[Graduate Research in Engineering and Technology (GRET)](https://www.interscience.in/gret)

Volume 1
Issue 7 *Rockets and Missiles Technologies.*

Article 17

June 2022

A Study on Concepts De-Laval Nozzle using CFD Tool

Krishna Yadav Graduate Engineer
Department of Aerospace Engineering Sandip University, krishnayadav8433@gmail.com

Digambar Daware Mr.
Department of Aerospace Engineering Sandip University, digambardaware7@gmail.com

Vedant Akolkar
Department of Aerospace Engineering Sandip University, vedantakolkar9511@gmail.com

Pallavi Padalwar
Department of Aerospace Engineering Sandip University, padalwarpallavi@gmail.com

Follow this and additional works at: https://www.interscience.in/gret

Part of the Aeronautical Vehicles Commons, Computer-Aided Engineering and Design Commons, Other Aerospace Engineering Commons, Other Mechanical Engineering Commons, Propulsion and Power Commons, and the Space Vehicles Commons

Recommended Citation

Yadav, Krishna Graduate Engineer; Daware, Digambar Mr.; Akolkar, Vedant; and Padalwar, Pallavi (2022) "A Study on Concepts De-Laval Nozzle using CFD Tool," *Graduate Research in Engineering and Technology (GRET)*: Vol. 1: Iss. 7, Article 17.
DOI: 10.47893/GRET.2022.1132
Available at: https://www.interscience.in/gret/vol1/iss7/17

This Article is brought to you for free and open access by the Interscience Journals at Interscience Research Network. It has been accepted for inclusion in Graduate Research in Engineering and Technology (GRET) by an authorized editor of Interscience Research Network. For more information, please contact sritampatnaik@gmail.com.

A Study on Concepts De-Laval Nozzle using CFD Tool

Cover Page Footnote

Authors would like to acknowledge professor Shiva Prasad U, For his help given in understanding CFD tool, who is well versed with the tools of ANSYS Fluent and CFX.

This article is available in Graduate Research in Engineering and Technology (GRET): https://www.interscience.in/gret/vol1/iss7/17

A Study on Concepts and CFD Analysis of De-Laval Nozzle

[1]Er. Krishna Rajaram Yadav, [2]Digambar Daware, [3]Vedant Akolkar, [4]Pallavi Padalwar
[1,2,3,4]Graduate Engineer, Aerospace Engineering, SOET, Sandip University, Nashik, Maharashtra, INDIA

***Abstract*— The pressure, density, temperature, and speed are all within the nozzle's control. It makes for quicker and easier mobility. Typical De-Laval nozzles are comprised of three distinct sections: the throat, the converging part, and the diverging section. The De Laval nozzle will be explored. The operation of nozzles is investigated in this study. Theoretical flow analysis is carried out all the way along the length of the nozzle. Changes in pressure, temperature, velocity, and density may sometimes be uncovered through the use of computational fluid dynamics (CFD). CFD is utilized in the simulation of shockwaves.**

Keywords-CFD, Nozzle, Pressure, Velocity, Flow, Mach Number

I. INTRODUCTION

A nozzle is a device used to control fluid properties such as pressure, density, temperature, and velocity. Nozzles are available in a wide range of forms, sizes, and combinations. Nozzles are most commonly used to accelerate the flow of fluid by converting the energy stored in the form of pressure and heat into kinetic energy. It is used to generate thrust, which is required for lift and is most typically seen in rockets or air-breathing engines. It is possible to boost the speed of a fluid traveling at subsonic speeds to supersonic levels by using a rocket engine nozzle. The most crucial aspect to consider when constructing a nozzle is the amount of thrust it can produce. The working fluid's molecular weight, specific heat at constant pressure or volume, and specific heat ratio are the properties of the working fluid that determine flow. Its operating altitude is also a determining factor.

II. LITERATURE REVIEW

The article entitled "Theoretical and CFD Analysis of a De Laval Nozzle" was written by Nikhil Deshpande and Suyash Vidwans. In this study, a detailed computational fluid dynamics (CFD) examination of a De Laval nozzle model was carried out. The comparison in this study is based on the findings from both theoretical analysis and computational fluid dynamics (CFD). Throughout the course of our research, we covered all of the fundamental aspects of the De Laval nozzle and presented the CFD findings for each specific instance.

