Scientia Iranica B (2019) 26(2), 834–842

Sharif University of Technology
Scientia Iranica
Transactions B: Mechanical Engineering
http://scientiairanica.sharif.edu

Designing an axisymmetric aerospike nozzle based on modified MOC

M. Masdari*, **M. Tahani, and F. Mohammadi Fard**

Faculty of New Sciences and Technologies, University of Tehran, Tehran, P.O. Box 1417614418, Iran.

Received 24 August 2017; received in revised form 17 November 2017; accepted 30 January 2018

KEYWORDS
Plug nozzle design;
Aerospike nozzle design;
Method of Characteristics;
Supersonic nozzle;
Plug design.

Abstract. This study provides a direct method based on the method of characteristics with assumptions of non-viscous flow and thermally perfect gas to design an axisymmetric plug nozzle. Inputs for numeric code include output Mach number, specific heat ratio, global gas constant, and a number of discrete steps of Prandtl-Meyer expansion fan. In the design process, no simplification was made and only unique spike geometry was created. Based on the criterion that locates the tip of the spike on the axis of symmetry, for each output Mach number, a correction factor was extracted which is the sole source of theoretical errors. The studied parameters include spike geometry, parallelism of exhaust nozzle flows and uniformity. Based on the numerical analysis, obtained results are consistent with desired output Mach number, and output flow is parallel to the axis of symmetry. A simple and direct formulation was used which performed very fast. Due to the error detected, which resulted in the surface slope reduction factor, some movement in spike geometry relative to ideal conditions was raised. Thus, the effect of a reduction factor may tend to zero by increasing the design Mach number.

1. Introduction

Bell-shaped nozzles are used widely in the spacecraft propulsion; however, a major problem about the nozzle is its limited range of their efficiency in changes in atmospheric pressure. At atmospheric pressure higher than the set point, which is generally close to the ground surface, the nozzle is positioned in an over-expanded condition and shock waves occur at the outlet of plug lip. In situations where atmospheric pressure is lower than the set point, the nozzle is positioned in under-expanded condition, and a generation of an expansion wave fan at the outlet nozzle lip is detected [1,2]. In each of these conditions, the amount of nozzle propulsion is less than the ideal state; then, the nozzle efficiency is reduced. The mentioned problems bring about the idea of aerospike nozzles where, due to their geometry, propulsion forces can be generated close to the design requirements in a wide range of ambient pressures, and the pressure through the calculation of thrust is removed theoretically by creating expansion/compression through interaction with the environment. Aerospike or plug nozzles generally have a plug and, in general, the flow supersonic expansion partially or completely occurs outside the outer shell of the nozzle and interacts with the atmosphere. Over time, geometrical types of the aerospike nozzles or plugs include axisymmetric, two-dimensional, and annular- and linear-shaped geometries. In this study, the purpose is to design an axisymmetric aerospike nozzle with a unique geometry for any Mach numbers by means of a direct method.

The concept of the plug nozzle was first conceived by the Germans prior to the Second World War for jet

*. *Corresponding author.*
E-mail addresses: m.masdari@ut.ac.ir (M. Masdari); m.tahani@ut.ac.ir (M. Tahani)

doi: 10.24200/sci.2018.5065.1072

airplane applications. Rolls-Royce, Ltd proposed the plug nozzle concept for rocket propulsion in U.S. in 1950 [3]. In 1961, Rao introduced the advancement of the plug nozzle equipped with rocket engines where, in the study, aerospike nozzles were raised as a new design of types of nozzles [4]. Between 1950s and 1970s, experiments were conducted on both annular- and linear-shaped geometries of the aerospike nozzle at General Electric and Rocketdyne that led to further development in plug nozzles. Later, in the 1990s, the experimental X33 SSTO program reinvigorated an interest in the linear aerospike nozzle. Over the past two decades, there has been a renewed interest in plug nozzles for flow physics knowledge and also in development and optimization of the nozzle that led to various experimental studies and understanding of the nozzle performance [5-11]. The computational studies mainly focus on validating the present-day solvers for predicting the plug nozzle flow fields [12-15]. Major researchers have used designing methods for the plug nozzle in the [16-18]. In 1974, Gearold et al. [19] studied the design of maximum thrust plug nozzles with variable inlet geometry. The results showed that length of the plug nozzles can be reduced to optimum level with the slight loss of axial thrust; the nozzles' geometry with constant length is so much dependent on base pressure, and this parameter should be considered to obtain a correct design and bring it closer to reality. Finally, the optimal inlet angle of the nozzle is a function of pressure and specific heat coefficient and is highly sensitive to these two parameters [19]. In a study conducted by Wuye et al. the aerospike nozzle performance and its contour optimization for solid propellant rocket motor were investigated. The study noted that thrust is reduced at the beginning of path according to experimental results with respect to solid rocket motors equipped to aerospike nozzle and, then, the mentioned thrust increased with an increase in pressure behind the nozzle. This phenomenon is common against a variety of bell-shaped nozzles; in addition, experimental results showed that the force of the aerospike nozzle during flight varies in a wide range. In this regard, this study has proceeded towards optimization and the fixing of special impulse [20]. Besnard et al. conducted a study named Design, Manufacturing and Test of a Plug Nozzle Rocket Engine in 2002. Corporations of industry and university of California state conducted the motor study. The design of the nozzle was performed by engineering estimation and, then, was investigated by CFD tool for flow quality. This research indicates the ability of design, manufacturing, and experimental test of the nozzle in an academic environment [21]. Besnard and Garvey in 2004 investigated the motor applicability equipped with aerospike nozzle in a small propellant and control for nano-satellite. This study highlights the priority of using an aerospike nozzle in the first step of a two-stage launcher of nano satellite. The reason for the priority in this research is explained as high amounts of available specific impulse through the aerospike nozzle [22]. Zebbiche and Youbi investigated a method based on the use of the Prandtl-Meyer function of perfect gas to design the contour of a plug nozzle of arbitrary shape and specified exit flow conditions. Using this method, the condition of designed nozzles in supersonic flow was compared with common bell-shaped nozzle whose results point to optimization of the plug nozzle in terms of thrust generation [23]. Other researchers, such as Kraiko and Tillyayeva investigate the optimal contouring of two-dimensional and axisymmetric aerospike nozzles providing maximum thrust in the given dimensions and external pressure [24]. Wang et al. conducted a study in the field of aerospike nozzle design and optimization. In their study, a simplified design and optimization method of aerospike nozzle contour and the results of tests and numerical simulation of aerospike nozzles are presented. The primary nozzle contour is approximated by two circular arcs and a parabola; the plug contour is approximated by a parabola and a third-order polynomial [25]. There are many researches that study the flow fields associated with truncated annular plug nozzles of varying lengths both experimentally and using computational tools. The ongoing investigation and identification of physics of flows is of great importance among the researches for several reasons: the importance of the issue, unveiling of some unknown phenomena, and the need for physical interpretations in the nozzle. Chutkey et al. investigated the analysis of annular plug nozzle flow field. Four nozzles with different lengths were studied experimentally along with computational tools. The applicability of RANS method and its limitations in prediction of flow in truncated spikes were studied [26].

The used method for designing the aerospike nozzles similar to kinds of bell-shaped nozzles is the method of characteristics. Rao used the common method for designing an axisymmetric aerospike nozzle is developed [16]. In this method, the maximum thrust is obtained by a nozzle contour according to fixed length of the nozzle and constant ambient pressure. The assumptions of the study are non-viscous and isentropic flow expansions. In this study, the variational integral is formulated along with control surface in output of nozzle, and the characteristic of flow is determined in the control surface and the nozzle contour is constructed by the method of characteristics to meet the desired flow. The major problem of this approach is the assumption of a constant length of the nozzle, including a constant characteristic slope contour and the complexity of the design process.

Another presented method makes the nozzle contour geometry using analytical approximate conditions.

This method was introduced by Angelino and uses an approximate method to design a plug nozzle [18]. In this method, the geometry was generated approximately, and parallelism of exhaust flow vectors were not guaranteed.

The present study was carried out due to the need for a design code for axisymmetric plug nozzle design with the least amount of simplifications and assumptions such as straight Mach lines and fixed length of nozzle.

In this study, using the method of characteristics, a direct method is presented to design an axisymmetric aerospike nozzle contour design geometry and external expansion of flow with given assumptions. In this method, all the expansion processes of the flow occur as Prandtl-Meyer expansion waves in external wall of the nozzle and in outside lip. The initial assumptions are non-viscous flow, thermally prefect gas, and isentropic expansion. A gradual numerical solution method of four-order Runge-Kutta was used to capture the characteristic lines; therefore, the slope of the characteristic lines and its progress in every step can continue until reaching the spike level point. In this method, considering the slope of the characteristic lines in every step and using a non-linear relationship as a derivate based on the numerical and discretional axisymmetric design, there is no errors due to considered fixed slope. Errors in the method include deviation from the thermally prefect gas, viscous flows, Prandtl-Meyer fan as well as the reduction factor. Compared to the method presented in [16], the presented method in this study has not considered length limitation for the nozzles and achieved the nozzle length with the desired output Mach number. In addition, compared to the method proposed in [18], lack of reflection characteristic lines of the surface and the output angle of the flow parallel to the axis of symmetry are considered.

2. Problem explanation and governing conditions

In the present study, initially, it is assumed that the entire process of expansion into the supersonic area occurs outside the nozzle body and the yield of design belongs to spike contour of plug nozzle such that the reflection of characteristic contours (Mach lines) does not occur. The design assumptions include thermally prefect gas and non-viscous fluid. In this condition and in every point over spike, slope of contour must be equal to flow angle; thus, the nozzle spike contour was achieved. The characteristic conditions were used to design the axisymmetric flow. As noted earlier, the purpose of axisymmetric aerospike nozzle design with external expansion of flow is to capture the nozzle spike geometry, which should be a unique solution ideally. In Figure 1, the aerospike nozzle geometry is in two-dimensional plane; Figure 2 shows axisymmetric geometry of the aerospike nozzle, including characteristic lines. Using [1], the characteristic and compatibility equations are equal to Eqs. (1) and (2) as follows:

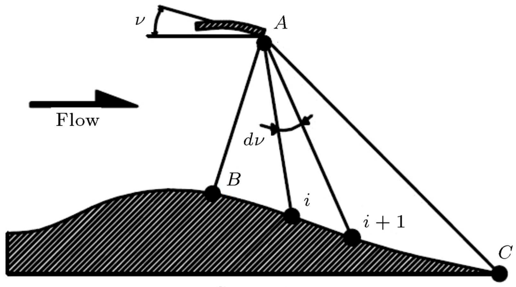

Figure 1. Aerospike nozzle geometry in flat two-dimensional flow.

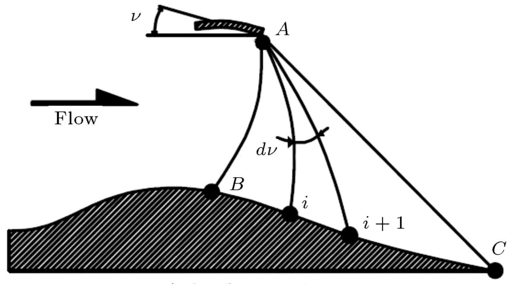

Figure 2. Aerospike nozzle geometry in axisymmetric flow.

$$\left(\frac{dr}{dx}\right)_{chr.} = \tan\left(\theta \mp \mu\right), \tag{1}$$

$$d\left(\theta \pm \vartheta\right) = \frac{dr}{r \times \sqrt{\mathrm{M}^2 - 1} \mp \cot\theta}. \tag{2}$$

In the above equations, the superscript sign is dedicated to both right-running and left-running characteristics. Compared to the two-dimensional plane flow, the compatibility equation is not linear and explicit; besides, the slope of characteristic lines is not constant; therefore, a numerical method is required to solve the mentioned equations; to this end, in the current study, the four-order Runge-Kutta method was used.

It is proven that the design of the subsonic portion of nozzle with a throat area is not very critical, and as observed, the same results in the nozzle outlet with different inlet geometries can be detected [3]. The nozzle design in the study was carried out with completely external expansion, meaning that the whole acceleration process of flow and its expansion around the lip of the nozzle is done as in Prandtl-Meyer fan cocentrality. By determining input values of the specific heat coefficient, nozzle throat length, and desired outside Mach number, the total amount of rotation of flow is obtained from Prandtl-Meyer function (Eq. (3)) [1]:

$$\vartheta(\mathrm{M}) = \int \frac{\sqrt{\mathrm{M}^2-1}}{1+\frac{\gamma-1}{2}\times \mathrm{M}^2}\frac{d\mathrm{M}}{\mathrm{M}} = \sqrt{\frac{\gamma+1}{\gamma-1}} \times \arctan\left(\sqrt{\frac{\gamma-1}{\gamma+1}\times(\mathrm{M}^2-1)}\right) - \arctan\left(\sqrt{\mathrm{M}^2-1}\right). \tag{3}$$

This value is equal to all flow rotation angles during expansion until reaching the desired Mach number at the nozzle outlet. By determining the number of characteristic lines, value changes of the discretized angles of expansion fan were obtained. The results of Eq. (3) determine the angle of flow to the nozzle throat, because, at the end of expansion, the same rotation is applied to flow; finally, flow exits in parallel with the axis of symmetry of the nozzle. Assumptions of solving the isentropic flow with the fixed specific heat coefficient and Prandtl-Meyer fan are centered on the lip of the nozzle outlet.

In Figures 1 and 2, AB line is the sonic characteristic line, Ai line is one of the central characteristic lines on the solution, $A(i+1)$ line is the next characteristic line, and AC is the terminal characteristic line. The AC in both geometries is a line with a constant slope. As mentioned, in the two-dimensional plane flow, the slope of characteristic lines is fixed, and that is why all characteristic lines on the geometry are straight lines. By measuring amount $\vartheta(\mathrm{M})$ to design Mach number from Eq. (3), the angle of the throat nozzle flow (the total required angle to rotation to achieve Mach number equal to 1 to the design Mach number) is obtained. First, the total required angle for rotation of flow $\vartheta(M)$ is discretized to small angles of $d\vartheta(M)$. For the two-dimensional plane geometry, the length of AB line can be obtained from the area of the nozzle throat, and spike geometry is obtained from the intersection of two lines $i(i+1)$ and $A(i+1)$. In axisymmetric geometry, the solution process becomes more complicated. In this geometry, calculating $\vartheta(M)$ and discretizing solution domain are similar to two-dimensional plane geometry; however, given the dependence of the slope on the distance from the axis of symmetry from Eq. (2), the characteristic lines must be discretized and calculated using the four-order Runge-Kutta and Eq. (2) must be solved in every discretized linear step. For axisymmetric geometry, the first step is the calculation of sonic line AB. In every step, the area of formed axisymmetric is calculated, and when the area reaches the calculated value of the nozzle throat, the solution stops, and the first characteristic line is formed as an indicator of sonic line of throat AB. In this step, the flow angle value is obtained at point B that is different and slightly bigger than axisymmetrical design and the amount of flow angle at point A. This is in contrast to the flat two-dimensional solution. From this point onwards, a continuous solving process leads to an unconventional geometry, because, in every step, the terminal angle on characteristic line located on spike is more relative to discretized central angle at point A, and the final point of spike is located under the bottom of symmetry line so that the resulted geometry cannot be proper. To overcome this problem, caused by physics of axisymmetric flow, a fixed reduction factor of resulted terminal slope is considered for each point i so that, at the end of the solution, terminal point C is placed on the axis of symmetry. Next, this reduction factor is determined by increasing the amount of flow Mach number, and because of a reduction of characteristic line slopes in each of terminal stations, the reduction factor becomes close to one and spike geometry will be very close to the ideal geometry. The impact of the coefficient is a reduction in the amount of required rotation of flow on the spike so that its impact will be dominant in creating the local compressive waves on the spike in practice.

3. Results

The design code based on the MOC method was explained in the previous section and was carried out for different supersonic Mach number plug nozzles. The values of a reduction factor based on output Mach number are shown in Table 1.

The results for exit Mach numbers 1.3, 1.7, and 2 were presented in ideal pressure ratio in each Mach number, including designed spike geometry and numerical results. The obtained geometry for the mentioned Mach numbers was achieved from nozzle design code and was compared with CFD Mach contour, as shown in Figures 3, 4, and 5 showing a consistent Mach distribution between CFD and MOC. Output radius of nozzle is considered equal to unit, and the number of characteristic lines in the designed cases is equal to

Table 1. The values of the reduction factor based on output Mach number.

Nozzle exit Mach number	Reduction factor
1.1	0.603
1.3	0.678
1.7	0.795
2	0.849
2.3	0.888
3	0.939
4	0.964
5	0.974

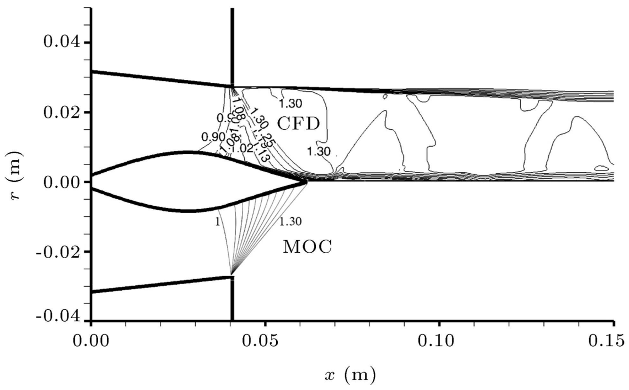

Figure 3. Nozzle geometry in Mach number 1.3.

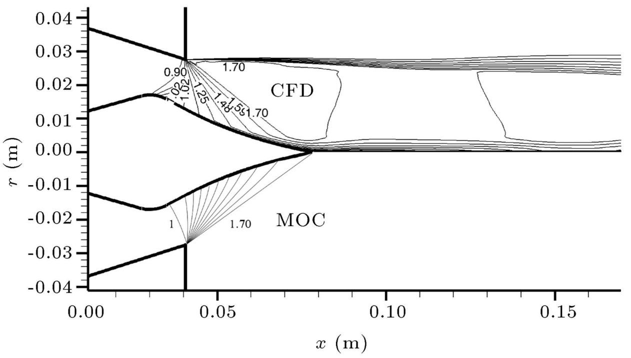

Figure 4. Nozzle geometry in Mach number 1.7.

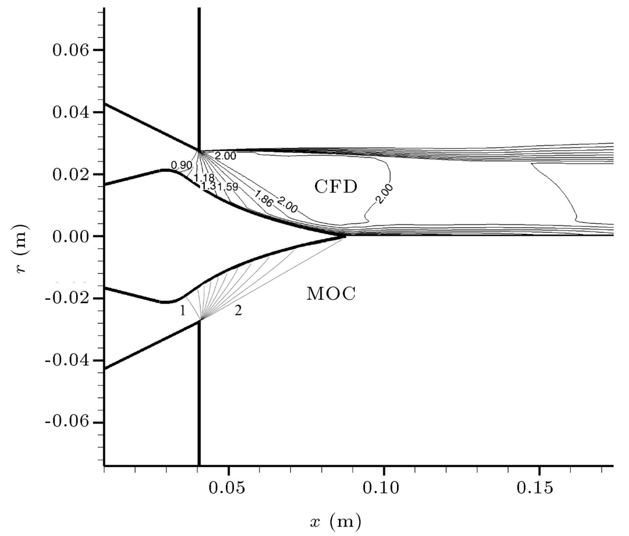

Figure 5. Nozzle geometry in Mach number 2.

200 lines where, due to the discretizing solution for every characteristic with discretized distance of about 0.001, these lines are created as curves at the throat of nozzle, and terminal characteristic line is created as a line with a constant slope because zero flow angle is zero at outlet in Eq. (2). As mentioned earlier, there is no limitation apart from assumptions of non-viscous flow and thermally prefect gas in the design process.

According to the design requirements, it is expected that the Mach number distribution along the axis of symmetry of nozzle and over spike would increase uniformly from a unit to setting Mach number; in the design pressure ratio, the Jet stream output exits uniformly and is parallel to the axis of symmetry of the nozzle. