IJARnD INTERNATIONAL JOURNAL OF ADVANCE RESEARCH AND DEVELOPMENT

(Volume 3, Issue 10)

Available online at: www.ijarnd.com

CFD analysis and parameter optimization of Divergent Convergent Nozzle

Uttam Kumar[1], Sudhir Singh Rajput[2], Dr. Praveen Borkar[3]

[1]*Student, Raipur Institute of Technology, Raipur, Chhattisgarh*

[2]*Assistant Professor, Raipur Institute of Technology, Raipur, Chhattisgarh*

[3]*Head of Department, Raipur Institute of Technology, Raipur, Chhattisgarh*

ABSTRACT

The current research work is related to the computational fluid dynamic analysis of two-dimensional convergent-divergent nozzle in Ansys software. It using the CVM (control volume method) to solve the governing equation of fluid flow problem formulated under the given boundary condition. The basic aim of the current study is to determine the most suitable or optimum configuration of convergent-divergent angle in DC Nozzle. The parameter of a nozzle is taken according to the DC nozzle geometry. The different configuration has made by vary angle from 15 to 40 degree at the step of 5 degrees for Convergent angle and for divergent angle, it varies from 12.5 degrees to 20 degrees at the step of 2.5 degrees. The analysis was performed in the fluent workbench of ansys software. The input data for the nozzle is taken as the temperature of exhaust gas and pressure at the inlet. The output data is obtained by fluent in the form of temperature plot and pressure distribution and velocity gradient and Mach number are calculated for each combination.

Keywords*— *CFD (Computational Fluid Dynamics), CVM (Control Volume Method), DC (Divergent Convergent Nozzle)

1. INTRODUCTION

In flight at high altitude and high forward speeds, the use of a convergent-divergent Propelling nozzle on turbo-jet or ram-jet engines is essential in order to achieve the greatest possible net thrust. However, at the off-design condition, such as would occur at take-off or when flying at reduced speed, a fixed geometry divergent nozzle is inefficient due to the large negative pressure thrust which arises as the result of over expansion within the nozzle. Before the part load performance of an engine fitted with a fixed divergent nozzle can be calculated, it is necessary to know under what condition the nozzle runs full and how the position of internal shock varies with the applied pressure ratio.

The nozzle is a mechanical device designed to control the direction or characteristics of a fluid flow. It is a specially shaped tube through which hot gases flow. Nozzles are frequently used to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the stream that emerges from them. Nozzles come in a variety of shapes and sizes depending on the mission of the aircraft. Simple turbojets, and turboprops, often have a fixed geometry convergent nozzle. Turbofan engines often employ a co-annular nozzle. The core flow exits the center nozzle while the fan flow exits the annular nozzle. Mixing of the two flows provides some thrust enhancement and these nozzles tend to be quieter than convergent nozzles. Afterburning turbojets and turbofans require a variable geometry convergent-divergent - CD nozzle. In this nozzle, the flow first converges down to the minimum area or throat and then is expanded through the divergent section to the exit at the right. The variable geometry causes these nozzles to be heavier than a fixed geometry nozzle, but variable geometry provides efficient engine operation over a wider airflow range than a simply fixed nozzle.

Fig. 1: De Laval nozzles

2. PROBLEM FORMULATION

The release of heat energy in the combustor serves to raise the internal energy of the combustion products. In order to create thrust, it is necessary to convert that energy into kinetic energy and thereby increase the velocity of the flow when it exits the propulsion device. A simple device for accelerating a fluid in the nozzle, a duct whose area is varied in such a fashion as to increase the velocity of the flow through it. The following assumptions have been taken for simplification to be made without introducing error in the analysis:

1. Steady one-dimensional flow
2. Adiabatic flow
3. No shaft work
4. The area on both the side of the shock may be considered the same.

3. METHODOLOGY

3.1 CFD Model

In order to create the CFD model of DC nozzle first, we need to create the project schematic in ansys workbench. CFD fluent workbench used for the modelling and analysis of Dc nozzle. First of all, we need to create the 2 D sketch of nozzle geometry according to the marc linares setup and convert it to the 2D plain geometry.

