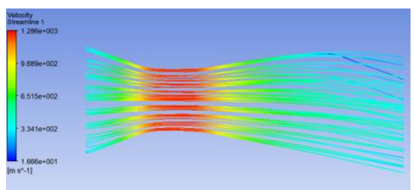


Fig.16: Stream Line-Velocity of Model-1

MODEL-2

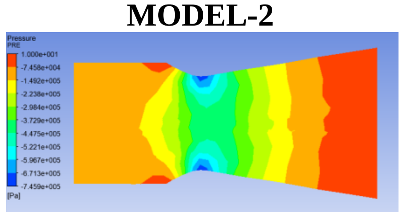


Fig.17: Pressure of Model-2

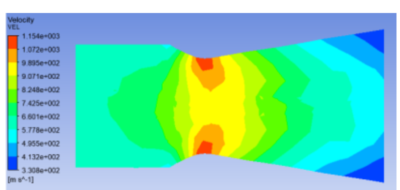


Fig.18: Velocity of Model-2

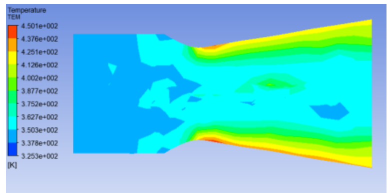


Fig.19: Temperature of Model-2

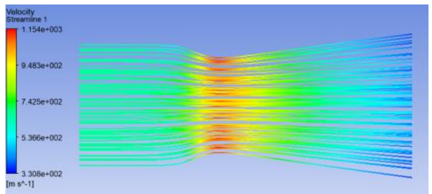
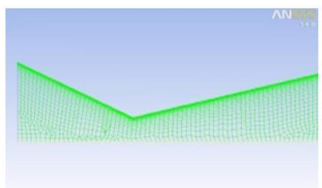


Fig.20: Stream Line- Velocity of Model-2

Meshing Pattern for model-3:

Mesh generation also called as a „grid generation‟ is an important part of CFD. Meshing for CFD, is pivotal in achieving realistic renderings and physical simulations. Realistic rendering and high precision simulations depend on the quality of meshing for CFD.



Mesh generation

Velocity magnitude at divergent angle of 5°:

It can be infer from velocity contour of the nozzle that the divergent angle of 20° shows the formation of oblique shock in the divergent section. As it is infer in Figure , Across the shock, the velocity suddenly drops from 1.38e+03 m/s to 1.18e+03 m/s. After this the velocity of flow again increases. The positions where the shock occurs can be determined from the Velocity magnitude Vs position plot.

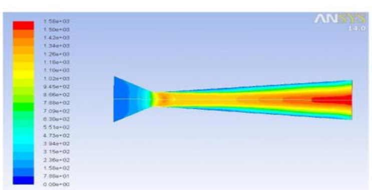


Fig.21: Contour of velocity at divergent angle of 5°

You can easily visualize effect of viscous near the wall of nozzle which resulted in to lower velocity region depicted by string of blue shade. It is found that shock occurs at the position 0.25m from the throat section. The velocity magnitude is found to increase as we move from inlet to exit. The velocity at the inlet is 0.98e+02 m/s (sub-sonic). At the throat section the velocity varies from 4.32e+02 m/s to 1.35e+03 m/s. The velocity at the exit was found to be 1.592e+03 m/s (super-sonic). From this plot it is clearly observed that the velocity is increased as goes from inlet to outlet section of nozzle.

International Journal of Research
Available at
https://edupediapublications.org/journals
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 05 Issue 19
August 2018

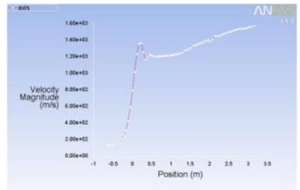


Fig.22: Velocity magnitude VS Position plot at 5° of divergent angle

Static Pressure at divergent angle of 5°:

Static pressure is the pressure that is exerted by a fluid. Specifically, it is the pressure measured when the fluid is still, or at rest. Contours of static pressure in convergent, throat, divergent and exit section is shown. The below figure reveals the fact that the gas gets expanded in the nozzle exit. The static pressure in the inlet is observed to be 8.52 e+06 Pa and as we move towards the throat there is a decrease and the value at the throat is found out to be 5.88e+06 Pa.