Computational fluid dynamics is used to model and simulate a supersonic nozzle that was developed by Venkatesh V and C Jayapal Reddy. In this paper, both theoretical and CFD assessments of a wide variety of nozzle designs are presented. The circumstances of the thrust as well as the design variables are discussed in this nozzle. We utilised examples and theories that were relevant to the issue at hand (2)

III. EXPANSION RATIO

The expansion ratio of the nozzle is the ratio of the area of the exit to the area of the throat. When creating a path for a rocket, the expansion ratio that maximizes performance must be chosen in order to account for the fact that a rocket often does not pass via a single height. While designing, other criteria such as the nozzle's weight, length, manufacturability, heat transfer, and aerodynamic characteristics are considered, consequently modifying the nozzle's form.

1) De-Laval Nozzle

Supersonic speeds may be achieved by the use of the De-Laval nozzle. Converging Diverging Nozzles, or De-Laval nozzles, are the most frequent type of nozzle. An asymmetric hourglass is created by squeezing a tube in the middle, generating a perfectly balanced asymmetry. Supersonic jet engines and steam turbines both use it often. In astrophysics, they can be used in jet streams for certain purposes.

At maximum flow rate, the cross section of a converging nozzle's throat narrows. Fluid can travel at supersonic speeds in a converging nozzle [Ma=1]. If you want your fluid velocity to reach supersonic levels, you need a diverging component in the nozzle. The term "De-Laval Nozzle" or "Converging Diverging Nozzle" refers to a similar nozzle design.

2) Operation

A gas's subsonic flow speed increases as the conduit it is travelling through gets smaller since the mass flow rate remains constant. Gas flow is believed to be isentropic. Because gas may be compressed in a subsonic flow, sound can travel through it. A condition known as chocked flow occurs around the neck when the cross-sectional area is at its smallest. A sound wave cannot go back through the gas as perceived from the nozzle's frame of reference [Ma>1] as the cross-sectional area of the nozzle rises.

To produce supersonic velocity, it is not enough to simply force a fluid through a nozzle. In the diverging area, the fluid might decelerate rather than accelerate if the backpressure is not within the range. The nozzle flow state is determined by the Overall Pressure Ratio.

1. When $P_0=P_b$

When $P_0=P_b$, there is no pressure differential and consequently no flow through the nozzle.

2. When $P_0>P_b>P_c$ The flow stays subsonic through the nozzle. The fluid velocity increases in the converging portion and comes to the maximum at the throat, although the velocity is still subsonic at the throat. Thus in the diverging section, fluid loses its energy and diverging part works as a diffuser. The pressure lowers up to the throat and again rises in the diverging region.

3. When Pb=Pc

At the throat, the pressure changes to P* and the fluid achieves sonic velocity. P* is the lowest pressure at the throat, whereas the velocity reached is the greatest velocity after passing through a converging nozzle. Additional reduction has no influence on the flow of water through the converging segment, but it has an effect on the diverging segment.

4. When PC>Pb>Pe

The fluid that reaches sonic velocity in the diverging region continues to accelerate to supersonic speeds in this area. Acceleration comes to an abrupt halt when the pressure drops. Shocks normally occur between the throat and exit plane, resulting in a decrease in velocity and a slowing of the flow. Isentropic flow cannot mimic shock fluxes because they are so irreversible.

Pb=Pe causes an abrupt change in the exit plane, resulting in a shockwave. Because of this, the supersonic flow that enters the nozzle becomes subsonic before it exits and crosses the normal shock.

5. When Pe>Pb>0

Due to the fact that the flow in the diverging portion is supersonic, the fluid is able to expand to PF at the nozzle exit without experiencing the normal shock. Therefore, flow may be analysed using the concept of isentropic flow. Because of this, when Pb equals PF, the pressure within the nozzle will increase from PF to Pb and remain at that level forever.

