

A Peer Reviewed Open Access International Journal

Modeling and Simulation of Supersonic Nozzle Using Computational Fluid Dynamics

Ms. K. Sree Lakshmi Assistant Professor, Aerospace Engineering MLRIT, Hyderbad.

ABSTRACT

Advances in rocket performance depend heavily upon improved and properly integrated propulsion system. This project provides a discussion about the design procedure of supersonic convergent-divergent (C-D nozzle). The C-D nozzles both conical and contour are designed on an assumption of the isentropic flow of the perfect gas. The computer code which uses the method of characteristics and the stream function to define high efficiency nozzle profile for isentropic, inviscid, irrotational supersonic flows of any working fluid for any user-defined exit Mach number. The designed nozzle area ratio is compared to theoretical area ratios for the selected fluid and desired exit Mach number. The nozzle geometry obtained from the code is independently checked with the commercial Computational Fluid Dynamics (CFD) code. ANSYS-FLUENT has been used to simulate flow on nozzle to verify the isentropic flow.

Critical to the design of a rocket system is the design of the exhaust nozzle. The analytical results of the nozzle are obtained through the isentropic flow assumption. The supersonic nozzles are modelled and are simulated using computational fluid dynamics (FLUENT) and ANSYS workbench which gives a numerical approach of the flow and is compared with the analytical results.

Introduction NOZZLE

The work outlined in this report is to design a supersonic convergent-divergent (CD nozzle). The C-D nozzles are designed on assumption of the isentropic flow of the perfect gas. A design procedure which can determine the configuration of C-D nozzle is shown by

Volume No: 3 (2016), Issue No: 9 (September) www.ijmetmr.com Mr. K. Venkatesh M.Tech Student, Aerospace Engineering MLRITM, Hyderbad.

arranging the experimental results using CFD (FLUENT).

The primary design of a rocket propulsion system order is to produce maximum thrust. Nozzle is an important and basic piece of engineering hardware associated with propulsion and the high speed flow of gases. In this chapter, the basic functions of a nozzle and a brief description of the kinds of nozzle are discussed. It also gives an overview of the basic concepts and the definition of CFD.



Figure 1.1: A typical Rocket Nozzle

Nozzle

A nozzle is a device that increases the velocity of a fluid at the expense of pressure. Nozzle is a part of rocket which is used for the expansion of combustion gases through it and produces thrust. Nozzle is a passage used to transform pressure energy into kinetic energy. During the combustion of fuel, chemical energy is converted into thermal energy and pressure energy. The combustion gases at this stage are at a



A Peer Reviewed Open Access International Journal

high pressure and temperature and these gases under such high pressure expand through the nozzle during which the pressure energy is converted into kinetic energy which in turn moves the vehicle in a direction opposite to that of the exhaust gases, according to Newton third law of motion. Aircraft exhaust nozzles serve two primary functions. First, they must control the engine back pressure to provide the correct, and optimum engine performance, which is accomplished through jet area variations. Second, they must efficiently convert the potential energy of the exhausting gas to kinetic energy by increasing the exhaust velocity, which is accomplished through efficiently expanding the exhausting gases to the ambient pressure.

Generally three types of nozzles are used namely convergent, divergent and convergent – divergent. To obtain a supersonic flow from a subsonic flow convergent – divergent nozzle is necessary. A variety of flow fields can be generated in the convergent – divergent nozzle by independently governing the back pressure downstream of the nozzle exit.



Figure 1.1.1: typical multiple Rocket Nozzle

Nozzle Configuration

Nozzles are chambers usually of circular cross – section and have a converging section, throat corresponding to the narrowest location and also a diverging section for supersonic nozzles.

The converging nozzle section between the chamber and the nozzle throat has been less critical in achieving high performance. The subsonic flow in this section can easily be turned at very low pressure drop and any radius, cone angle, wall contour curve, or nozzle. Inlet shape is satisfactory. A few small attitude control thrust chambers have had their nozzle 90 degrees from the combustion chamber axis without any performance loss. The throat contour also is not very critical to performance, and any radius or other curve is usually acceptable. The pressure gradients in these two regions and the flow will adhere to the walls. The principle difference in the different nozzle configuration is found in the diverging supersonic flow section. The wall surface throughout the nozzle should be smooth and shiny to minimize friction, radiation absorption, and convective heat transfer due to surface roughness.

