IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

Aerodynamic study of flow through convergent-divergent nozzle and study of various parameters by using Ansys15.0

Shakeel Ahmad[1], Devesh Kumar[2]

[1]M.Tech Student, Department of mechanical engineering, Madan Mohan Malaviya University of TechnologyGorakhpur, U.P. india.
[2]Assistant Professor, Department of mechanical engineering, Madan Mohan Malaviya University ofTechnology Gorakhpur, U.P. india.

ABSTRACT

In this paper, we have been to analyses two bell nozzles of varying length and a similar cone nozzle, with the same exit area ratio. By running these nozzle configurations under similar conditions at different altitudes, the velocity and pressure contours are obtained. By studying these contours, it is found that the 85% Bell Nozzle runs at design conditions over a greater range of altitude than the Cone Nozzle. Also below 70% Bell Nozzle length, it is found that the exhaust gas flow becomes highly unstable and the thrust produced is very low. On calculating the specific impulses for each condition, it is found that the 85% Bell Nozzle has a 6-7% loss when compared to the cone Nozzle. The k- model with standard wall treatment was used for analyzing the flow. Here the working fluid has been taken as air. All the calculations were done by using ANSYS-15.0 Workbench.

*Keywords : **Bell type nozzle, CFD, Numerical Analysis, Reynolds number.***

INTRODUCTION:

A rocket nozzle is a drive nozzle (typically de Laval type) used in a rocket engine to expand and accelerate the combustion gases generated by burning propellant. A propulsion system has a source of mechanical power (some sort of motor or engine), and a few methods for uses this capacity to produce power, for example, wheel and axles, propellers, a propulsive nozzle, wings, balances or legs. Gas Propulsion Systems area broadly divided into two classifications dependent on their source of oxygen:
1. Air-Breathing. 2.Non-Air-Breathing.
In these nozzle theory, thrust is the power delivered a rocket drive following up on a vehicle. the rearrange way, this response structured by its experienced because of discharges issue to high speed. This represent to a similar phenomenon into the pushes a patio nursery hose in reverse or makes a firearm draw back. In the last case, the slug frontage and powder charges are equivalent to the response or reward of the adjustable weapon. The army is a number and is characterized by the result of the mass-time speed. The cause, due to the adjustment in energy, is given below. Stream mass stable and gas leaves uniform velocity and axis.

$F = \frac{dm}{dt} v_2 = \dot{m} v_2 = \frac{w}{g_0} v_2 \, F = \frac{dm}{dt} v_2 = \dot{m} v_2 = \frac{w}{g_0} v_2$

This force represent to the absolute propulsion force when the spout leave weight rises to the surrounding weight.

The axial thrust can be controlled to combine the pressure following the zones that can be the common plane to project the nozzle hub. The extraordinary power is extraordinary, but does not contribute to the axial core as usual as axial symmetry space. The condition that enters the nozzle is basically unchanged. As a result of the nozzle geometry and surrounding pressure as the change of change to altitude, Therefore, homogeneous atmosphere for rocket propulsion system is moving on, and then the thrust force is

$$F = \dot{m} v_2 + (p_2 - p_3) A_2 \, F = \dot{m} v_2 + (p_2 - p_3) A_2$$

The first term of the thrust force is represented by the mass of flow, velocity and exhaust velocity its respect to the vehicle. After the second term is represented on the pressure thrust which includes the cross section of area result at the boundary of A2 (where leaves the vehicle into the exhaust jet) so that the difference between the atmospheric pressure and exhaust gas pressure into the exit. And the liquid, pressure or the thrust pressure is negative to these are not include to the thrust pressure. So, this low pressure into the rocket nozzle is usually planned that the weight of the smoke is equal to or higher than the weight of the surrounding liquid. At the point when the weight is equal to the weight of the weight, the weight is zero and is simultaneously subtracted by adding it to the state of the power, in vacuum P3 space and the thrust moves towards being the rocket nozzle is designed that the exhaust pressure is equal and slightly higher than the ambient pressure into the fluid. And then the atmospheric pressure is aligned with the exhaust pressure, into the term pressure is zero and the thrust is equal to the driving force equation, in the vacuum P3 column and the thrust becomes

$$F = m v_2^{\cdot} + p_2 A_2$$

So, the state of the pressure in which the pressure exhaust is actually adjusted into the fluid ambient pressure (P2 = P3) is referred to the ideal development part into the rocket nozzle. The main terms of thrust are as follows:

F= q x Ve + (Pe - Pa) x Ae

Where,
F = Thrust
q = Propellant mass flow rate Ve = Velocity of exhaust gasesPe = Pressure at nozzle exit
Pa = Ambient pressure Ae = Area of nozzle exit

Product qVe is known of the momentum of velocity thrust and into the product (Pe-Pa) Ae is known to the thrust pressure.

