#### PAPER • OPEN ACCESS

# Design Optimization of Nozzle and Second Throat Diffuser System for High Altitude Test using CFD

To cite this article: Kumar K Naveen et al 2019 J. Phys.: Conf. Ser. 1355 012012

View the article online for updates and enhancements.

# You may also like

- Initial studies of the directional reflectance changes in pressed and sintered PTFE diffusers following exposure to contamination and ionizing radiation G T Georgiev, J J Butler, K J Thome et al.
- <u>Toward Space-like Photometric Precision</u> from the Ground with Beam-shaping <u>Diffusers</u> Gudmundur Stefansson, Suvrath Mahadevan, Leslie Hebb et al.
- <u>Study of the vortex based virtual valve</u> <u>micropump</u> Tao Wang, Jian He, Chunquan An et al.





DISCOVER how sustainability intersects with electrochemistry & solid state science research



This content was downloaded from IP address 3.141.200.180 on 06/05/2024 at 04:29

# **Design Optimization of Nozzle and Second Throat Diffuser** System for High Altitude Test using CFD

# Naveen Kumar K\*, Vijayanandh R\*, Ramesh M

Department of Aeronautical Engineering, Kumaraguru College of Technology, Coimbatore Tamil Nadu, India - 641049

\* Corresponding author E-mail: naveenkumar.k.aeu@kct.ac.in, vijavanandh.raja@gmail.com

Abstract. Nozzles are used to produce thrust in airplanes, rockets, and even satellites. Different designs are employed for various applications. For propelling satellites in space, High Area Ratio supersonic nozzles (A.R. > 100) are used. During ground tests of these nozzles, it is found that a shock wave is formed and flow tends to separate in divergent section. A second throat ejector diffuser system (STED) is employed to simulate ambient vacuum conditions at high altitudes, in sea level, around the nozzle. This vacuum level is maintained throughout the test times and is used for smooth expansion of the gas across the nozzle without flow separation. The Shockwaves formed in the diffuser help in maintaining the desired lowpressure levels and reduce the flow velocity and increase the pressure. A High Area Ratio nozzle (A.R.= 114) and a Second Throat Diffuser are designed and modeled together. The flow analysis is carried out using CFD and the pressure, velocity and temperature variations in the nozzle and diffuser are studied. The diffuser design is then optimized, by varying different parameters of the diffuser, and analysis is carried out for each design in order to obtain full flow in the nozzle and the diffuser. The design of the diffuser must be such that the flow in the nozzle must be stable and should provide a suitable pressure recovery with the reduction in flow velocity. After the diffuser design is finalized, performance analysis of the diffuser is carried out, along with the nozzle, in order to achieve the working conditions of the diffuser.

Keywords: Aerodynamics; Nozzle; Area Ratio; Mach Number; Optimization; Rocket;

#### 1. Design of Nozzle and Second Throat Diffuser

#### 1.1. Design of Nozzle

#### 1.1.1. High A.R. nozzles

A nozzle is used to produce thrust by converting pressure energy into kinetic energy. The highpressure gases from the combustion chamber expand in the nozzle providing the required thrust for any flying body such as aircraft, rockets or missiles. As the pressure decreases the flow velocity inside the nozzle increases. The nozzle accelerates the gas through its expansion section. There are different types of the nozzle that are used depending on the exit velocity the nozzle shape varies. For subsonic

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI. Published under licence by IOP Publishing Ltd 1

International Conference on Aerospace and Me	echanical Engineering, ICA	AME'18	IOP Publishing
Journal of Physics: Conference Series	1355 (2019) 012012	doi:10.1088/1742-	-6596/1355/1/012012

operation, the nozzles have a convergent portion only from the combustion chamber. For supersonic exit velocities, a convergent-divergent nozzle is used.