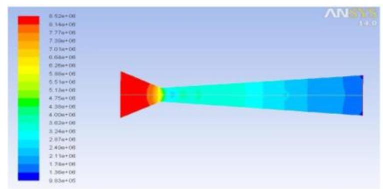


Fig.23: Contour of static pressure at divergent angle of 5°

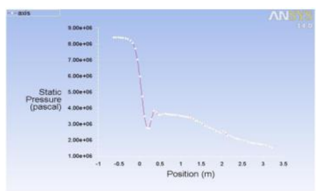


Fig.24 Static Pressure VS Position plot at optimum of divergent angle

Static Temperature at divergent angle of 5°:

The temperature almost remains a constant from the inlet up to the throat after which it tends to decrease. At the inlet and the throat the temperature is 3.32e+03 K. After the throat, the temperature decreases till the exit. As we move from the centre vertically upwards and downward temperature increased.

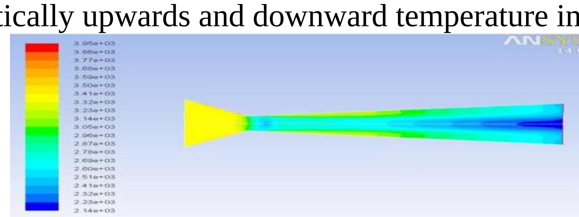
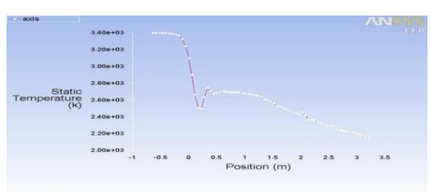


Fig.25: Contour of Static Temperature at 5° of divergent angle



e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 05 Issue 19
August 2018

Fig.26: Static Temperature VS Position plot at 5° of divergent angle

Turbulent intensity at divergent angle of 5°:

The turbulent intensity contour shows that the inlet section has very low turbulence of the value 1.01e+00% and it increases towards the nozzle. Just as the divergent section starts the contour show very high value of turbulent intensity which is due to the sudden expansion of the flow into the divergent section. Here in this case the flow in the divergent section is highly turbulent because of the formation of two shocks inside the section. From the contour the region of shock has a turbulence intensity of 2.24e+01% and then it drops to 1.56e+01% at the exit section.

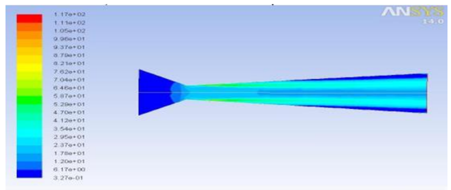


Fig.27: Contour of turbulent intensity at divergent angle of 5°

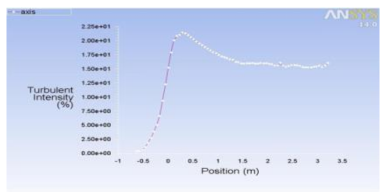


Fig.28: Turbulent intensity VS Position plot at 5° of divergent angle

Velocity magnitude at divergent angle of 15°:

The contour of velocity magnitude of convergent-divergent nozzle when the divergent angle is made 15° is shown in the figure . It is clear that the shock has been completely eliminated from the divergent section of the nozzle. The inlet section has a velocity of 1.11e+03 m/s and it increases to a value of 3.61e+01m/s at the throat section.