G. Hagemann, Hans Immich, in their study “Advanced Rocket Nozzles”, journal of Propulsion and power, This paper deals with conventional nozzle performance analysis, heightened computer nozzles. And it is found that in conventional nozzles, separation of flow at low altitude due to high back pressure. And it also found that by using a nozzle bell contoured, another wider height compensation could be achieved.[1]

Ralf H. stark, in their study “Flow separation in rocket nozzles, a simple criteria”, in this paper the cold and heat flow tests are carried out to investigate the separation of streams in the rocket nozzles. Separation of flow is the result of adjustment to ambient conditions. This adjustment can be divided into two areas (oblique shocks offset the wall pressure to the plateau pressure, followed by a re-compression wave system in the separated backward region where high pressure pressure is adjusted to ambient conditions.[2]

Z. F. Nasuti, M. Onofri, “Structure of flow and separation in over expanded rocket nozzles,” This paper analyzes the basic flow structure that may occur in the nozzle divergent section during startup and low altitude flight, when the nozzle operates in an overly regime.The possible generation of the separation assessment phenomenon and its role towards the occurrence of free and limited shock isolation regime. And this paper provides to provide indications on the possible generation of a recirculating flow region behind it that may yield side load on the nozzle wall.[3]

K.M. Pandey and A.P. Singh in their study was instructed to understand the flow of gas in cones such as nozzles at edge advantage using the 2-dimensional axisymmetric model. Similarly, they find flows of various parameters, such as pressure proportioning, nozzle part or area of nozzle area, and flow Mach numbers in out nozzle are described before propagation. The results show a variety in Mach no., Pressure. For the 40 edge Mach number at the low exit looks different in relation to a variety of nozzles and turbulence or hardness interruptions are at 160 at the exit and practically 3,045e + 05. Even when the edge is 80 Mach numbers at the nozzle is 2.91 and equal to 120 , therefore, the advantage of 160 so given the Mach number at the nozzle out is 2.92 and it is least or decreased at the edge of 40. The dimensions of the plots for the nozzle of the pipe model can be very large like the 12 to 18 degrees most unusual so to push the best, we can continue to run with 120 or160 nozzles. Terma-Conical Nozzle nozzle, point dimension, supersonic.[4]

Hocine MZAD and Mohamed ELGUERRI are considered on compressible stream in a merging wandering spout. From a hypothetical and exploratory examination of the stream in an illustrative channel with a simultaneous divergent area assortment, the lead of1Venkatesh .V, 2C Jaya buddy Reddy contemplate on The C-D spouts both funnel shaped and form are planned on a presumption of the isentropic progression of the ideal gas The computer code which uses the technique for traits and the stream ability to describe high adequacy spout profile for isentropic, inviscid, disturbance supersonic floods of any working fluid for any customer portrayed leave Mach number. The arranged spout zone extent is appeared differently in relation to hypothetical district extents for the picked fluid and needed leave Mach number.[5]

METHODOLOGY:

Computational fluid dynamics (CFD) investigation to the system begins with the development of desired or wanted geometry and meshing for demonstrating the domain. firstly, geometry is disentangled for the computational Fluid Dynamics thinks about. Meshing is the desecrating of the area of little volumes where into the equations are comprehended by the assistance of iteration techniques. Modelling begins with the describing of the boundary and beginning conditions for the dominion and leads to demonstrating of the whole system. Genarally, it is followed by the investigation of the outcomes,ends and exchanges.