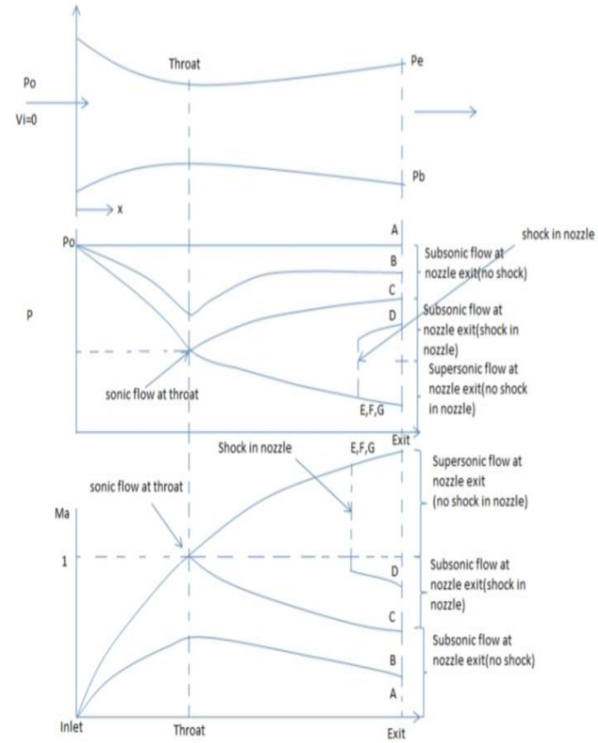

Figure:1 Pressure Conditions

IV. *SHOCKWAVE*

A. The formation of a shockwave is caused by sudden shifts in the fluid characteristics that take place in a very tiny segment of a De Laval nozzle when it is subjected to circumstances of supersonic flow for certain back pressure values.

B. Normal shocks

Shockwaves that travel in a plane perpendicular to the fluid's movement are referred to as normal shocks. The flow slows down to subsonic speeds after the shock when it was previously supersonic. For a shockwave to occur, the flow speed must be reduced from supersonic to subsonic. Just before the shock happens, the amplitude of the shock will be precisely proportional to the Mach number. The shock wave converts into a basic sound wave in the extreme case where Ma = 1.

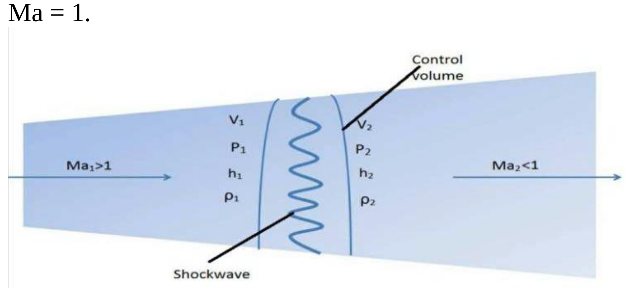

Figure:2 Shockwaves

The stagnation enthalpy remains constant across the shock, according to the conservation of energy principle; $h_{01}=h_{02}$. $T_{01}=T_{02}$ is also the temperature of stagnation. Because of irreversibility, the stagnation pressure drops over the shock, but the ordinary temperature rises dramatically due to the conversion of kinetic energy into enthalpy due to a substantial reduction in fluid velocity.

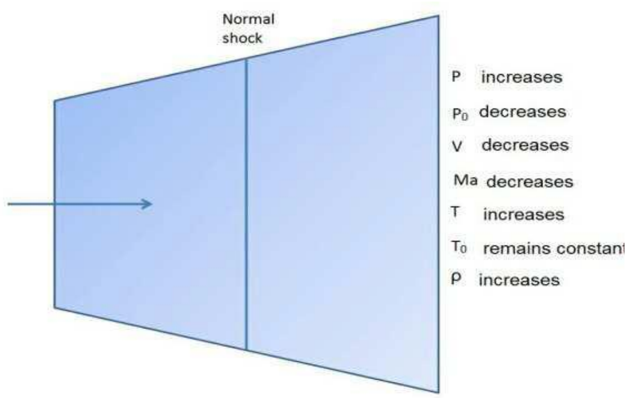

Figure:3 Variation of parameters in shockwave

C. Oblique Shocks

An oblique shock occurs when a shockwave is angled away from the flow direction. During downstream flow, oblique shock is usually created at the nozzle's snout. When supersonic flow encounters a corner, it essentially turns the flow toward itself and compresses it, causing oblique shock. A Galilean transformation can be used to convert it to a regular shock. In an ideal gas (k=1.4), a 2MPa pressure and a 300K temperature are applied to a converging diverging nozzle at a negligible velocity. Flow pressure, temperature, velocity, and density through the nozzle are evaluated both theoretically and through computer simulations (CFD). The chosen model is shown below.