Based on the nozzle configuration, they are classified as:

- Convergent
- Divergent
- Convergent-divergent or De-Laval
- Conical
- Contoured

Nozzle functions

The nozzle has two main functions as follows:

- The nozzle serves as an acceleration device converting gas thermal and pressure energies into kinetic energy.
- It provides required thrust and / or thrust vectoring.

COMPUTATIONAL FLUID DYNAMICS

In the fields of engineering and sciences, fluid dynamics is covered within three distinct subdisciplines, namely experimental; theoretical; and computational fluid dynamics (CFD). Computational Fluid Dynamics, known today as CFD, is defined as the set of methodologies that enable the computer to provide us with a numerical simulation of fluid flows. CFD is a relatively new field, it refers to the computational solution of fluid dynamics problems.



A Peer Reviewed Open Access International Journal

We use the word "simulation" to indicate that we use the computer to solve numerically the laws that govern the movement of fluids, in or around a material system, where its geometry is also modelled on the computer. Hence, the whole system is transformed into a "virtual" environment or virtual product. This can be opposed to an experimental investigation, characterized by a material model or prototype of the system, such as an aircraft or car model in a wind tunnel, or when measuring the flow properties in a prototype of an engine. This terminology is also referring to the fact that we can visualize the whole system and its behaviour, through computer visualization tools, with amazing levels of realism. Hence the complete system, such as a car, an airplane, a block of buildings etc. Can be "seen" on a computer, before any part is ever constructed. Computational fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. However, even with simplified equations and highspeed supercomputers, only approximate solutions can be achieved in many cases. More accurate software that can accurately and quickly simulate even complex scenarios such as transonic or turbulent flows are an ongoing area of research. Validation of such software is often performed using a wind tunnel. Thus the end product of CFD is indeed a collection of numbers, in contrast to a closed-form analytical solution.

Need for CFD

- Need to forecast performance.
- Cost or impossibility of experiments.
- Advances in solution algorithms.
- Advances in computer speed and memory.

Advantages

- Detailed information in space and time.
- Can simulate tests that cannot be done in laboratories.
- No scaling effects.

Limitations

- Results are only as good as the models employed particularly for complex turbulence.
- Results accuracy depends upon discretization level, boundary conditions and numerical precision.

NOZZLE DESIGN CONCEPTS

This chapter gives a main focus on the parameters required to design a nozzle. It gives the thermodynamic relations based on which the nozzle can be designed. The chapter also deals with the operation of a CD nozzle.

To design a nozzle the major requirement is the magnitude of thrust to be produced by the nozzle, the altitude at which nozzle operates and properties of propellant the used. In the design of the nozzle the main constraints of the adiabatic flame temperature and the total temperature at the inlet of the nozzle. The flame temperature is known by the type of propellant used and the pressure is obtained by rate at which the propellant is burned. The properties of the propellant to be known are its molecular weight and any one of the its specific heat at constant pressure or constant volume, specific heat ratio.

THRUST

The thrust is the force produced by the rocket propulsion system acting upon a vehicle. In a simplified way, it is the reaction experienced by its structure due to the ejection of matter at high velocity. It represents the same phenomenon that makes the gun recoil. The forward momentum of the bullet and the powder charge is equal to the recoil or rearward momentum of the gun barrel. Momentum is a vector quantity and is defined as the product of mass times velocity. All ship propellers and oars generate their forward push at the expense of the momentum of the water or air masses, which are accelerated towards the rear. Rocket propulsion differs from these devices primarily in the relative magnitude of the accelerated masses and velocities. In rocket propulsion relatively small masses are involved which are carried within the



A Peer Reviewed Open Access International Journal

vehicle and ejected at high velocities. The schematic of nozzle for thrust equation is shown figure.



Thrust = $\mathbf{F} = \mathbf{m} \mathbf{V}_{e} + (\mathbf{p}_{e} - \mathbf{p}_{0}) \mathbf{A}_{e}$

Figure 3.1: Thrust equation

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 a nozzle. It mainly consists of designing of a Conical and Contour nozzle.

DESIGN OF CONICAL NOZZLE

To design a nozzle the initial data to be known is the amount of thrust to be produced, properties of propellant used, the altitude at which the nozzle operates and the constraints such as total pressure, total temperature in the combustion chamber. As we get the ambient pressure as per standard conditions and hence the pressure ratio, we can find Mach number using equation (2.2) using specific heat ratio which we can get from propellant properties. Using Mach number we can get the temperature ratio, Density ratio and Area Ratio using the equations (2.1), (2.2), (2.4). To get the exit area we recollect the equation for mass flow rate

$$\dot{m} = \rho_e A_e v_e$$

To get the density at exit, we use the relation

$$P_0 = \rho_0 R T_0$$

As P_0 and T_0 are known, we get ρ_0 . The velocity at exit is from the definition of Mach number after

calculating the speed of sound by the following relation

$$c = \sqrt{\gamma RT}$$

 $M = V/c$

We know that thrust is given by the relation

 $F = \dot{m}V_e$

From the above relation, we get the magnitude of mass flow rate since the amount of thrust is already known and the exit velocity is obtained in the previous equation. As we found the values of density, velocity at the exit and mass flow rate, we can find the area at the exit by the mass flow rate relation as

$$A_e = \dot{m}/\rho_e V_e$$

Thus we can find the throat area also from area ratio. Hence we can get the dimensions to design the nozzle to produce the required amount of thrust.

DESIGN OF CONTOUR NOZZLE

A supersonic nozzle is used to transform parallel flow at sonic velocity into parallel, uniform flow at a supersonic Mach number. The conventional twodimensional supersonic nozzle consists of the following four main parts arranged in the direction of the flow:

- A subsonic inlet converging in the direction of the flow.
- A throat in which the streamlines are parallel to the nozzle axis and sonic velocity of the compressible flow is reached.
- An expanding part with constant or increasing angle of the inclination of the nozzle wall to the axis of the nozzle, in which flow accelerates to supersonic speeds.
- A straightening part of increasing area of cross section in the direction of flow but decreasing angle of inclination of the wall to the nozzle axis; in this part, the flow is turned parallel to the nozzle axis with the desired final Mach number uniform across the exit section.



A Peer Reviewed Open Access International Journal

Results and Discussion

Comparison of analytical results to the numerical results

If we compare the analytical results with the numerical results at exit of both the nozzles conical and contour, the resulted values are as shown below.

Table 6.1: Comparison of results

Analytical results	Numerical results
Me (for conical) = 2.526	Me (for conical) = 2.04e
Me (for contour) = 3.154	Me (for contour) = 2.64e
Pe (for conical) = 89457	Pe (for conical) = 90000 pascals
Pe (for contour) = 98547	Pe (for contour)=99955 pascals
Te (for conical) =165	Te (for conical) = $1.6e+02$
Te (for contour) = 125	Te (for contour) =1.25e+02
Density (for conical) = 2.2	Density (for conical) =2.54e-01
Density (for contour) = 1.2	Density (for contour) = 1.3e-01

Contours: Conical Nozzle

The pressure contours as shown in 5.8 gives us the variation of static pressure across the nozzle. The pressure decreases from inlet to outlet of the nozzle, during which pressure energy is converted into kinetic energy. This can be seen mach contour referring to 5.9. In converging section the velocity increases and mach number reaches 1 at the throat and it increases in the divergent section until the exit of the nozzle at the expensive of pressure and temperature. We can also use the variation of static pressure along the nozzle as shown in 5.10. The temperature at the inlet is maximum because the combustion gases are high temperature and it decease along the nozzle due to expansion.

Contours: Bell nozzle

The variation of static pressure, temperature and mach number are shown in figures below. In contour nozzle, the loss of thrust component is less when compared to conical nozzle and this can be seen in mach number contour that mach number is maximum at axis of exit section. The velocity is maximum at the axis and it decreases as we move towards wall. The variation of static temperature is minimum at the axis of exit section than the wall.

Calculation of Tabulated data



Figure 6.15: Symmetric diagram of convergentdivergent nozzle





Graph between X-Y coordinates



Graph between X- AREA





September 2016



A Peer Reviewed Open Access International Journal



Graph between X-TEMPERATURE



Graph between X-DENSITY



Graph between X- VELOCITY



Graph between X-SPEED OF SOUND



Graph between X-THRUST



Graphs of both the Nozzles

Conclusions

1. A Convergent-Divergent nozzle is designed on an assumption of Quasi-One dimensional isentropic flow. Conical nozzle has been designed in the modelling software. Along with this conical nozzle, a designed contour nozzle has been analyzed using CFD (FLUENT).

2. Fluent is utilised to simulate the transient gas flow by a coupled explicit solver and it gives a 2-D result. An overall first order and second order scheme is employed spatially and temporally. Simulated Pressure histories, Temperature histories and Mach number distributions agree well with the corresponding reported static and pilot pressure measurements. Comparison of the Pressure ratio, Density ratio, Temperature ratio and Mach numbers are done between the analytical and Fluent output.