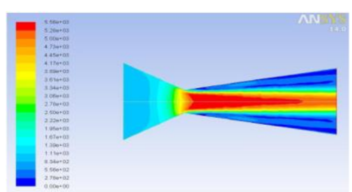


Fig.29: Contour of Velocity magnitude at divergent angle of 15°

Static Pressure at divergent angle of 15°:

The below Figure , shows that the gas gets over expanded in the nozzle exit and oblique shock is prevented. The static pressure in the inlet is observed to be 1.80e+07 Pa and the value at the throat is found out to be 5.40e+06 Pa. At the exit the pressure is experienced to be 1.25e+05 Pa. Right from the inlet to the throat to the exit the static pressure tends to decrease and remains constant till exit section. There is a considerable decrease observed after the throat to the exit where there is a large drop in the static pressure. As compared to the previous case of divergent angle of 15 degree there is a change in the exit static pressure value because of change in a geometry of nozzle.

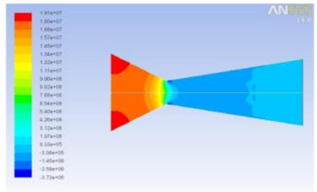


Fig.30: Contour of Static Pressure at divergent angle of 15

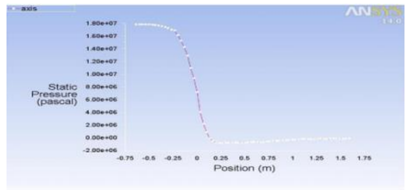


Fig.31: Static Pressure VS Position plot at 15° of divergent angle

Static Temperature at divergent angle of 15°:

The static temperature almost remains a constant in the inlet up to the throat. Increase is observed after some distance from the throat towards the exit as seen in the below Figure:, Near the walls the temperature decreases to 2.65e+03 K. In the inlet and the throat the temperature is 3.32e+03 K. After the throat, the temperature increases to 4.06e+03 K at the exit. At the exit, moving vertically upward there is variation. The maximum value is not attained at the centre but at some distance from the centre. At the centre it is 3.59e+03 K and at the wall it is 2.65e+03 K.

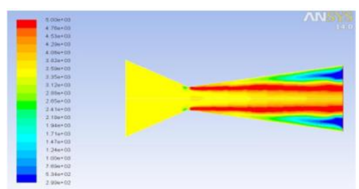


Fig.32: Contour of Static Temperature at divergent angle of 15°

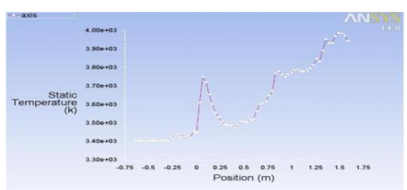


Fig.33: Static Temperature VS Position plot at 15° divergent angle

Turbulent intensity at divergent angle of 15°:

The turbulent intensity contour shows that at the inlet the turbulent intensity is 1.01e+00%. It has increased to 2.40e+01% at the throat section. Since the divergent angle is higher, the sudden expansion has caused the turbulence at the beginning of the divergent section. It is also seen that the increasing velocity towards the exit section also has caused a turbulence of flow towards the exit section. The value of turbulence is found to be 2.25e+01% in this region.

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 05 Issue 19
August 2018

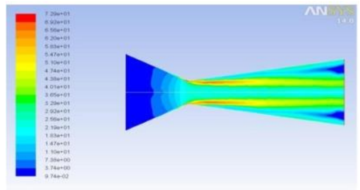


Fig.34: Turbulent intensity at 15° divergent angle

NO	DIVERGENT ANGLE (degrees)	VELOCITY (m/s)	PRESSURE (Pa)	TEMPERATURE (KELVIN)
1	27	1.286e+003	1.314e+005	4.823e+002
2	26.6	1.154e+003	1.154e+001	4.501e+002

Table 1: Influence of variation in divergent angle on different flow property is shown in below tables for model-1 and model- 2 nozzles.