1. Geometry:

The ideal rocket nozzle takes a conical state with a side edge of 16 °. For accommodation items, the ratio of area 25.4 is nozzle due to the design of the parameters taken. At lower altitudes or elevations, the nozzle launched on the bottom is prolonged at a higher altitude, the nozzle acts less developed. Also modeled by joining outlets that allow the nozzle to break it. This creates exhausting nozzles into the air that gives us the opportunity to reflect on the flow of exhaust and the development of gas flows. The nozzle dimensions are given below

- Throat Dia- 11.5 cm
- outlet Dia- 62.7 cm
- Thrust Dia -25 cm
- Nozzle length- 114.435 cm
- Convergent of length- 22.342 cm
- Divergent of length - 92.3 cm
- Area Ratio- 24.4
- Area ratio of convergent nozzle-4
- Exit Area- 3087.96 cm2
- Area ratio of throat in nozzle-121.72 cm2
- Inlet Area-489.87 cm2

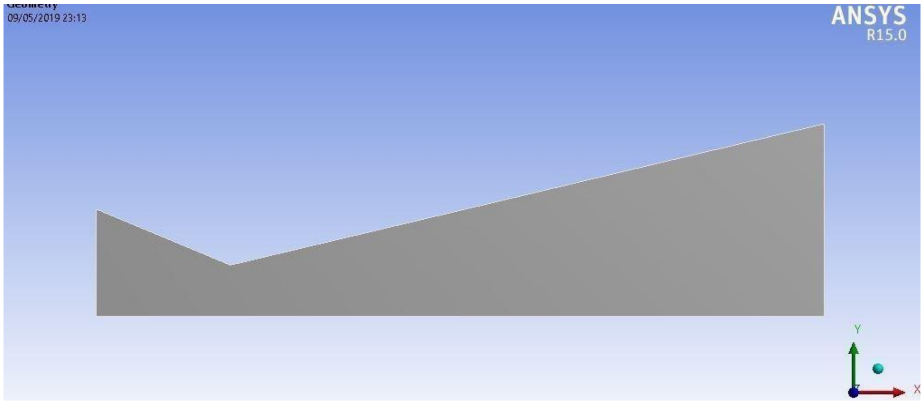

Fig.1 Ideal nozzle of geometry.

2. Extrude:

By choosing first sketch in xy plane expel it up to a length of 1600mm.again select another sketch Extrude it up to 1600mm and change activity from add material to include solidified. Essentially expel sketch 3 and sketch 4 upto 1600mm length in z-course and change task add material to include frozen. By extrudeing entire body and by changing activity from add material to include frozen we have four different parts and four bodies.

Table 1: Naming of various parts of the body with state type

PartNumber	Part of the model	State type
1.	Inner fluid	Fluid
2.	Inner Nozzle	Solid
3.	Outer Fluid	Fluid
4.	Outer Nozzle	Solid

3. Meshing:

The mesh is a combination of discrete "components" representing to the geometry. The centers and componentsrepresenting to the geometry indication make up the mesh. A "default" mesh is naturally created in the middle of a solution.It is by and large prescribed that extra controls be added to the default mesh before beginning. In cross section mode, fluentcapabilities as a hearty, unstructured network age program that can deal with a mesh of for all plans and purposes never- ending size and multifaceted nature. Cross sections may comprise of tetrahedral, hexahedral, prismatic, or pyramidalchambers. Unstructured mesh age strategies couple fundamental geometric building hinders with broad geometricinformation to computerize the work age process.

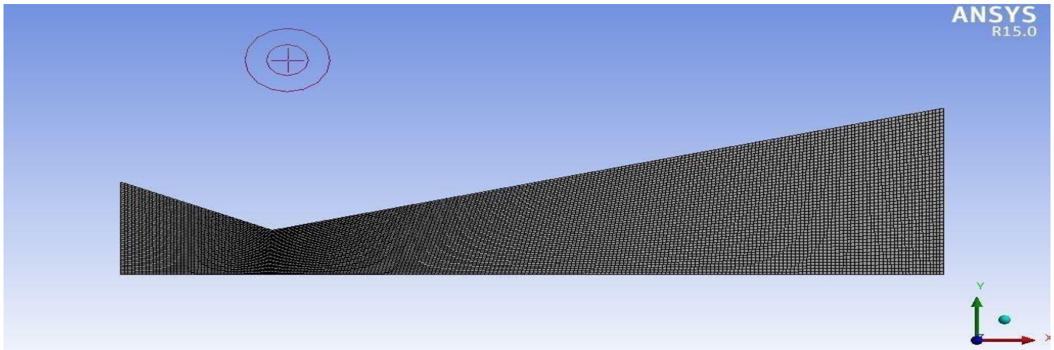

Fig.2 Ideal nozzle of meshing.

