Sindh Univ. Res. Jour. (Sci. Ser.) Vol.49 (004) 727-732 (2017)

http://doi.org/10.26692/sujo/2017.12.0048

SINDH UNIVERSITY RESEARCH JOURNAL (SCIENCE SERIES)

Influence of Convergent Section Length and Angle on Performance of Supersonic Nozzle

I. MIR++, S. SAMO, T. HUSSAIN, I. ALI, H. A. K. DURANI*

Department of Mechanical Engineering, Mehran UET, Jamshoro

Received 6th March 2017 and Revised 12th October 2017

Abstract: Performance of propulsion system is the key parameter in getting efficient flying vehicles. The exhaust nozzle is highly important component of propulsion system, in order to improve nozzle performance understanding of geometrical parameters is mandatory. The nozzle design need optimization for various parameters like inlet, throat and exit width, but impact of inlet convergent angle and length have got less attention. In this research effect of convergent section length and angle of supersonic nozzle were analyzed by keeping same expansion ratio and boundary conditions. The numerical simulation was made on unstructured gird. The computational work is carried out on 2D axi-symmetric density based, coupled solver with viscous $k-\omega$ SST turbulence model and linearized through implicit scheme. Study reveals that 28.5° inlet angle would give best result and maximum thrust force would be produced at this angle. From simulation results it was analyzed that flow behavior inside the nozzle is highly complex nature.

Keywords: Convergent divergent nozzle, Supersonic flow, Thrust performance, computational Fluid dynamics.

1. INTRODUCTION

Since aircrafts consume exhaustible, polluting and high price fossil fuels. Therefore its efficient design would be highly important to meet global energy and environment requirement. The propulsion system is highly significant part that is responsible power generation and flight of aircraft. Efficiency of propulsion system is greatly affected by nozzle performance. Nozzle is designed to control the rate of flow, speed and pressure as well as to convert heat energy into kinetic energy. Nozzle transforms high pressure and high temperature flue gases into high velocity, low temperature and low-pressure gases. Main purpose of nozzle is to reduce pressure to possible low level and to exhaust gases at very high velocity. This result increase in rearward momentum which in turns enables heavier machines to fly in air. Usually nozzle with fixed convergent section followed by fix divergent section are used in aircrafts, aerospace shuttles, Ramjets, scramjets and in rocket engines (Boyanapalli, 2013) (Benson., 2013). The number of research has been conducted to understand effect of nozzle geometry to enhancing nozzle performance e.g (Dusa,1989) (Antipas, *et al.*, 2011). Outcomes of these investigations gave the new concept of divergent portion to the convergent nozzle, which made it possible to get supersonic exhaust from Convergent-divergent nozzle. In ref (Bayt, and Breuer. 1998) (Khattab, and Barakat, 2002) further illustrated that divergent portion further increase flue gas velocity by causing increase in volume which in turns causes further drop in density by increasing velocity in supersonic range.

The nozzle geometry is highly important because it directly affect the overall performance of propulsion system. Particularly the mean fluid velocity field, pressure as well as turbulent characteristics plays strong role in occurrences of many physical processes inside the nozzle. In propulsion system, main function of supersonic nozzle is to provide thrust force. Whereas thrust generated by nozzle is the function of many geometric parameters such inlet area, expansion ratio, nozzle pressure ratio as well as lengths of convergent-divergent sections etc.

Due to the number of applications of Convergent-divergent nozzle numerous research was conducted to improve nozzle aerodynamic performance. In ref (Grisnik, *et al.*, 2013) (Pearson, *et al.*, 1996) authors conducted research to analyze losses occurs inside the nozzle. Their Research concluded that there are basic three main contributors which enhance losses inside the nozzle which are viscous losses, chemical kinetic loss and other is divergence losses. Their research also illustrated that variation in nozzle divergence portions highly affects viscous losses. To predict the effect of nozzle divergence angle, His research reveals that decrease in divergence angle from 45° to 20° give rise another desirable affect by reducing chemical kinetic loss. His research also concluded that increase in