3. From the report, it can be observed that the Contour nozzle gives a greater expansion ratio comparatively to a conical nozzle. Thus a Conical nozzle has to be used at sea-level and a Contour nozzle has to be used at a higher altitude since greater expansion ratio is required at an higher altitude for a given length. A conical nozzle has a simple geometry and easy to fabricate,



A Peer Reviewed Open Access International Journal

whereas a Contour nozzle has a complex geometry and is difficult to fabricate.

REFERENCES

[1]. Anderson JD, [2001], Fundamentals of Aerodynamics, 3rd Edition, pp.532-537, pp.555-585.

[2]. Anderson JD, [1982], Modern Compressible Flow with Historical, pp.268-270,pp.282-286.

[3]. Shapiro, AH, [1953], The Dynamics and Thermodynamics of Compressible Fluid Flow, Vol.1, pp. 294-295.

[4].Shapiro, AH, [1953], The Dynamics and Thermodynamics of Compressible Fluid Flow, Vol.2, pp. 694-695.

[5]. Farley John M , Campbell Carl E, [1960], Performance of several characteristics exhaust nozzles, NASA Lewis Research center.

[6]. Angelino, G., Oct. [1964], "Approximate Method for Plug Nozzle Design", AIAA Journal, Vol. 2, No. 10, pp. 1834-1835.

[7].J.Reid, [1964], An Experiment on Aerodynamics nozzles at Mach 2, Aeronautical research council reports & memoranda, R & M. No 3382.

[8]. G T Galesworthy, JB Robert and C Overy [1966], the Performance of conical convergent divergent nozzles of area ratio 2.44 and 2.14 in external flow. Aeronautical Research Council CP No 893.

[9]. TV Nguyen and JL Pieper [1996], Nozzle separation prediction techniques and controlling techniques, AIAA paper.

[10]. Gerald Hagemann, [1998], Advanced Rocket Nozzles, Journal of Propulsion & Power, Vol.14, No.5.

[11]. Dale, D., Kozak, J., Patru, D., [2006], "Design and Testing of a Small Scale Rocket forPico- Satellite

Launching", Rochester Institute of Technology METEOR Project, SeniorDesign Project 06006.

[12] Nicholas J Georgiadis, Teryn W DalBello, Charles J Trefny, and Albert L. Johns,[2006],Aerodynamic Design and Analysis of high performance nozzles for Mach 4 Accelerator Vehicles.

[13]. Taro Shimizu, Manatoshi Kodera, Nobuyunki Tsuboi, [2008], Internal & External flows of rocket nozzle, Journal of Earth Simulator, Vol 9, pp.19-26.

[14]. T.S.Leu, C.T.Wang, J.M.Sun, [2010], Optimal design & operation on convergent divergent nozzle type no moving parts valves in micro channel, Journal of Mechanics, Vol 26 No.3.

[15]. G Satyanarayana, Ch Varun, SS Naidu, [2013], CFD Analysis of Convergent Divergent Nozzle, ACTA Technica Corviniensis Bulletin of Engineering Tome VI Fascicule 3, ISSN 2067 – 3809.

[16]. Bijju Kuttan P, M Sajesh, [2013], Optimization of divergent angle of a rocket engine nozzle using CFD, THE IJES Vol 2 issue 2 pp. 196-207.

[17]. Balaji Krushna P, P Srinivas Rao, B Bala Krishna, [2013], Analysis of Dual Bell Rocket Nozzle using CFD, IJRET, Vol.2 issue 11.

[18]. Gutti Rajeswara Rao, [2013], Flow analysis in a convergent divergent nozzle using CFD, IJRME, ISSN Online: 2347-5188.

[19]. Nazar Muneam Mahmood, [2013], Simulation of back pressure effect on behaviour of convergent divergent nozzle, Diyala Journal of Engineering Sciences, Vol. 06 No. 01 pp 105-120.

[20]. Nikhil d Deshpande, [2014], Theoretical & CFD Analysis of De Laval Nozzle, IJMPE, ISSN 2320-2092

Volume No: 3 (2016), Issue No: 9 (September) www.ijmetmr.com



A Peer Reviewed Open Access International Journal

[21]. Ralf H.Stark, Flow Seperation in Rocket Nozzles-A simple criteria, AIAA, pp.1-8.

[22]. Sutton GP, Rocket Propulsion Elements, 7th Edition.

Volume No: 3 (2016), Issue No: 9 (September) www.ijmetmr.com

September 2016