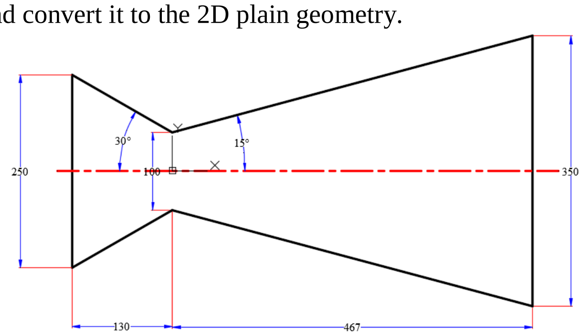


Fig. 2: The 2D geometry of DC conical Nozzle

The modelling has done in geometry workbench by using sketch and modelling tools. There are several commands available in geometry workbench by which we are able to create any complex geometry also same as other modelling software.

3.2 Meshing of geometry

In order to create the meshing of a geometry, we need to switch the workbench from geometry to meshing. After creating the geometry, it is required to divide the control volume into a smaller number of Nodes and element of finite size, therefore it is called a finite volume method. The method of splitting the Control volume into small finite size volume is known as a meshing of the control volume.

Table 1: Specification of nodes and elements

S. no.	Parameter	Size/Specification
1.	Advanced size function	Curvature
2.	Minimum Size	0.344680 mm
3.	Maximum Size	68.9360 mm
4.	Node	2937
5.	Element	2816

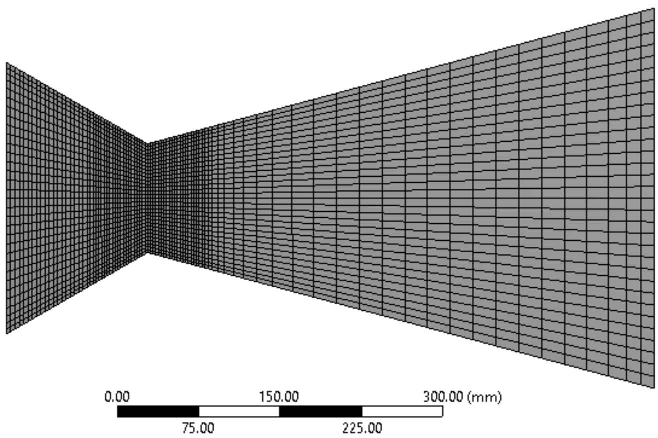


Fig. 2: Mesh Geometry of DC Nozzle

3.3 Boundary condition

The boundary condition provided to the DC Nozzle as per the actual working condition and scaled value given to the nozzle due to the limits of calculation and time. the boundary condition of the nozzle is given in the table below.

Table 2: Boundary condition of DC Nozzle

Boundary	Type
Inlet	Gauge total pressure 300000 (Pa), Temp. 300 k
Outlet	Gauge Pressure
Wall	Wall function
Surface	Interior Surface

4. RESULT AND DISCUSSION

4.1 Comparison of CFD and experimental result

In the first phase, we compare the result obtained by the analysis of DC nozzle with the experimental method referred to by Olivera P. Kostić[1]. The validation of CFD result has done by comparing the experimental result of macro [1] & Olivera P. Kostić [2] for the same geometry of DC nozzle. The experimental result was performed by taking the convergent and divergent angle about 40° and 20° angle. The similar geometry of DC nozzle was created in Ansys 15 and same boundary condition applied for the analyses. The inlet pressure for the nozzle is 3e5 pa and the outlet contour set the pressure outlet to find out the pressure ration in between the inlet and outlet.

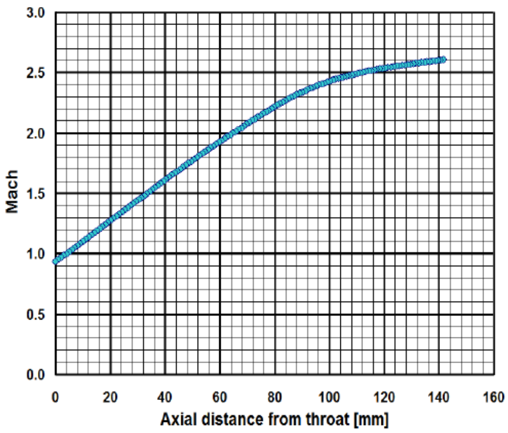


Fig. 4: Experimental result of Mach Number

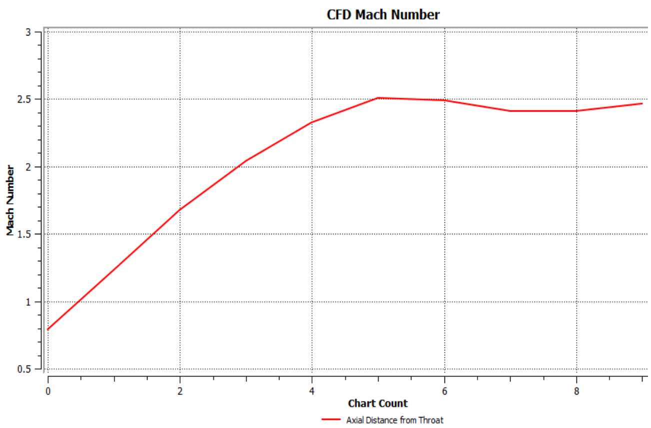


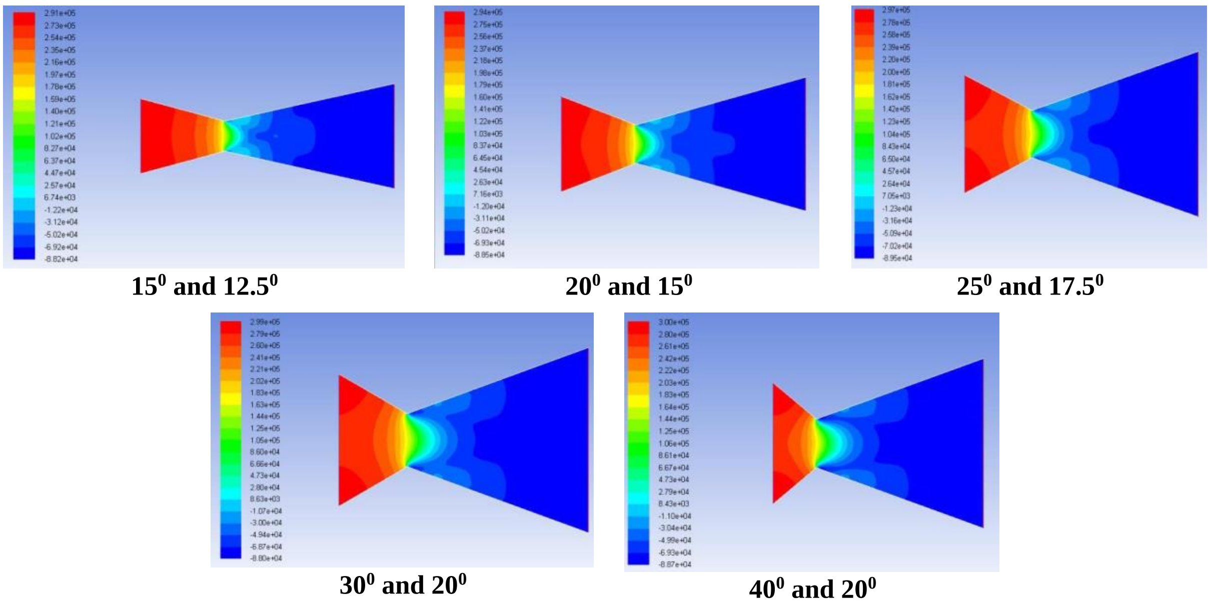
Fig. 5: CFD result of Mach Number

4.2 Optimization of divergent convergent angle

After the verification of CFD result, the next step was to optimize the geometry of DC nozzle by creating a different combination of inlet and outlet angle of the nozzle. For this purpose, we create the five different geometry of DC nozzle including the existing model of the nozzle. Each combination of geometry was analyzed in CFD workbench and mach number, pressure and velocity contour is taken as a design parameter to find out the effectiveness and feasibility of DC nozzle. Here all these parameters are taken as the output parameter while the divergent and convergent angle is taken as an input parameter. During the analysis of each set, the boundary condition is constant and same fluid media is used for the CFD workbench. The convergent angle varies from 15° to 40° and the divergent angle vary from 12.5° to 20°.