V. COMPUTER SIMULATION OF DE LAVAL NOZZLE

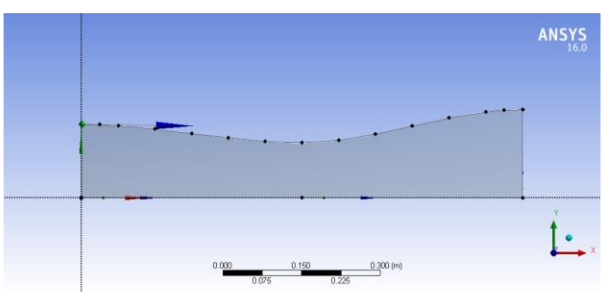

Figure:4 CAD model of nozzle

Computer simulation of nozzle was done using computational fluid dynamics (CFD). CFD is method to solve complex problems involving fluid flow.

The above conditions were taken as mentioned in the previous section and the computer simulation that is the analysis was done using Ansys-Fluent. When we performed this analysis, we found out the variation of parameters as we did in the case of theoretical treatment.
The CFD Analysis was done in the following steps:

- Modelling
- Meshing
- Pre processing
- Solver/Processing
- Post processing

1) Modeling

Ansys design modeller was used for the modelling of the De Laval nozzle. There was a construction point command and then 3D curves were used to join up the points. Additionally, the option of drawing lines from points was used in this case. Surface from edges command was used to create a 2D surface. Here is a breakdown of the nozzle's dimensions.

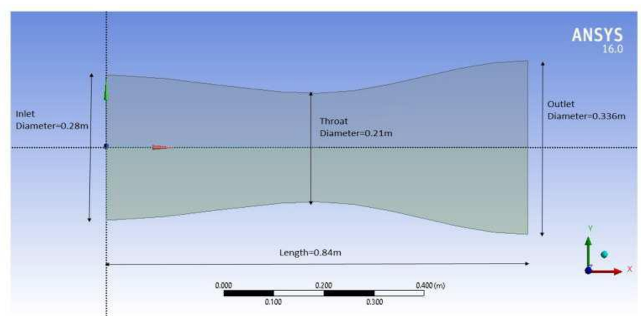

Figure:5 Model

VI. MESHING

After applying the previously specified dimensions to produce a model, the model was subsequently meshed by employing the mesh mode of the Ansys component systems. The process of meshing is nothing more than the transformation of a model that has an infinite number of particles into one that contains a limited number of particles. This transformation takes place throughout the meshing process. For the particulars of the mesh, CFD was the approach that was most frequently used in Physics. The importance of meshing has been cranked up to a level of 100 priority. Because the nozzle is a curved piece of equipment, the approach that takes both proximity and curvature into consideration was selected as the best option. A high degree of smoothing was applied, and the number of cells spanning the gap was increased to fifty. The smoothing was also turned up to a high level. The blending was carried out quite successfully. As a direct consequence of this, the total number of modes was found to be 12760, whereas the total number of components was found to be 12427. mapped face meshing software was utilised in the modelling of both of the faces. The focal point of significance was moved to a more appropriate location, and the features that were included into the design were of the quadrilateral form.

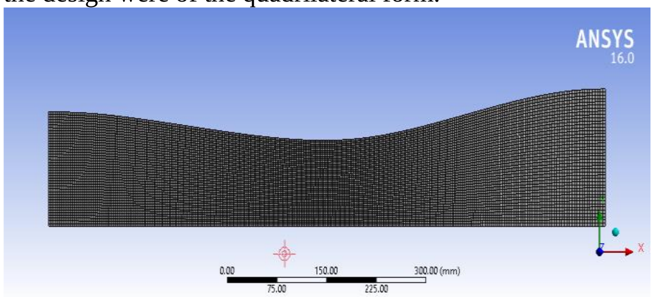

Figure:6 Meshed model of nozzle

1) Post Processing/Result

The rate of acceleration continues to rise as one moves from left to right. The entrance part has the lowest value, while the output section has the highest value. When measured at the neck, velocity is equivalent to sonic velocity. The minimum speed is 161.571 metres per second, while the highest speed is 525.648 metres per second.

