CHEMICAL ENGINEERING TRANSACTIONS

VOL. 66, 2018

A publication of AIDIC

The Italian Association of Chemical Engineering
Online at www.aidic.it/cet

Guest Editors: Songying Zhao, Yougang Sun, Ye Zhou

ISBN 978-88-95608-63-1; **ISSN** 2283-9216

DOI: 10.3303/CET1866132

Numerical Investigation on Effect of Divergent Angle in Convergent-Divergent Rocket Engine Nozzle

Raghu Ande, Venkata N. Kumar Yerraboina*

Department of Mechanical Engineering, B.V Raju Institute of Technology, Narsapur 502313, India
raghu.a@bvrit.ac.in

In rocket, a nozzle is used to control mass flow rate, velocity distribution and pressure of the exhaust gas that emerges out from the combustion chamber. The nozzle is used to alchemize the chemical and thermal energy generated in the combustion chamber into thrust. The nozzle metamorphoses the high pressure, high temperature, low velocity gas in the combustion chamber into high velocity, low pressure and temperature. The design of nozzle is an important aspect for achieving the maximum Mach number and minimum turbulent intensity. In this study, the numerical investigation is done at different divergent angles to find the effect of divergent angle on Mach number and static pressure. The different divergent angles used for analysis are 9°, 12°, 15° and 18°. The inlet boundary conditions were specified according to the available experimental information. The results evaluated and compared with the help of different contours and graphs to figure out the optimum divergent angle with maximum Mach number.

1. Introduction

The convergent – divergent nozzle is the most common type used in rocketry and it works by converting pressure energy from the fuel flow and heat energy from the combustion of fuel into kinetic energy in the form of high exhaust velocity. In the converging section of a rocket nozzle, the exhaust is travelling at relatively low speed (sub-sonic) and it becomes sonic at throat. The compressible exhaust increases until it reaches the exit and it is supersonic in the divergent section. Size and shape of a rocket nozzle is also very important. the converging section starts at the combustion chamber is usually shaped in a way to make sure that flow is not disrupted in any way i.e., the convergence is not too steep and has no harsh edges. The size of the throat is determined by certain characteristics of engine such as chamber pressure of combustion chamber and chemistry of the exhaust gas. The shape of the divergent section depends on the expansion ratio and amount of required thrust. The ratio of area of exit to the area of throat section is called expansion ratio. Area of exit varies by varying the divergence angle.
Numerous research works have been done on design, fluid flow and effect of various parameters on rocket engine nozzle. Bogdam et al. (2015) carried out the study of flow through the convergent divergent nozzle by with the help of finite volume rewarding code, ANSYS Fluent. They performed nozzle geometry analysis by using rocket nozzle equations and calculated the values for nozzle geometry. A simple analysis was done on the geometry model and observed that nozzle created based on the exit parameters is in accord with the scope. Safayet et al. (2014) performed a comparative flow analysis of two different nozzles. They did the analysis by keeping the same input parameters and varying the shape of nozzle. Velocity, pressure and temperature distribution on both nozzles have been studied. Du et al. (2008) did the computational investigation on the nozzle of orifices distributing in different space layers. Yu et al. (2012) performed the Reynolds-Averaged Navier Stokes simulations to investigate the effect of nozzle geometry on the turbulence characteristics of incompressible fluid flow through nozzle. They observed that Reynolds stress model shows significant improvement over the standard one equation and two equation modes for the prediction of turbulent flow parameters. Mesh topology also plays a major role in determining the accuracy of the results. Hussain et al. (2010) experimental investigated the performance characteristics of four different axi-symmetric contraction

Please cite this article as: Ande R., Yerraboina V.N.K., 2018, Numerical investigation on effect of divergent angle in convergent-divergent rocket engine nozzle, Chemical Engineering Transactions, 66, 787-792 DOI:10.3303/CET1866132

nozzle shapes with the same contraction ratio and reveled that all the nozzles are of essentially equal effectiveness as long as the core flow in the exit plane is concerned. Pandey et al. (2010) conducted a numerical study to investigate the supersonic flow in conical nozzle for Mach 3 at various degree of angle. Kunal et al. (2010) has presented the analysis of performance and flow characteristics of convergent-divergent nozzle and calculated the strength of shock in the divergent portion of the nozzle under varying conditions and with different nozzle geometry.