4.3 Pressure contour of DC Nozzle

The pressure distribution shown in the figure is a key example of the flow's performance based on the initial conditions set forth in ANSYS at 5 bar at the inlet and the outlet condition of an average static pressure of 1 bar. Again, as demonstrated, a maximum pressure is found near to the inlet location and goes down gradually with the length of the nozzle and its minimum at the outlet location for each case.



15^0 and 12.5^0 **20^0 and 15^0** **25^0 and 17.5^0**

30^0 and 20^0 **40^0 and 20^0**

Fig. 6: Pressure plot of DC nozzle

4.4 Mach number distribution in DC Nozzle

In the nozzle exit section of analysis results produced by the static pressure and Mach number contour of the CFD has to be corresponding design parameters. The supersonic condition found due to the value of Mach number at the exit location and its value is near about one at the throat location. At the inlet location, it’s less than or equal to the one which represents the subsonic condition. The maximum value of Mach no found in the third configuration of the parameter and it's about 30° and 15° at the convergent and divergent section.

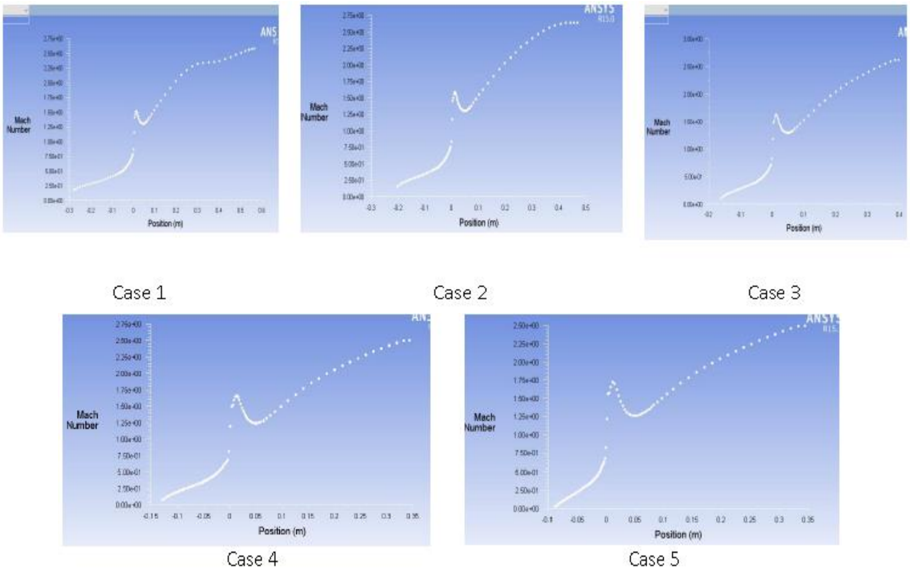


Fig. 7: Distribution of Mach number of DC Nozzle

5. CONCLUSION

The result obtained by the fluent is validated by the experimental result of Mr. Olivera P. Kostić [1]. The phase I consist of the result obtained by analysis of DC Nozzle of convergent angle 40° and divergent angle 20° only, after the examination of pressure, Mach number and temperature distribution, its concluded that it shows very good agreement with the experimental result of Mach no with respect to the nozzle axial length. The important conclusion can be summarized as follows:-

- It is observed that the experimental and CFD result of Mach number for DC Nozzle shows very good agreement for same boundary condition.
- In order to get the supersonic condition at the outlet, location preferred the convergent angle at 25° and 17.5° for the divergent angle.
- It is observed from pressure contour is maximum in inlet location and it’s falling down with the axial length of the nozzle.
- Similarly, the contour of velocity is low at the inlet location and it goes up with the axial length of the nozzle.
- The temperature distribution of air inside enclosed spaces is obtained with the help of temperature contours.
- The present results of pressure drop and Mach number for two-dimensional, Dc nozzle is obtained by using Computational Fluid Dynamics Software and compared with the results of Mr. Olivera P. Kostić [1].