Properties at throat section:

No.	Divergent Angle (degrees)	Velocity magnitude (m/s)	Static pressure (Pascal)	Turbulent Intensity (%)	Static Temp. (K)
1	5	1.35e+03	5.88e+06	2.24e+01	3.32e+03
2	10	2.5e+03	5.62e+06	2.30e+01	3.25e+03
3	15	3.61+01	5.40e+06	2.40e+01	3.02e+03

Table 2 : Values at throat section

Properties at exit section:

No.	Divergent Angle (degrees)	Velocity magnitude (m/s)	Static pressure (Pascal)	Turbulent intensity (%)	Static Temp. (K)
1	5	1.592e+03	1.36e+05	1.56e+01	2.18e+03
2	10	3.06e+03	1.20e+05	2.01e+01	2.96e+03
3	15	5.28e+03	1.26e+05	2.25e+01	4.06e+03

Table 3 : Values at exit section

VII. CONCLUSION

Above results were found after conducting this investigation. It was observed that the velocity at outlet of **model-1nozzle** is high when the divergent angle was **27°**. It is observed that the velocity from the nozzle outlet of **model-2** is high when divergent angle increase to **26.6°** and this could be considered as a good design for the nozzle.

For **model-3 nozzle** it was observed that oblique shocks are formed during flow through the nozzle, when the divergent angle was 5°. It is observed that the shock is completely eliminated from the nozzle when divergent angle increase to **15°** and this could be considered as a good design for the nozzle. At the exit section velocity magnitude is found to be increase with increment in divergent angle. similarly, at throat section velocity magnitude goes on when divergent angle increased. The static pressure decrease with increasing divergent angle. The efficiency of de Laval nozzle increase as we increases divergent angle of nozzle up to certain limit.

We conclude that the divergent angle 15^0 is suitable for high thrust of the de laval nozzle.

REFERENCES

[1] Varun, R.; Sundararajan,T.; Usha,R.; Srinivasan,k.; Interaction between particle-laden under expanded twin supersonic jets, Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering 2010 224: 1005.

[2] Pandey,K.M.; Singh, A.P.; CFD Analysis of Conical Nozzle for Mach 3 at Various Angles of Divergence with Fluent Software, International Journal of Chemical Engineering and Applications, Vol. 1, No. 2, August 2010, ISSN: 2010-0221.

[3] Natta, Pardhasaradhi.; Kumar, V.Ranjith.; Rao, Dr. Y.V. Hanumantha.; Flow Analysis of Rocket Nozzle Using Computational Fluid Dynamics (Cfd), International Journal of Engineering Research and Applications (IJERA), ISSN: 2248-9622,Vol. 2, Issue 5, September- October 2012, pp.1226-1235.

[4] K.M. Pandey, Member IACSIT and A.P. Singh. K.M.Pandey, Member, IACSIT and S.K.YadavK.M.Pandey and S.K.Yadav, ―CFD Analysis of a Rocket Nozzle with Two Inlets at Mach2.1, Journal of Environmental Research and Development, Vol 5, No 2, 2010, pp- 308-321.

[5] Shigeru Aso, ArifNur Hakim, Shingo Miyamoto, Kei Inoue and Yasuhiro Tani“ Fundamental study of supersonic combustion in pure air flow with use of shock tunnel” Department of Aeronautics and Astronautics, Kyushu University, Japan , Acta Astronautica 57 (2005) 384 – 389.

[6] P. Padmanathan, Dr. S. Vaidyanathan, Computational Analysis of Shockwave in Convergent Divergent Nozzle, International Journal of Engineering Research and Applications (IJERA), ISSN: 2248-9622 , Vol. 2, Issue 2,Mar-Apr 2012, pp.1597-1605.

[7] A damson, T.C., Jr., and Nicholls., J.A., “On the structure of jets from Highly under expanded Nozzles into Still Air,” Journal of the Aerospace Sciences, Vol.26, No.1, Jan 1959, pp. 16-24.

[8] Lewis, C. H., Jr., and Carlson, D. J., “Normal Shock Location in under expanded Gas and Gas particle Jets,” AIAA Journal, Vol 2, No.4, April 1964, pp. 776-777.