2. Design and modelling of rocket engine nozzle

The design of the nozzle is formulated by using the following formulae available in the literature survey.
Mass flow rate in the rocket is calculated by

Table 1: Formulae to calculate mass flow rate

$\dot{m} = \frac{\dot{F}_{thrust}}{v_e}$	$\boldsymbol{v_e}$ Indicates the velocity of exhaust gases
Exhaust velocity is given by	$v_e = \sqrt{\frac{2\gamma}{\gamma - 1} . \frac{RT_0}{M} . (1 - (\frac{p_e}{p_0})^{\frac{\gamma-1}{\gamma}}}$
Expansion ratio in Nozzle is given by	$\varepsilon = \frac{A_e}{A_t}$ Where $A_t = \frac{\dot{m}}{P_c \sqrt{\gamma (\frac{2}{\gamma+1})^{\frac{\gamma+1}{\gamma-1}} \frac{M}{R_u T_c}}}$ $A_e = \pi {r_e}^2$
Convergence area in nozzle is given	$A_c = 3A_t$
Radius of throat is given by	$r_t = \sqrt{\frac{A_t}{\pi}}$
Length of the diverging nozzle is	$L_{dn} = \sqrt{\frac{A_c}{\pi}} \frac{1}{tan\theta}$
Length of the converging nozzle	$L_{cn} = \sqrt{\frac{A_c}{\pi}} \frac{1}{tan\beta}$

From the above equations, the dimensions of the rocket nozzle are calculated with the help of experimental inlet conditions and are tabulated. A 2-D modeling of rocket engine nozzle is obtained and then by changing the divergent angle to 9, 12, 15 and 18 degrees and design of rocket engine nozzle is created by using Solid works. Further the designs meshed by using Ansys and further analysis is done by using Ansys Fluent.
A 2-D numerical simulation is done using the commercial CFD software ANSYS FLUENT. A Density based approach is taken, a Steady solver using an implicit equation and SIMPLE algorithm is used, control-volume-based finite volume scheme to discrete the conservation equations. The discretized finite volume equations yield a set of linear simultaneous equations for all cells which is solved through a least-square cell method. As the problem deals with the flow the energy equation is considered.

3. Results and discussions

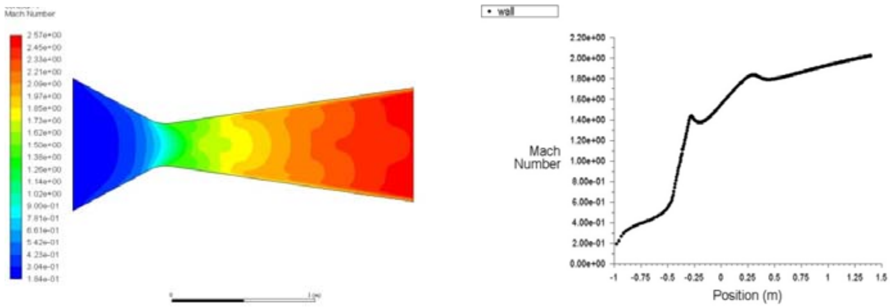
By using SOLIDWORKS, four different dimensional designs are modeled by changing the divergent angle and then CFD analysis is done for four different models and the variation in Mach number and static pressure is being observed in each case.

3.1 Case 1

Divergent angle taken is 9 degrees and analysis is done.

3.1.1 Mach number

From the contour of Mach number for 9° divergent angle, it can be observed that the velocity distribution is sub-sonic i.e., 1.84e-01 at inlet. The Mach number increases as the exhaust moves from the inlet to the throat. Across the throat, the Mach number varies from 5.42e-01 to 9.00e-01. The velocity distribution at the exit is supersonic (2.57e+00), which is required to get the thrust.

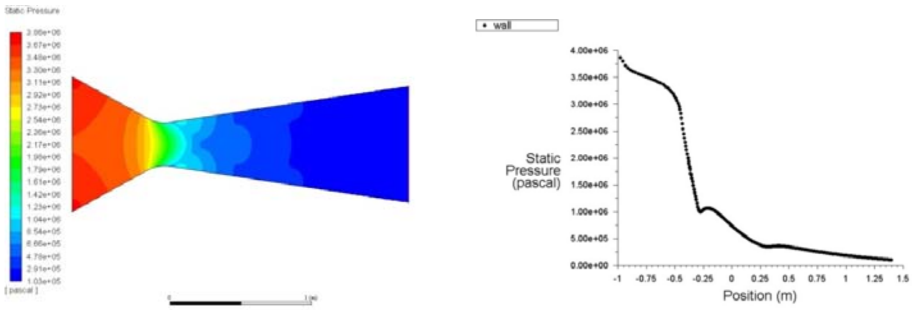


(a) (b)

Figure 1: (a) Contour of Mach number for 9° diverging angle; (b) Position (m) vs Mach number

3.1.2 Static pressure

The high pressurized combusted gases from the combustion chamber enter the inlet of convergent section with pressure 3.86e+06 Pa. The static pressure declines as it moves across the throat from 2.92e+06 to 1.61e+06 Pa. The analysis reveals the evidence that the fluid gets expanded in the nozzle exit. At the exit, the static pressure is very low of magnitude 1.03e+04 Pa.