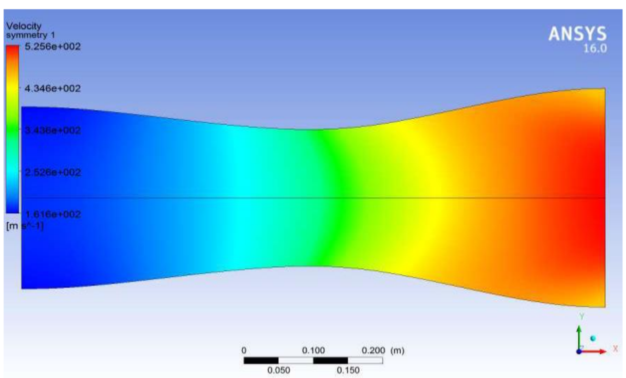

Figure:7 Velocity contour and graph (m/s)

2) Temperature Variation

The fluctuation of temperature is analogous to the fluctuation of pressure. It is at its highest point at the inlet and its lowest point at the outflow. The highest possible temperature is 287.082 Kelvin, while the lowest possible temperature is 163.15 Kelvin.

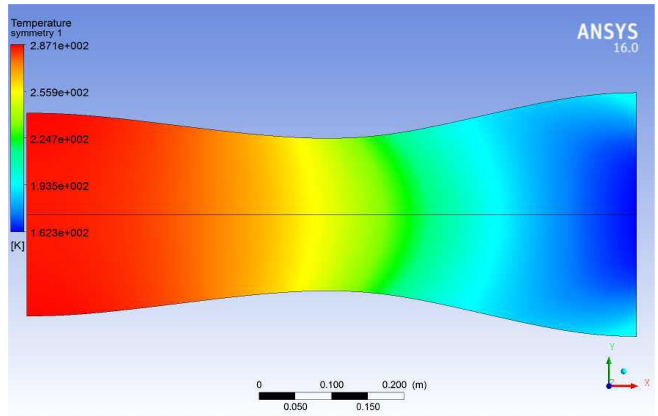

Figure:8 Temperature contour and graph (K)

3) Density Variation

The flow's density is at its highest point at the intake and its lowest point at the outflow. The density can range from a minimum of 4.99 kg/m^3 all the way up to a maximum of 20.8171 kg/m^3.

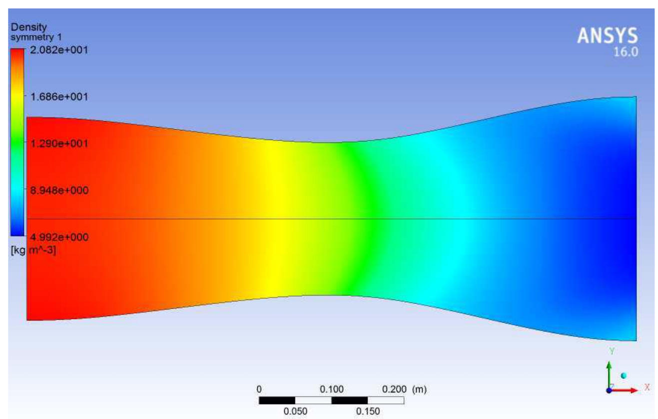

Figure:9 Density contour and graph (kg/m^3)

4) Shockwave

Shockwave results across the flow are as follows.

a) Pressure variation due to shockwave

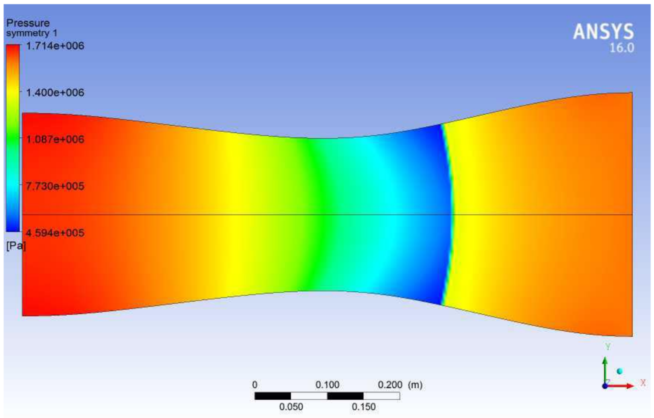

Figure:10 Pressure variation in shockwave

b) Velocity variation due to shockwave

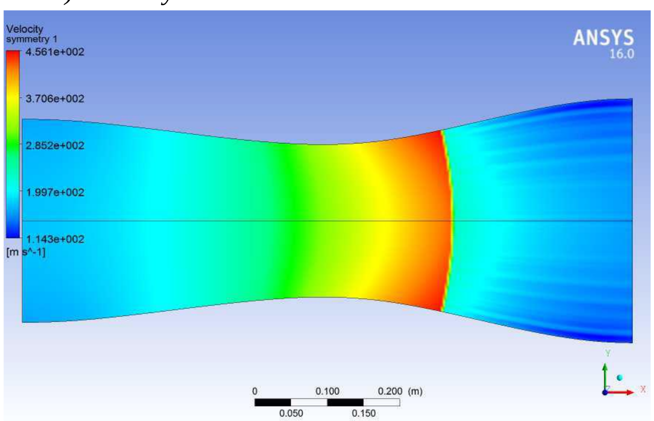

Figure:11 Velocity variation during shockwave

c) Temperature variation due to shockwave

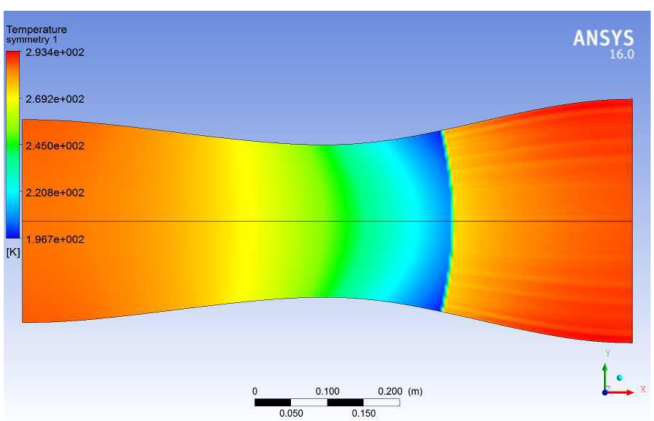

Figure:12 Temperature variation in shockwave

d) Density variation due to shockwave

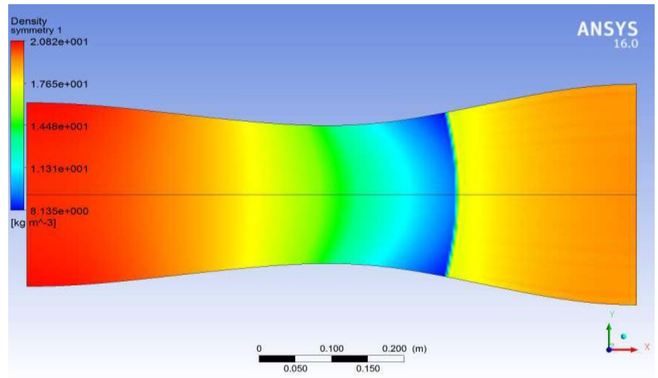

Figure:13 Density variation in shockwave

VII. CONCLUSION

This paper has discussed the fundamental concepts of a converging-diverging nozzle in this study. Through this research and analysis, authors have determined the values of variables of nozzle parameters using a theoretical approach, namely equations and CFD. As a result, we may conclude that the results obtained using these two methods are equivalent.

REFERENCES

[1] Nikhil Deshpande and Suyash Vidwans, "Theoretical and CFD Analysis of De Laval Nozzle" , vol.2,No 4,April 2014.

[2] Venkatesh V and C Jayapal Reddy, "Modelling and simulation of Supersonic nozzle usingcomputational fluid dynamics,"vol. 2,2015, No 6, pp. 16-27.

[3] Mr. Lijo P Varghese, Mr. Rajiv Saxena And Dr. R.R.Lal, Analysis of the Effect of Nozzle Hole Diameter on CI Engine Performance Using Karanja Oil-Diesel Blends. International Journal ofMechanical Engineering and Technology (IJMET), 4(4), 2013,pp. 79–88.

[4] Yunus Cengel and John Cimbala,"Fluid Mechanics" 3rd edition, pp. 659-723.

[5] Yunus Cengel and Michael Boles, "Thermodynamics an engineering approach,"5th edition,pp.823-907

[6] Radhakrishnan,"Gas Dynamics'by printice hall publishers.

[7] Nagesha S, Vimala Narayanan, S. Ganesan and K. S.Shashishekar, CFD Analysis in an Ejector of Gas Turbine Engine Test BED. International Journal of Mechanical Engineering and Technology (IJMET), 5(9), 2014,pp. 101–108.