High Area Ratio nozzles are nozzles having an A.R. above 100, which employed as thrusters for satellites in space. In space, the satellite needs to be carried to the desired altitude and due to atmospheric disturbances, the satellites tend to shift from its desired orbit. For propelling the satellite and rectifying any shift in its orbit, small thrusters are designed for operation at high altitudes. The thrust provided by the thrusters are very small and used for Attitude and Orbit Control of Satellites (AOCS), and form a crucial system for control of satellites for correcting its orbit. The thrusters are located in different positions of the satellite and are used only in space. In which there are two types of high A.R nozzles used, which are the main engine and steering engine nozzle. The Main engine nozzle, having A.R. 114, is considered for analysis in this paper [2]. The High A.R. nozzles need to be tested in the ground level before it can be used on satellites. However, while performing a ground level test, it is found that a shock wave is formed and complete expansion of the gases is not obtained in the divergent portion due to the high A.R. and high back pressures at the exit in sea level. To obtain full flow in the entire nozzle low-pressure atmosphere should be created at the nozzle exit while testing it. This low-pressure atmosphere replicates the vacuum conditions around the nozzle in space and also enables complete flow expansion of the gases inside the nozzle [3].

#### 1.1.2. Design methodologies used

Bell nozzles are the commonly used nozzles as they are more advantageous than the other type of nozzles, which provides smooth expansion, maximum thrust along the axis and gives the required Mach number at a length less than that of the conical nozzle. Length is of major importance in the Aerospace industry so the bell nozzle is the optimum shape preferred for the supersonic applications. Unlike a conical nozzle where the deflection angle is 150 and the line connecting the throat and the exit cross sections, an arc connects the throat and exit area of the bell nozzle and the deflection angle varies at each point of the bell nozzle.

G.V. Rao method is employed for designing minimum length nozzles. In this method, the divergence angle at the inflection point and at the exit are determined using Prandtl Meyer flow relations and compressible flow equations. As a preliminary step the nozzle exit Mach number should be calculated using the Area-Mach number relation. The length of the bell nozzle is within 60% to 80% of that of the conical nozzle depending on the amount of expansion needed.

After the length is determined the arc must be drawn from the throat to the exit, which is completed by determining the throat radius and drawing a curve to the inflection point. The initial curve is plotted such that at the throat the slope is zero and increases gradually till the inflection point. At the inflection point, there is a sharp increase in the slope which is equal to the initial diverging angle. From this point, a series of points are plotted with each point corresponding to a bigger A.R. corresponding to a higher Mach number. The series of points are plotted until the exit point is reached which should be the same as the required exit radius. There are no formulae to calculate the curve in the converging section. Hence a suitable curve with a gradual decrease in slope is considered [6].

1.1.3.	Nozzle Design	Conditions and	Calculated Values	
--------	---------------	----------------	-------------------	--

The conditions that are used for designing	the high A.R.	main engine nozzle are:
Туре	=	Bell/Contoured nozzle
A.R.	=	114
Chamber Pressure	=	33.86 bar
Chamber Temperature	=	3472 K
Mass Flow rate	=	8.7 kg/s
Length of Chamber	=	0.111 m

International Conference on Aerospace and Me	chanical Engineering, ICA	AME'18	IOP Publishing
Journal of Physics: Conference Series	1355 (2019) 012012	doi:10.1088/1742	-6596/1355/1/012012

Initial expansion angle	=	32 <sup>o</sup>
Final expansion angle	=	9 <sup>0</sup>

From the above conditions [7] the other nozzle parameters are calculated using compressible flow relations: The contour points of the designed nozzle have been plotted using commercial software package ICEM-CFD, which is used for designing and meshing. An arc is drawn connecting all the points. The subsonic portion is drawn using straight lines. The nozzle has meshed and the boundary conditions are specified as Pressure Inlet, Pressure Outlet, Axis, and Wall. Type of Mesh used here is Quadrilateral and the number of cells is 7200, a typical discretized view of the nozzle has been shown in figure 1.