- Meshing the minimum size: 1.07e-003m
- Face size of maximum: 0.106066
- Maximum Size: 0.21342m
- Edge Length of minimum size: 4.85e-012m
- Element Size: 1e-012m
- Node:10656
- Elements:10387

4. Solution:

Problem Setup

With the near aim of the investigation, some condition are become identified with the fluid stream. In issue detail, air stream at quick through a focalized different nozzle having a cross-sectional area that shifts with hub eliminate from the throat. Also, some broad condition is utilized like thickness based solver which is used for the fast compressible stream. It gives better accuracy advantage over the pressure based solver for the quick compressible stream. Energy equation needs to be turned on since this is a compressible flow where the energy equation is coupled to the continuity and momentum.

Table 2: Setup condition and solution control

PROCEDURE	DETAILS
Problem SetupGeneral-Solver	Type: Density basedVelocity : Absolute Time: Steady 3D space: Planar
Models	Energy: On Viscous: turbulent k-epsilonStandard wall treatment
Materials	Fluid: Air Density: Ideal Gas Viscosity: Sutherland

Boundary equitation	Inlet : Pressure Inlet gauge Total Pressure (pa): 3e5Outlet : Pressure Outlet Gauge Pressure(pa): 0 Temperature: 300k
Reference Values	Compute from: Inlet Reference Zone: Solid Surface body
Monitors/Residual	Create-walls –cd 1 Residual- 10-6 Select Print to console and plot.

Models:

Energy is set to on position. Viscous model is selected as "k-ε" model (2 equation).

Selection of Materials:

All the nozzle configurations or arrangements are kept running with a similar material characteristics. The different materialis:

- Fluids - Air
- Material of nozzle- Titanium

In the material of the air is a uses into the working of fluid for analysis of cold flow. And the nozzle of material to reduces,and a loss of friction, then the material uses titanium into the interior rocket of the nozzle.

BOUNDARY CONDITIONS:

Pressure Inlet

In these condition, the flow and properties into scalar of boundary condition to found the fluid of pressure, so the calculation of the compressible and non-compressible is suitable. Therefore in this boundary condition pressure of inlet are used so this function is called for rate of flow and not known the velocity. And include of the buoyancy force flow in this situation. The incoming pressure limit condition is also used to the define of "free" to the external boundary conditions. in these condition combustion chamber of simulated is-

- ❖ Pressure- 7.09275 bar
- ❖ Temperature- 550 °c

Pressure Far-Field

In this solver certain static condition to the streaming of the mach number are uses to the model of free flow at infinity from the ansys fluent. These boundary conditions are often called characteristic boundary condition, because they use the characteristic boundary conditions, because they use of the characteristic of the boundary conditions information estimate the boundaries of the flow into the variable. And these under condition are simulated to the reservoir. So finally exhausting of the rocket nozzle into the atmosphere.

Solution Methods and initialization

The solution methods effectively solve the system of non-linear algebraic equations generated by the discretization methods. Dependent upon the matter, transient or reliable precise and the course of action of equation are linearized by methods for prominence by methods for various plans. The transient problem will have a period secondary beginning quality issue for Ordinary Differential Equations (ODEs), or a proportionate cycle arrange for a reliable state matter.

Solution initialization is done. Initial values of velocity are taken as 0 along each direction. And temperature is taken as300k. Residual monitoring is done and convergence criteria are set up.

Solution Control

Under control of the arrangement, and the factor for momentum under the pressure, density and energy is converted to 0.6 because this is a high slope that will cause a breakdown in the breaker if the factors under leeway are not reduced.This arrangement is inserted with the calculated quality of the inlet.

MEASURE OF CONVERGENCE:

The combination is a significant matter with the utilization of CFD user interface design. Fluid mechanics is included with non-direct procedures, managing usually unjustified phenomena, for example, and turbulence. CFD user interface design is expected to reform these physical procedures, and in this way is liable to indistinguishable issues from the procedures it is attempting to speak to. All things considered, it isn't ensured that there will be a stable state 'converged' solution for a matter. As this definition determines, the correct answer for the iterative matter is doubtful, yet you need to be adequately near the answer for a specific required level of accuracy.

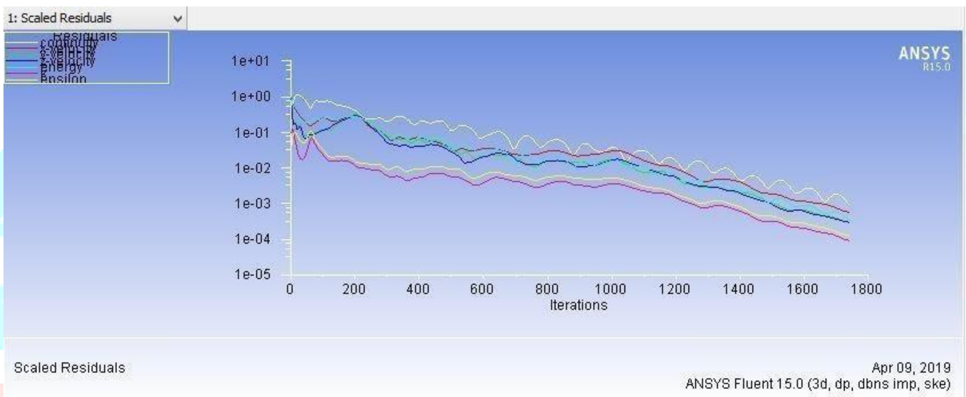

Fig.3 Residual Graph of Ideal Nozzle**.**

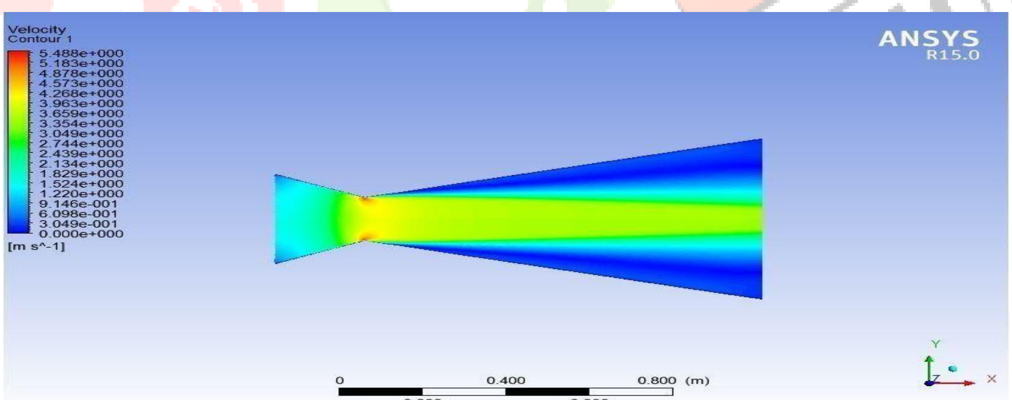

Fig.4 Velocity contour of Ideal Nozzle

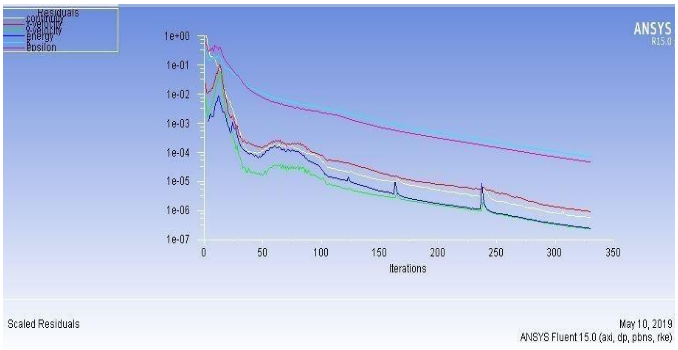

Fig.4. 85% bell nozzle of residuals graph

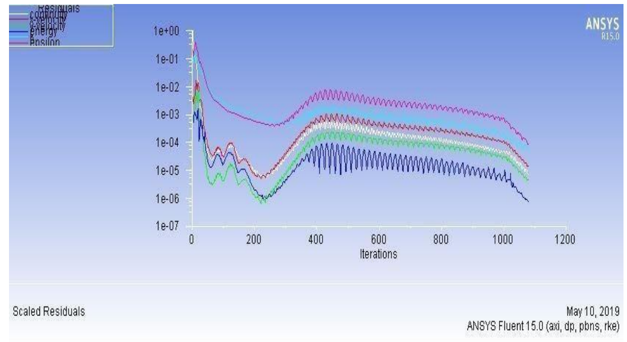

Fig.5.70% bell nozzle of residuals graph

Different conditions of mass flow rates are given below:

1. Mass flow rate of Ideal nozzle - 175kg/s
2. Mass flow rate of 85% Bell Nozzle - 193 kg/s
3. Mass flow rate of 70% Bell Nozzle - 196 kg/s

RESULTS AND DISCUSSION:

A CFD package (ANSYS FLUENT 15.1) was made for the numerical investigation of two bell nozzles of varying length and a similar cone nozzle, with the same exit area ratio. By running these nozzle configurations under similar conditions at different altitudes, the velocity and pressure contours are obtained. By studying these contours, it is found that the 85% BellNozzle runs at design conditions over a greater range of altitude than the Cone Nozzle. Also below 70% Bell Nozzle length, it is found that the exhaust gas flow becomes highly unstable and the thrust produced is very low. On calculating thespecificimpulses for each condition, it is found that the 85% Bell Nozzle has a 6-7% loss when compared to the cone Nozzle. The k- model with standard wall treatment was used for analyzing the flow. Here the working fluid has been taken as air. All the calculations were done by using ANSYS-15.0 Workbench.

REFRENCES

[1] H. Versteeg &W. Malalasekra,(2007) "Intoduction to Computational Fluid Dynamics" Pearon Publications. [2] Dieter K. Huzel, David H. Huang, "Modern Engineering for Design of Liquid-Propellant Rocket Engines", Progress inAstronautics and Aeronautics,Vol.147

[3] Paradhasaradhi Natta, V. Ranjith Kumar, Dr . Y. V. Hanumantha Rao,(2012) "Flow Analysis Of Rocket Nozzle Using Computational Fluid Dynamics (CFD)", International journal of Engineering Research and Application (IJERA), Vol.2

[4] George P. Sutton, Oscar Biblarz,(2001) "Rocket Propulsion Elements, 7 th edition" John Wiley&Sons

[5] Joel H. Ferziger Milovan Pric, (2002) "Computational Methods for Fluid Dynamics" Springer Publications.

[6] Taro Shimizu, Masatoshi Kodera, Nobuyuki Tsuboi,(2008) "Internal and External Flow of Rocket Nozzle", Journal of Earth simulator, volume 9

[7] Z. F. Nasuti, M. Onofri, "Flow Structures and Seperation in Overexpanded Rocket Nozzles", European conference for Aerospace Sciences (EUCASS)

[8] Robert Zucker, O. Biblarz,(2002) "Fundamental of Gas Dynamics" John willey & Sons

[9] H. kbab, M Sellam, T. Hemitouche, S. Bersheul and L.Lagab, "Design and evaluation of dual bell nozzle, Acta Astronautica,2016 52

[10] S.M. Yahya, "Fundamental of compressible flow with aircraft and Rocket propulsion," New Age internationalpublishers

[11] K.M. Pandey,Member IACSIT and A.P. Singh study on "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

[12] I Venkatesh .V, 2C Jaya pal Reddy work on "Modelling and Simulation of supersonic Nozzle Using Computational Fluid Dynamics" Vol.2, Issue 6, 2015

[13] Hocine, MZAD 1, a and Mohamed ELGUERRI 2, b study on "Theoretical and Experimental investigation of compressible flow through convergent-Divergent Nozzles" Advance Materials Research Vols.452-453(2012) [14] Separation behaviour of A. Grob and C. Weiland, "Investigation of Shock Patterns and several Subscale Nozzle" in 36th AIAA/ASME/SAE/ASEE joint Propulsion Conference & Exhibit,2000

[15] Gerald Hagemann, Hans Immich, et.al. "Advanced Rocket Nozzles", Journal of Propulsion and Power, Vol.14, No.5

[16] Beena D.Baloni, Sonu P.Kumar, S.A. channiwala "Computational Analysis of Bell Nozzles" Department ofMechanical Engineering SVNIT, Surat Gujrat. India 2017

[17] S. Bhushan Verma , R. Stark, O.Haidn, "Relation Between shock unsteadiness and the origin of side-loads inside," Aerospace Science and Technology, Elsevier,2006

[18] Y.Yu, J, Xu, J. Mo and M, Wang, "Principle parameters in flow separation patterns of Over-Expanded Single Expansion Ramp Nozzle," Engineering applications of Computational Fluid Dynamics,vol. 8, no.2, 2014