(a) (b)

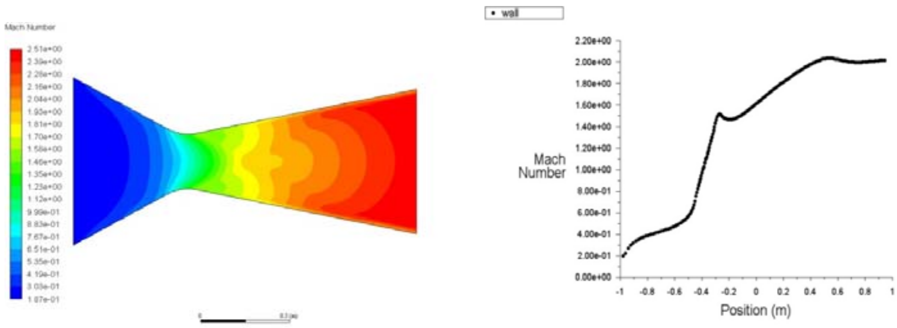
Figure 2: (a) Contour of Static Pressure for 9° diverging angle; (b) Position (m) Vs Static Pressure (Pascal)

3.2 Case 2

Divergent angle taken is 12 degrees and analysis is done.

3.2.1 Mach number

From the contour of Mach number for 12° divergent angle, it can be observed that the velocity distribution is sub-sonic i.e., 1.87e-01 at inlet. The Mach number increases as the exhaust moves from the inlet to the throat. Across the throat, the Mach number varies from 6.51e-01 to 9.99e-01. The velocity distribution at the exit is supersonic (2.51e+00), which is required to get the thrust.

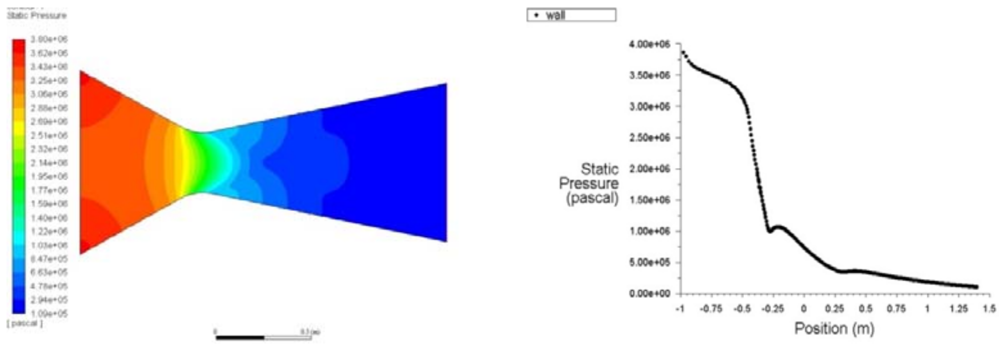


(a) (b)

Figure 3: (a) Contour of Mach Number for 12° diverging angle; (b) Position (m) Vs Mach Number

3.2.2 Static pressure

The high pressurized combusted gases from the combustion chamber enter the inlet of convergent section with pressure 3.80e+06 Pa. The static pressure declines as it moves across the throat from 2.88e+06 to 1.59e+06 Pa. At the exit, the static pressure is very low of magnitude 1.09e+05 Pa.



(a) (b)

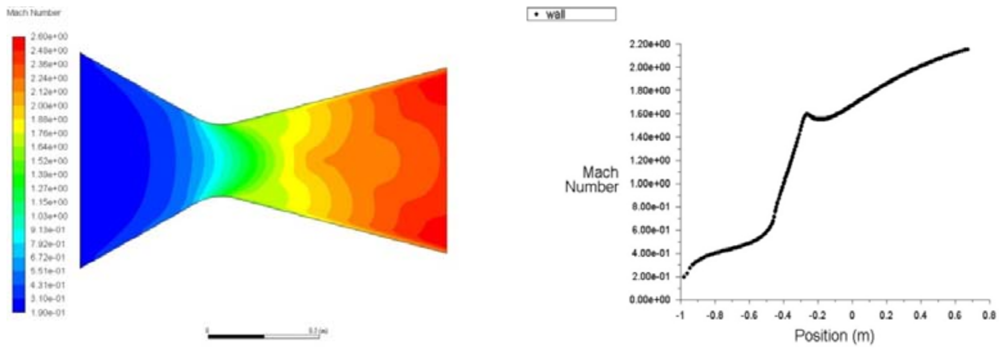
Figure 4: (a) Contour of Static Pressure for 12° diverging angle; (b) Position (m) Vs Static Pressure (Pascal)