Figure 1. Nozzle contour after meshing

# 1.2. Diffuser Design Conditions and Calculated values

The location of the diffuser is aft of the nozzle so the inlet parameters are considered as the exit conditions of the nozzle. The throat of the nozzle is considered as the first throat and the throat of the diffuser is considered as the second throat. The diffuser design conditions [8] are as follows:

Inlet Pressure	=	10 mbar
Minimum Exit Pressure	=	40 mbar
Inlet Temperature	=	481 K
Net mass flow rate	=	8.7 kg/s
Minimum Throat Diameter	=	0.8 m
Minimum Throat Length	=	80 m
Converging section angle	=	4 <sup>0</sup>
Diverging section angle	=	$7^{\mathrm{O}}$

The different sections of the diffuser are comprised of the straight section, convergent section, throat section, and divergent section. A weak final shock is formed at the end of the throat in the diverging section which causes a considerable increase in the pressure. The location of the final shock should be inside the diverging section for smooth operation of the diffuser. Hence all the components of the diffuser, the convergent section, throat section, and the divergent section play an important role in giving pressure recovery and thereby reducing the flow velocity from high Mach number to subsonic speeds [9].

The designed nozzle and diffuser system has meshed and the boundary conditions are applied as pressure inlet, pressure outlet, axis, and wall. Type of Mesh used in this paper is quadrilateral and the number of cells is 14050, a typical discretized view of the diffuser has been shown in figure 2.



Figure 2. Nozzle and Diffuser mesh

# 2. Result Analysis of Nozzle and Second Throat Diffuser

### 2.1. Nozzle and Second Throat Diffuser System Analysis

The designed nozzle and second throat diffuser system which is plotted and meshed using ICEM-CFD are then imported into Ansys Fluent. The Axis-Symmetric option is selected to perform the analysis for 3D flow through the nozzle and diffuser system. The flow analysis is continuous as the results for complete expansion as well as pressure recovery are needed so the steady state option is selected. To obtain the flow solution faster and for quicker convergence, the implicit solver has been selected. The flow is compressible and the density is an ideal gas, hence the energy equation is selected and the Spalart-Allmaras has been selected as the Viscous Model, which is widely used in aeronautical applications also used to specify that the flow has turbulent viscosity. The major advantage of this model is that it assumes a thin boundary layer for the flow along the wall and there is no requirement to feed a user-defined function for the boundary layer. Water vapor has been selected as working fluid and the density is set to ideal gas which is required for the compressible flow of gases. Operating condition is set to zero Pascal because the initial pressure and when the flow enters the nozzle and the second throat diffuser system it is in vacuum condition. The pressure inlet is used to specify the total pressure and total temperature and the Initial Starting Pressure at the nozzle inlet. The pressure far field is used to specify the pressure outlet of the diffuser such that the nozzle flow is not affected by it [12, 13, and 14].

# 2.2. Nozzle Analysis

The nozzle is first analyzed in order to check the flow expansion through it, without the diffuser. The analysis is also used to check the pressure and temperature at the nozzle exit. This is to verify the exit Mach number, exit pressure, and exit temperature to ensure that at the specified total pressure there are no shock formations inside the nozzle. Here the nozzle exit is specified as the pressure outlet boundary condition. The Mach distribution over the nozzle is shown in figure 3.



Figure 4 shows the pressure variation along the nozzle



Figure 4. Pressure contour

The nozzle provides an exit Mach number of near 6.8, exit pressure of 250 Pa and an exit temperature of 1000 K approximately. These results, for exit Mach number and temperature, are similar to the flow results which are calculated using isentropic flow relations. The nozzle gives a smooth expansion and

International Conference on Aerospace and Med	chanical Engineering, ICA	AME'18	<b>IOP</b> Publishing
Journal of Physics: Conference Series	<b>1355</b> (2019) 012012	doi:10.1088/1742-659	6/1355/1/012012

accelerates the flow to the required Mach number. Hence this nozzle can be used for the analysis along with the second throat diffuser [15].

#### 2.3. Second Throat Diffuser Analysis

The initial design of diffuser, which is analyzed for different back pressures:

#### 2.3.1. Case -1 At Back Pressure, Pb = 200 m bar

The flow enters the nozzle, in which there is no complete expansion of the flow inside the nozzle. A shock is created within the nozzle and there is a drop in the velocity from supersonic to subsonic. This is diffuser unstarted condition, which means that there are no shocks created in the diffuser and the diffuser does not do any work in reducing the velocity or increasing the pressure of the gas (figure 5).

#### 2.3.2. Case - 2 At Back Pressure, Pb = 40 m bar

This is the minimum pressure that can be provided by the diffuser in full flow condition, i.e. when both the diffusers are in operation. The flow expands through the nozzle but oblique and normal shocks are created within the straight portion of the diffuser. Hence the flow velocity is decreased and the diffuser does not start, which means the diffuser is in unstarted condition and the diffuser does not do any changes to the flow velocity or pressure (figure 6). As the back pressure value is reduced the maximum Mach number increased from 6.8 to 7.5. This means more expansion occurs inside the nozzle as the back pressure decreases, causing the flow velocity to increase.



Figure 5. Diffuser at Back Pressure, Pb = 200 m bar.



Figure 6. Diffuser at Back Pressure, Pb = 40 m bar

#### 3. Optimization of the Diffuser

#### 3.1. Need for Optimization

It is clear from the flow analysis of the second throat diffuser is not suitable for the nozzle, as the diffuser remains in unstarted condition and there is no pressure recovery or reduction of velocity required. If tests are conducted on this diffuser it would yield inaccurate and unsatisfactory results. The design of the diffuser needs to be optimized to obtain the desired flow within the nozzle and the second throat diffuser.

International Conference on Aerospace and Me	chanical Engineering, ICA	AME'18	IOP Publishing
Journal of Physics: Conference Series	1355 (2019) 012012	doi:10.1088/1742	-6596/1355/1/012012

The method is used for optimizing the diffuser is by fixing the backpressure value at 200 m bar and varying various design parameters of the diffuser and carrying out the analysis using the same conditions each time. Complete expansion of the gases inside the nozzle and the diffuser should be started are primary criteria for the design optimization. The flow stability in the nozzle and diffuser is the next vital criteria used for finalizing a particular diffuser design.

# 3.2. The methodology used for Optimization

The major factor that is used in optimizing the diffuser design is to obtain a complete expansion of flow within the nozzle. Initially, the straight section length (z) is adjusted as it is the primary reason for the flow to not enter into the diffuser. When the flow touches the convergent section it gets deflected into the throat immediately and causes a series of oblique shocks to be formed inside the throat. The straight length is reduced and the analysis is carried out. Few of the lengths that are analyzed (z = 0.5; 1; 1.7) are found to give complete flow inside the diffuser. These lengths are used to vary other parameters in order to obtain a stable flow inside the diffuser and provide a suitable pressure recovery.

The convergent angle is kept constant and the throat radius is varied. Strong shocks are also observed when the straight section length is 0.5 m. The throat section is adjusted so that the convergent angle remains at 40. Using A.R. and shock relations the diameter of the throat is calculated as 1.15 m. The divergent angle is set at 70. This angle is increased to 100 and then 140 for reducing the pressure as much as possible. The angle is not decreased since the flow velocity does not decrease and also it is not increased further because the flow will become unstable.

# 3.3. Unstable Flow with Pressure Recovery

A diffuser design having a throat diameter of 1.15 m and length 12 m, exit diameter 3 m and divergent angle of 140 gave sufficient pressure recovery. However, the flow is unstable as the final shock is formed within the throat of the diffuser. The exit pressure after analysis is found to be near 225 m bar. In this design, any flow disturbance will cause a shock to be formed within the nozzle affecting the nozzle test results. Hence this design can also not be used for testing the high A.R. nozzle. Figures from 7 and 8 shows the Mach number and pressure variation in unstable flow.



Figure 7. Mach number variation in unstable flow



Figure 8. Pressure variation in unstable flow

International Conference on Aerospace and Mec	hanical Engineering, IC	AME'18	IOP Publishing
Journal of Physics: Conference Series	1355 (2019) 012012	doi:10.1088/1742-6	596/1355/1/012012

#### 3.4. Stable Flow with Pressure Recovery

The optimized diffuser design that gave a stable flow and a suitable pressure recovery is a diffuser of throat diameter 1.2 m, straight section length of 1 m, throat length of 12 m, exit diameter of 3 m and divergent angle of 70. The flow in this diffuser is stable since the final shock wave is created at the divergent section of the diffuser. This second throat diffuser also gives a suitable pressure recovery near 200 m bar at the exit and 190 m bar along the wall. This increase in pressure occurs in the divergent section hence any variation in the exit pressure will not cause any disturbance to the flow inside the nozzle. Figures 9 to 12 show the Mach number, pressure and temperature variation of the diffuser in stable flow operation.

At the throat, the flow velocity decreases sharply and a series of oblique shocks are created. The oblique shocks also cause the pressure to increase gradually. In the divergent section, there is a slight increase in Mach number and the final shock is created in the divergent section. After the slight increase in Mach number, the flow velocity decrease to sonic speed. The final shock also causes an increase in the downstream pressure of the flow and the pressure rises to 200 m bar along the axis and 190 m bar along the wall. This flow obtained is stable and the designed diffuser can be used to test the high A.R. nozzle. The complete expansion is obtained in the nozzle and the shock in the divergent section will assist in testing the nozzle for steady full flow condition and obtain accurate test results.



Figure 11. Mach number plots for nozzle and diffuser flow

Journal of Physics: Conference Series

 nical Engineering, ICAME'18
 IOP Publishing

 1355 (2019) 012012
 doi:10.1088/1742-6596/1355/1/012012



Figure 12. Pressure plots for nozzle and diffuser flow

# 4. Performance Analysis and Optimization of Diffuser

The diffuser design that is optimized must be analyzed for performance at different exit pressures. This is to verify the stability of the flow inside the nozzle when a test is conducted on it. It is seen that the diffuser is steady at an exit pressure of near 200 m bar. Above this pressure, the flow is unstable and will yield inaccurate results during testing of the nozzle. The pressure value below which stable flow is attained is the stable flow limit. Below this recovery Pressure value, the flow is stable and there is complete expansion inside the nozzle. This is required for testing the High A.R. nozzle and will give accurate expansion results during testing of the nozzle



Figure 13. Pressure plot for different recovery pressures

The analysis that is carried out on the optimized nozzle and the second throat diffuser gives the various flow conditions at different recovery pressures. The unstable flow is obtained above 200 m bar and below this pressure value, the flow is stable. Hence 200 m bar can be considered as the stable pressure limit. If the exit pressure is maintained below 200 m bar and the nozzle is tested the expansion is stable and the flow results can be considered as accurate. When both the ejectors are in running condition, the minimum pressure value that can be obtained is 40 m bar.

#### 4.1. Discussion

The normal nozzle without second throat diffuser provides an exit Mach number of near 6.8; exit pressure of 250 Pa. Since the nozzle gives a smooth expansion and accelerates the flow to the required Mach number, it is used for the analysis along with the second throat diffuser. The analysis carried out with the initial design of second throat diffuser shows that the diffuser remained in Unstart condition and the expansion that occurs in the nozzle is unstable. As the back pressure value is reduced the maximum Mach number increased from 6.8 to 7.5, which indicates more expansion occurs inside the nozzle as the back pressure decreases, causing the flow velocity to increase thus the need for optimization. The optimization is done by only changing the second throat diffuser wall angles and dimensions. The optimized second throat diffuser gives complete flow expansion inside the nozzle and suitable pressure recovery near 200 m bar at the exit and 190 m bar along the wall thus allows the nozzle to work at maximum efficiency.

### 5. Conclusion

The second throat diffuser must provide a suitable pressure recovery and at the same time, the flow in the nozzle should be stable. Optimization of the diffuser is done to create a complete expansion of the gases inside the nozzle. Various sections of the diffuser modified in such a way to move the final normal shock wave from the throat section to the diffuser section to attain the maximum efficiency. Even if there are any flow disturbances causing variation in the diffuser exit pressure the flow inside the nozzle will be stable because of the oblique shocks that maintain the vacuum level inside the nozzle. The shock formation inside the diffuser throat will not allow the variations in exit pressure to affect the flow in the convergent section and the nozzle. Thus the expansion inside the nozzle will not be affected and the flow results can be considered as accurate. Hence as long as the exit pressure is maintained below 200 m bar, the designed diffuser will give accurate test results in the nozzle.

# References

- [1] Alan Vincent E V, CFD Analysis of Supersonic Exhaust Diffuser System for Higher Altitude Simulation, International Journal of Engineering Studies and Technical Approach, Volume 01, No.6, June 2015, pp 1 - 8.
- [2] Bruce Ralphin Rose J et al, A numerical optimization of high altitude testing facility for wind tunnel experiments, Chinese Journal of Aeronautics, (2015), 28(3), pp 636–648
- [3] Dominic Xavier Fernando, Design and Analysis of Second Throat Diffuser System at Different Back Pressures for High Altitude Test Journal of Basic and Applied Engineering Research, Print ISSN: 2350-0077; Volume 1, Number 5; October 2014, pp. 67-70
- [4] Ch V K N S N Moorthy, Computational Analysis Of A C-D Nozzle With 'Sed' For A Rocket Air Ejector In Space Applications, International Journal of Mechanical and Production Engineering Research and Development (IJMPERD), ISSN(P): 2249-6890; ISSN(E): 2249-8001, Vol. 7, Issue 1, Feb 2017, pp 53-60
- [5] Thomas M. Berens, Numerical and Experimental Investigations on Subsonic Air Intakes with Serpentine Ducts for UAV Configurations, Challenges in Europian AeroSpace 5th air and space conference, 2015 paper no. 074, pp 1 - 22.
- [6] Chen, F., Liu, C.F., Yang, J.Y., 1994. Supersonic flow in the second-throat ejector diffuser system, Journal of Spacecraft and Rockets
- [7] Kim H., Lee Y., Setoguchi T., and Yu S., 1999. Numerical Simulation of the Supersonic Flows in the Second Throat Ejector- Diffuser Systems, Journal of Thermal Science 8
- [8] Manikanda Kumaran R, Vivekanand P K, Sundararajan T, BalaSubramanian S, Raja Manohar D., August 2009. Analysis of Diffuser and Ejector Performance in a High Altitude Test Facility, 45th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, AIAA 2009- 5008, Colorado, USA.
- [9] Liepmann, H.W. and Roshko, A. (2007), Elements of Gas Dynamics, First South Asian Edition, Dover Publications, Inc., New Delhi. India
- [10] Park, B.H., Lim, J., Park, S., Lee, J.H. and Yoon, W. (2012), "Design and analysis of a second-throat exhaust diffuser for high altitude simulation", J. Propuls. Pow., 28 (5), 1091-1104.
- [11] Cooper GK, Phares CG. CFD applications in an aerospace engine test facility. Report No.: AIAA-1990-2003. 1990.
- [12] Manikanda Kumaran R, Sundrarajan T, Raja Manohar D. Performance evaluation of second-throat diffuser for a high altitude testing facility. J Propul Power 2010;26(2):248–58.
- [13] Mon KO, Lee C. Optimal design of supersonic nozzle contour for altitude test facility. J Mech Sci Technol 2012; 26(8), pp 2589–94.
- [14] Pugazenthi RS, McIntosh AC. Design and performance of a supersonic diffuser for plasma wind tunnel. Word Acad Sci Technol 2011;5(8), pp 1254–72.
- [15] Rathakrishnan E. Gas dynamics. 5th ed. New Delhi: Prentice- Hall of India Learning Private Limited; 2011, pp 110–35.
- [16] Bruce Ralphin Rose J, Veni Grace J. Performance analysis of lobed nozzle ejectors for high altitude simulation of rocket engines. International Journal Model Simul Sci Comput 2014; 5(4):1450019.
- [17] Senthil Kumar M, Vijayanandh R, Rahul Srinivas, Arun Karthik D, Tamil Mani M, Acoustic Analysis and Comparison of Chevron Nozzle Using Numerical Simulation, International Journal of Mechanical and Production Engineering Research and Development (IJMPERD), ISSN(E): 2249-8001, Vol. 8, Special Issue 7, Oct 2018, 1089-1103.