++Corresponding author: Email: tanweer.hussain@faculty.muet.edu.pk
*Department of Mechanical Engineering, Mehran UET ZAB, Khairpur

divergence angle up-to 65° reduce divergence losses to zero but in turns it increase weight of nozzle which is not practical suitable. In ref. (Ketsdever, 2005) author illustrated that exit flow velocity can be increased by reducing length of nozzle. Another finding of this research was that impulse force can be increased by increasing divergence angle from 20° to 40° with minute viscous effects. These numerical results also verifies the previous experiments conducted by Whalen (Whalen,1987)], where it was found that the conical nozzle with 25° divergence angle have better performance than conical nozzle with 20 divergence angle. In ref. (Noh, 2011) author conducted research to analyze the effect of divergence length where it is concluded that while reducing angle from 28 to 24 thrust decrease by 18% whereas for divergence angle variation from 18 to 16 degree causes 3% increase in thrust. His research also found that separation can't be eliminated. Study was carried out to analyze the effect of divergence angle on thrust coefficient. Recent research shows that divergence angle has significant effect on the discharge coefficient for the nozzle having throat diameter less the 1mm and operating at atmospheric conditions (Kim, *et al.,* 2010).

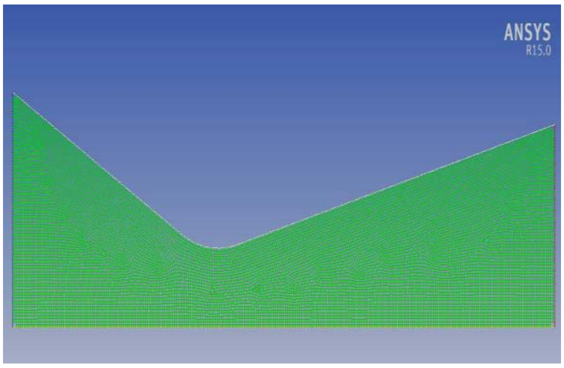

Fig. 1 Meshed model of axisymmetric CD nozzle

From literature, it is observed that still nozzle design was optimized for various parameters like inlet, throat and exit width, but impact of inlet convergent angle and length have got less attention. Hence present study aims to analyze the effect of nozzle convergent section length and angle by keeping same expansion ratio and boundary conditions. Present study mainly focuses on the convergent section effect on the thrust performance of supersonic nozzle.

2. NUMERICAL FEATURES

2.1 Nozzle Modeling and Meshing

In order to investigate the effect of convergent section length nozzle is modeled in ANSYS design modeler. Nozzle design specifications are given in **(Table. 1)** those parameters are considered as base parameters for nozzle design optimization. In this research number of 2D model of convergent divergent nozzle were designed at different length of convergent section and corresponding convergent angles. 2D model was further processed for gird generation in ANSYS ICEM. Hybrid meshing was done to improve mesh quality and divide physical domain into number of small sections named as cells. High quality mesh was generated to improve solution accuracy through applying mesh refinement and inflation. Inflation was applied on the divergent section wall so that flow separation and boundary layer effects can be clearly visualized

Table.1. Shows the design specification of supersonic nozzle

Parameter	Specifications
Inlet Diameter	1000mm
Throat Diameter	304mm
Exit Diameter	861mm
Curvature radius at throat	228mm
Convergent angle	30°
Divergent Angle	15°
Pressure at inlet	44.10 bar
Temperature at inlet	3400K
Inlet mass flow rate	860Kg/s

2.2 Governing Equation

The fluid flow governing equations like continuity equation, momentum equation and energy equations are derived through applying the basic conservation principal of mass, momentum as well as energy. Those equation are Partial Differential equations (PDE) which difficult to solve analytical or numerically because they take into account the small fluctuation in fluid properties. Therefore, theses complex equations are simplified through applying Reynolds averaging. By using averaging velocity of fluid in X-direction is

$$u_i = \bar{u}_i + u_i'$$

Whereas $\bar{u}_i$ and u_i' represents average and fluctuating components of velocity respectively.
Similarly, for any other scalar quantity

$$\rho = \bar{\rho}_i + \rho_i'$$

As in above expression density is decomposed into average and fluctuating components. By applying Reynolds averaging technique following equations are obtained in terms time averaged components.

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad (1)$$

$$\frac{\partial}{\partial t}\left(\rho u_i\right)+\frac{\partial}{\partial x_j}\left(\rho u_i u_j\right)=\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}+\frac{2}{3}\delta_{ij}\frac{\partial u_i}{\partial x_i}\right)\right]+\frac{\partial}{\partial x_j}\left(-\rho\overline{u_i'}\overline{u_j'}\right) \quad (2)$$

The Eq. (1) and (2) are known as Reynolds average Navier-Stokes equations. Additional terms in above equation are encountering the effect of turbulences. The last term $\left(-\rho\overline{u}_i'\overline{u}_j'\right)$ in Eq. (2) is called Reynolds stresses. Reynolds stress $\left(-\rho\overline{u}_i'\overline{u}_j'\right)$ terms should be modeled by selecting appropriate turbulence model to get accurate results (Wilcox, *1998*). To select best turbulence model various researches are conducted which come know that $k-\omega$ model provides good results then $k-\varepsilon$ model for near wall problems (Ferreira, 2007) (Wang, 2010). Though $k-\omega$ model is better but its great sensitivity for the ω values near ir-rotational boundaries creates problem in case of shear flows. But this problem was solved by Menter's $k-\omega$ SST model which combines merits of $k-\omega$ and $k-\varepsilon$ model for near the wall and away from wall flows (Dai, *et al.*, 2010) The Shear Stress Transport (SST) model proved to be very helpful for encountering the effects in boundary layer and flow separation regions. Their transport equations of $k-\omega$ SST model are (Kumar, and Chatterjee, 2008)

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}(\Gamma k\frac{\partial k}{\partial x_j})+\tilde{G}_k-Y_k+S_k \quad (3)$$

$$\frac{\partial(\rho\omega)}{\partial t}+\frac{\partial(\rho\omega u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}(\Gamma\omega\frac{\partial\omega}{\partial x_j})+\tilde{G}_\omega-Y_\omega+D_\omega+S_\omega \quad (4)$$

2.3 Gird Adoption

The accuracy of numerical solution is highly dependent on gird quality; therefore, to enhance gird quality in region of high pressure gradient is extremely important for getting accurate results at lower computational cost. This technique begins with entire computational domain which is meshed with coarse cells and faces, then all the individual cells are selected for refinement by using user defined criteria or on the basis of Richardson-extrapolation. In this process, all the coarse cells are refined to finer cell one. After mesh adaption, individual mesh cells on a single predetermined level of refinement are passed off for advancement of time by integrator. At the end, correction process is employed to perform transfer of conserved parameters between the interfaces of coarse and fine gird cells, so that quantities leaving from one gird cell should exactly balance the amount of entering quantities at the other cell borders. If at any point in computational domain level of refinement exceeds the required value, the highly refined gird may be removed or replaced with coarser one.

2.3 Computational Method/ Methodology

The numerical simulation of supersonic flow through convergent-divergent nozzle is performed on FLUENT code. FLUENT use Finite Volume Method (FVM) to solve Reynolds average Navier-Stokes (RANS) with turbulence models on unstructured mesh. In present study governing partial differential equations of fluid flow were linearized through implicit scheme because it converges more quickly than explicit scheme solver. Simulation is conducted on two-dimensional, axi-symmetric model in order to reduce computational time. Simulation of Steady state is Carried out by using density-based solver to take into account effects of compressibility, pressure based solver and density-based solver can be used for wide range of flows but density based solver is provides better accuracy for high speed compressible flows. There are number of turbulence models available from them k-ε & k-ω models are widely used in turbulence modeling. They both have good accuracy as well as don't need higher computational resources. In-spite of that each model have some limitation likewise k-ε model is more suitable for fully turbulent flows, initial iterative solution where as it doesn't provide suitable results in high pressure gradient, adverse pressure gradient and large separation regions flows (Schulz, 2011). In these types of flows k-ω model yield best results by allowing accurate near wall treatment as well as it gives better performance for adverse pressure gradient flows and also flows through complex geometries. Many researches reveal that k-ω model is more suitable than other one or two-equation RANS models for complex boundary layer flows and for flow through nozzle problems.

2.4 Discretization Scheme

In order to reduce errors and to get more accurate result of numerical simulation, setting of numerical schemes is key step likewise selection of turbulence model. Hence second order upwind Discretization scheme is used in this research work for finding the solution of equation s mentioned in **(Table 2).**

Table 2. Each Equation Discretize as

Pressure	**second order**
Momentum	**second order**
Turbulent Kinetic energy	**second order**
Turbulence dissipation rate	**second order**

Two upwind nodes considered to predict the value of eastern face. It assumes that gradient of considered node and eastern face is same as of western node and the considered node.

$$\frac{\phi_e-\phi_p}{\chi_e-\chi_p}=\frac{\phi_p-\phi_w}{\chi_p-\chi_w}\Rightarrow\phi_e=\frac{(\phi_p-\phi_w)(\chi_e-\chi_p)}{\chi_p-\chi_w}+\phi_p \quad (5)$$

Multi-dimensional linear reconstruction approach is applied to determine the quantities of cell faces for higher order accuracy. This technique uses Taylor expansion series for cell centered solution about cell-centroid to achieve higher order accuracy. Therefore, when second-order upwind discretization scheme is selected, the values of ϕ_f are calculated by using following expression.

$$\phi_f=\phi+\nabla\phi.\Delta\vec{s} \quad (6)$$

In equation (6), the cell-centered value is ϕ and gradient in the upstream cell is ∇ϕ, whereas Δs represent the displacement vector from upstream cell centroid to face centroid. In order to calculate value of the cell-centered the gradient in each cell should be known. Hence the gradient ∇ϕ is calculated from below equation by using divergence theorem,

$$\nabla\phi=\frac{1}{V}\sum_{f}^{N_{faces}}\tilde{\phi}_f\vec{A} \quad (7)$$

The value of ϕ_f is computed by taking average of ϕ from the two cells adjacent to that face. In equation (7) the maxima and minima points are introduced because the gradient $\nabla\phi$ is limited (FLUENT Inc., *FLUENT 6.3.2, User Guide*. 2006),

2.5 Full Multi-Gird initialization

Once the case is initialized, the numerical solution can be further improved through text user interface command of the FMG initialization. Basically, FMG is used to increase convergence rate of the simulation. It reduces computational time by two ways. In first method, it assumes that flow is in-viscid which in turn reduces number of flow governing equations hence reduces computational time. From the numerical simulations, it is found that by considering flow in-viscid computational time is reduced up to one half of the viscous flows. In the second case, FMG uses another strategy of multi-gridding. In this approach, adjacent cells are merged to form larger cells and then those new generated cells are combined to form a larger one yet. FMG then allows solver to find out solution on coarse grid, split to the intermediary refinement as well as re-converges and finally original mesh is restored and solution would be re-converging. In this way number of cells in computational domain is reduced to speed up solution. In case of three dimensional grids five tetrahedral cells are combined to form one tetrahedron therefore computational time is reduced by five times. The FMG initialization method is shown in Fig. 2 and it is also found that FMG initialized solution is far closer to final solution initialized by general initialization method.

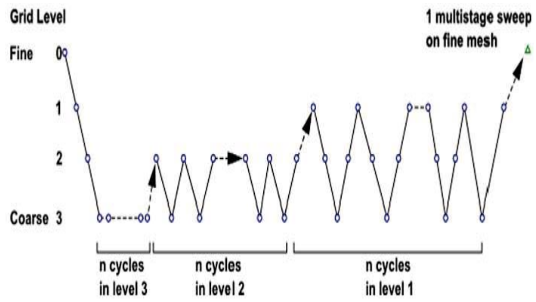

Fig. 2.The FMG initialization method

3. RESULT AND DISCUSSION

The CFD simulation results are presented to visualize the variation of various fluid parameters during flow through nozzle. In Fig. 3 velocity contour inside the CD nozzle is presented, from the velocity contour it is analyzed that fluid velocity increases in nozzle from throat to exit. Velocity at the inlet of nozzle is approximately zero while it achieved sonic condition near the throat and then fluid velocity reaches to 2210m/s at exit. In **(Fig. 4)** pressure contour is presented. Through visualizing pressure contour it is observed that very rapid change in pressure was observed in the vicinity of nozzle throat. **(Fig. 5)** presents turbulent viscosity ratio variation inside the supersonic CD nozzle. From turbulent viscosity ratio contour it was found that turbulent viscosity ratio experience significant change in convergent section whereas variation in turbulent viscosity ratio was also observed at the starting divergent portions. Variation in dynamic pressure was presented in **(Fig. 6)** and from dynamic pressure contour it was analyzed that dynamic pressure achieved its highest value near the throat and then starts decreasing with very low rate.

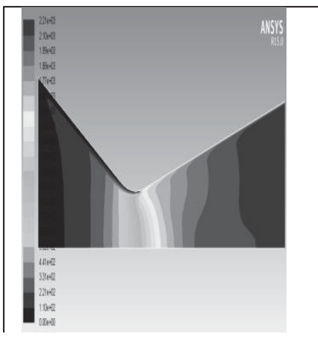

Fig.3. Show velocity variation in nozzle.

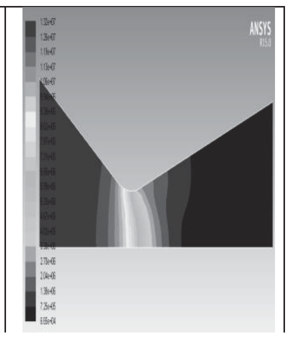

Fig. 4 Shows pressure variation in nozzle

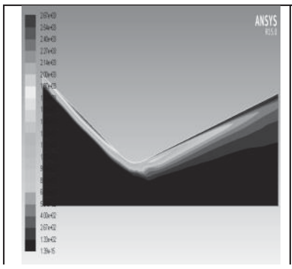

Fig. 5. Turbulent viscosity ratio contour inside nozzle.

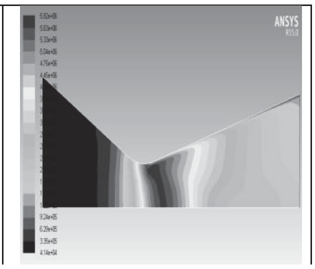

Fig. 6 Shows dynamic pressure variation inside nozzle.

The CFD simulation of nozzle also capable of providing numerical values of various fluid parameters like pressure, velocity, temperature and turbulences at various sections of nozzle. In this section, numerical calculations were performed on the basis of obtained results. For present study fluid velocity and pressure at the nozzle exit are required to calculate thrust force generated, those values were obtained from nozzle CFD simulation. Thrust produced as result of gases exhaust from the nozzle can be calculated through applying given equation.

$$T = \dot{m}V_e + \left(P_e - P_0\right)A_e \qquad (8)$$

In the above Eq. (8) T, $\dot{m}$, V_e, P_e, P_0 & A_e represents thrust generated, mass flow rate, exit velocity, exit pressure, free stream pressure and exit velocity respectively.

Table: 3 Shows the variation of nozzle design and their performance

Nozzle Convergent length (mm)	Convergent angle(degrees)	Thrust (KN)
602	30	2254.08759
615	29.5	2255.48579
628	29	2257.48634
640	28.5	2265.53869
645	28.3	2258.58668
650	28.2	2258.94225
655	28	2259.74716

Through applying Eq.(8) thrust force were calculated for various different nozzle model and results were plotted in **(Fig. 7)**. The results were plotted for thrust variation against convergent section length. From graph, it was analyzed that nozzle thrust performance is highest for convergent section length of 640mm and 28.5° convergent angle. From graph, it was observed that nozzle thrust performance increases from base design and at certain value of convergent section length it achieves highest value then again start decreasing.

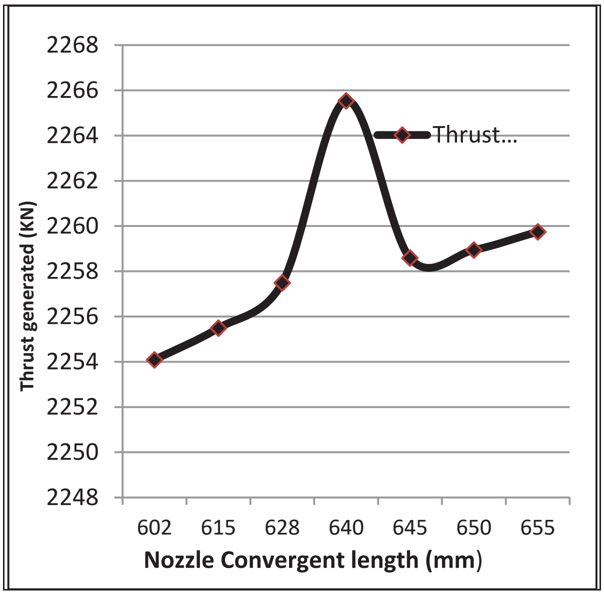

Fig. 7 shows the variation of thrust generated against nozzle convergent section length

4. CONCLUSION

In present study nozzle design was optimized through varying convergent section length and angle by keeping same expansion ratio and boundary conditions. The numerical simulation was made by using Fluent14.5 on unstructured gird. The computational work is carried out on 2D axi-symmetric density based, coupled solver with viscous $k-\omega$ SST turbulence model and linearized through implicit scheme. The research results revealed that nozzle thrust performance increases from base design and at certain value of convergent section length it achieves highest value then again start decreasing. Study reveals that 28.5° inlet angle would give best result and maximum thrust force would be produced at this angle. From simulation results it was analyzed that flow behavior inside the nozzle is highly complex nature.

REFERENCES:

Antipas, G., C. Lekakou, and P. Tsakiropoulos, (2011) Microstructural characterisation of Al-Hf and Al-Li-Hf spray deposits. Materials Characterization,. 62(4): 402-408.

Boyanapalli, R. (2013) Analysis of composite De-Laval nozzle suitable for rocket applications. International Journal of Innovative Technology and Exploring Engineering,. 2: 336-344.

Bayt, R. L. and K. S. Breuer. (1998) Viscous effects in supersonic MEMS-fabricated micronozzles. in Proceedings of the 3rd ASME Microfluids Symposium

Biju K. P. and M. Sajesh, (2013) Optimization of Divergent Angle of a Rocket Engine Nozzle Using Computational Fluid Dynamics. The International Journal Of Engineering And Science (Ijes),. 2(2): 196-207.

Castelli, M. R., A. Englaro, and E. Benini, (2011) The Darrieus wind turbine: Proposal for a new performance prediction model based on CFD. Energy,. 36(8): 4919-4934.

Dusa, D. (1989) Exhaust Nozzle System Design Considerations for Turboramjet Propulsion Systems. ISABE, 89-70-77.

Dai, Y. M., N. Gardiner, and W. H. Lam. (2010) CFD modelling strategy of a straight-bladed vertical axis marine current turbine. in The Twentieth International Offshore and Polar Engineering Conference.. International Society of Offshore and Polar Engineers

Ferreira, C. S., (2007.) Simulating dynamic stall in a 2D VAWT: modeling strategy, verification and validation with particle image velocimetry data. in Journal of physics: conference series. IOP Publishing.

Hegab, A. (2001) Nonsteady burning of periodic sandwich propellants with complete coupling between the solid and gas phases. Combustion and Flame,. 125(1):. 1055-1070.

Khattab, N. and M. Barakat, (2002) Modeling the design and performance characteristics of solar steam-jet cooling for comfort air conditioning. Solar Energy. 73(4):. 257-267.

Kumar, T. M. P. and D. Chatterjee, (2008) Numerical study of turbulent flow over an S-shaped hydrofoil. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science,. 222(9): 1717-1734.

Kim, J. H., H. D. Kim, and T. Setoguchi, (2010) The effect of diffuser angle on the discharge coefficient of a miniature critical nozzle. Journal of Thermal Science,. 19(3): 222-227.

Ketsdever, A. D. (2005) Experimental and numerical determination of micropropulsion device efficiencies at low Reynolds numbers. AIAA Journal. 43(3): 633-641.

Li, C., X. Peng, and C. Wang, (2010) Influence of diffuser angle on discharge coefficient of sonic nozzles for flow-rate measurements. Flow Measurement and Instrumentation,. 21(4):. 531-537.

Menter, F. R. (1994) Two-equation eddy-viscosity turbulence models for engineering applications. AIAA journal,. 32(8): 1598-1605.

Noh, M. H. M. (2011) Numerical Investigation of Chocked Converging-Diverging Nozzles For Thruster Applica Tion. Iium Engineering Journal, 12(3) 769-782.

Pearson, J., D. Landrum, and C. Hawk, (1996) Parametric study of solar thermal rocket nozzle performance. Journal of solar energy engineering,. 118(3): 194-195.

Schulz, V. (2011) Computational optimization of systems governed by partial differential equations. Vol. 8: 34-42. SIAM.

Wang, S. (2010) Numerical investigations on dynamic stall of low Reynolds number flow around oscillating airfoils. Computers & Fluids,. 39(9):1529-1541.