3.3 Case 3

Divergent angle taken is 15 degrees and analysis is done

3.3.1 Mach number

From the contour of Mach number for 15° divergent angle, it can be observed that the velocity distribution is sub-sonic i.e., 1.90e-01 at inlet. The Mach number increases as the exhaust moves from the inlet to the throat. Across the throat, the Mach number varies from 6.72e-01 to 1.03e+00. The velocity distribution at the exit is supersonic (2.60e+00), which is required to get the thrust.



(a) (b)

Figure 5: (a) Contour of Mach Number for 15° diverging angle; (b) Position (m) Vs Mach Number

3.3.2 Static pressure

The high pressurized combusted gases from the combustion chamber enter the inlet of convergent section with pressure 3.74e+06 Pa. The static pressure declines as it moves across the throat from 2.83e+06 to 1.55e+06 Pa. At the exit, the static pressure is very low of magnitude 8.53e+04 Pa.

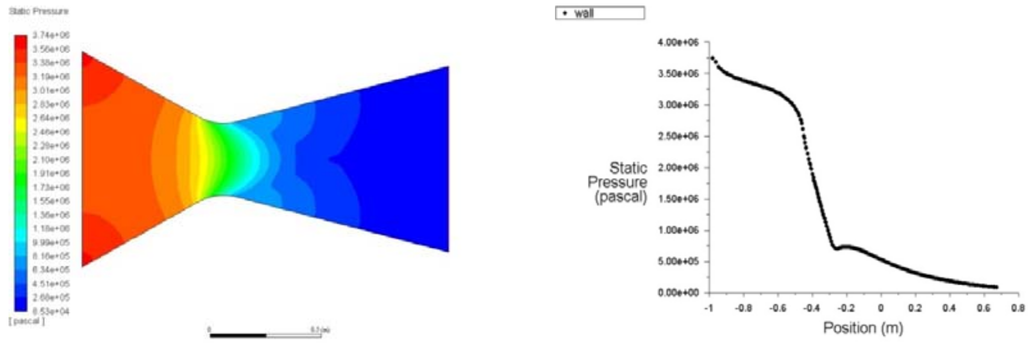
3.4 Case 4

Divergent angle taken is 18^{o} and analysis is done.

3.4.1 Mach number

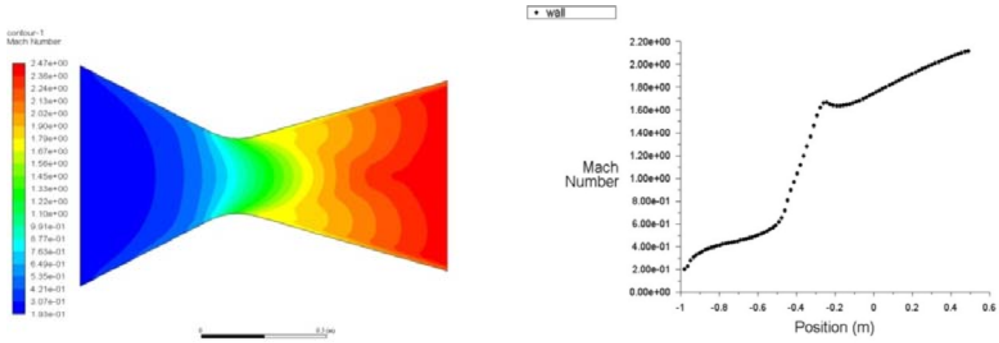
From the contour of Mach number for 18° divergent angle, it can be observed that the velocity distribution is sub-sonic i.e., 1.93e-01 at inlet. The Mach number increases as the exhaust moves from the inlet to the throat.

Across the throat, the Mach number varies from 7.63e-01 to 1.10e+00. The velocity distribution at the exit is supersonic (2.47e+00), which is required to get the thrust.