4. REFERENCES

[1] Olivera P. Kostić,” CFD Modeling of Supersonic Airflow Generated by 2D Nozzle with and Without an Obstacle at the Exit Section”. FME Transactions, (2015) 43.
[2] Marc Linares,” Design Optimization of a Supersonic Nozzle”, florida university 2015.
[3] “Analysis of Flow In Convergent-Divergent Rocket Engine Nozzle Using Computational Fluid Dynamics” International Conference Of Scientific Paper Afases 2015.
[4] Ghasemi B.” Effects of Orientation of An Inclined Enclosure On Laminar Natural Convection” Heat and Technology, 43-49, 2005.
[5] Chang B.H. “Numerical Study of Flow and Heat Transfer in differentially heated enclosures” International Journal of Thermal Science, 18, 451-463, 2014.
[6] Corcione, M., “Effects of the Thermal Boundary Conditions at the Sidewalls upon Natural Convection in Rectangular Enclosures Heated from Below and Cooled from Above”, *International Journal of Thermal Sciences, 42*, 199-208, 2003.
[7] Elsherbiny S.M., Raithby G.D., Hollands K.G.T., “Heat transfer by natural convection across vertical and inclined air layers”, Journal of heat transfer, 1982, 104, 96-102.
[8] Emery A.P., MacGregor R. K., “Free Convection through vertical plane layer: moderate and high Prandtl number fluids,” Journal of Heat Transfer, 91, 391, 1969.
[9] F.P Incropera, D.P. Dewitt, Fundamentals of heat and mass transfer. Fifth edition. Wiley India.
[10] Fomichev, A., D.C. Curcija, B. Balagurunathan, and M. Stocki. Investigation of heat transfer effects of sloped and ventilated internal cavities of framing systems, Final report.Amherst: Center for Energy Efficiency and Renewable Energy, University of Massachusetts, 2007.
[11] Fusegi, T., Hyun, J. M., and Kuwahara, K., “Natural Convection in a Differentially Heated Square Cavity with Internal Heat Generation,” Numerical Heat Transfer, 1992, 21, 215–229.
[12] G. De Vahl Davis, “Natural convection of air in a square cavity: A benchmark numerical solution” International Journal for Numerical Methods in Fluids, 3, 249–264, 1983.

[13] Henderson D., Junaidi H., Muneer T. Grassie T., Currie J., "Experimental and CFD investigation of an ICSSWH at various inclinations", Renewable and Sustainable Energy Reviews, 11, 1087-1116, 2007.
[14] Hollands K.G.T., Unny T.E., Raithby G.D., Konicek L., "Free convective heat transfer across inclined air layers", Journal of heat transfer, 98, 189-193, 1976.
[15] Hollands K.G.T., Konicek L., "Experimental study of the stability of differentially heated inclined air layers", International Journal of Heat Mass Transfer, 16, 1467-1476, 1973.
[16] Kothandaraman C.P., Subramanian S., *Heat and mass transfer data book,* New Age International (P) limited.
[17] Lage, J. L., and Bejan, A., "The Resonance of Natural Convection in an Enclosure Heated Periodically From the Side," International Journal of Heat Mass Transfer, 36, 2027–2038, 1993.
[18] Lage, J. L., and Bejan, A, "The Ra-Pr Domain of Laminar Natural Convection in an Enclosure Heated From the Side," Numerical Heat Transfer, 19, 21–41, 1991.
[19] Manz Heinrich., "Numerical simulation of heat transfer by natural convection in cavities of facade elements," Energy and Buildings, 35,305–311, 2002.
[20] Nogueria R.M., Martins.M.A., Ampessan F0 .., "Natural convection in rectangular cavities with different aspect ratio".Engenharia Termica (Thermal Engineering), 10,44-49,2011.
[21] Ostrach, S., "Natural Convection in Enclosures," ASME Journal of Heat Transfer, 110, 1175–1190, 1988.
[22] Ozoe, H., Sayma, H. and Churchill, S. W. "Natural Convection in an Inclined Rectangular Channel at Various Aspect Ratios and Angles-Experimental Measurements", Int. J. Heat Mass Transfer, 18, 1425-1431, 1975.
[23] Pons Michel, "Transition from Single-to Multi-Cell Natural Convection of Air in Cavities with an Aspect Ratio of 20: A Thermodynamic Approach," Int. J. of Thermodynamics, 11, 71– 79, 2008.
[24] Rajput R.K., "Textbook of *Heat and Mass transfer"* S. Chand & Company Ltd.
[25] Soong, C. Y., "Numerical Study on Mode-Transition of Natural Convection in Differentially Heated Inclined Enclosure", *International Journal of Heat and Mass Transfer*, 2869-2882, 1996.