(a) (b)

Figure 6: (a) Contour of Static Pressure for 15° diverging angle; (b) Position (m) Vs Static Pressure (Pascal)

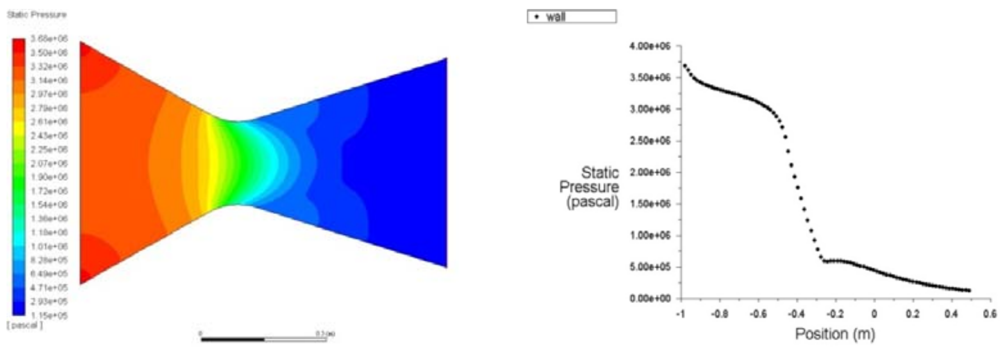


(a) (b)

Figure 7: (a) Contour of Mach Number for 18° diverging angle; (b) Position (m) Vs Mach Number

3.4.2 Static pressure

The high pressurized combusted gases from the combustion chamber enter the inlet of convergent section with pressure 3.68e+06 Pa. The static pressure declines as it moves across the throat from 2.79e+06 to 1.36e+06 Pa. At the exit, the static pressure is very low of magnitude 1.15e+05 Pa.



(a) (b)

Figure 8: (a) Contour of Static Pressure for 18° diverging angle; (b) Position (m) Vs Static Pressure (Pascal)

3.5 Exit conditions

Table 2: Mach number and static pressure at nozzle exit

Case	Diverging Angle	Mach number	Static Pressure (Pa)
1	9	2.57e+00	1.03e+05
2	12	2.51e+00	1.09e+05
3	**15**	**2.60e+00**	**1.01e+05**
4	18	2.47e+00	1.12e+05

4. Conclusions

The outcomes from the analysis on the rocket nozzle with varying divergent angle are as follows

- At the divergent section, the velocity distribution is found to be increasing with increase in divergent angle. It is 2.57e+00 Mach for 9 deg and it increased to a highest value of 2.6e+00 Mach for 15°.
- The static pressure decreased with increase in divergent angle at exit section. It is 1.03e+05 for 9 deg and decreased to 1.01e+05 for 15°.
- Mach number of optimum value is obtained at divergent angle of 15°

References

Bogdan-Alexandru B., 2015, Analysis of Flow in Convergent Divergent Rocket Engine Nozzle Using Computational Fluid," International Conference of Scientific Paper AFASES 2015 Brasov, 28-30.

Du H., Liu J., Tang J., 2008, A CFD Investigation on the Nozzle of Orifice Distributing in Different Space Layers, SAE International, SAE World Congress and Exhibition, USA.

Hussain Akmf J., Ramjee V., 2010, Effects of the Axi-symmetric Contraction Shape on Incompressible Turbulent Flow, Journals of Fluid Engineering, ASME, USA, 98, 56-68.

Kimura T., 2009, Experimental and Analytical Study for Design of Dual Bell Nozzles, Proceedings of the 45th AIAA/ASME/SAE/ASEE JPC and Exhibit, Denver, Colarado, AIAA.

Kunal Pansari S.A.K., 2005, Numerical Investigation of the Performance of Convergent Divergent Nozzle, International Journal of Modern Engineering Research (IJMER), 3(2), 2662-2666.

Pandey K.M., Singh A.P., 2010), CFD Analysis of Conical Nozzle for Mach 3 at Various Angles of Divergence with Fluent Software, International Journal of Chemical Engineering and Applications, 1(2).

Pandey K.M., Yadav S.K., 2010, CFD Analysis of a Rocket Nozzle with Two Inlets at Mach 2.1, Journal of Environmental Research and Development, 5(2): 308-321.

Pardhasaradhi N., 2012, Flow Analysis of Rocket Nozzle Using Computational Fluid Dynamics (CFD), International Journal for Engineering Research and Applications (IJERA), 2(5), 1226-1235.

Safayet Hossain D., 2014, Comparative Study of Supersonic Nozzles, International Journal of Research in Engineering and Technology, volume 3, pages 351-357, DOI: 10.15623/ijret.2014.0310056

Yu Y., Shadema M., 2012, CFD study of Effect of Geometry Variations on Flow in a nozzle, Engineering Application of Computational Fluid Mechanics, 6(3), 412-425, DOI: 10.1080/19942060.2012.11015432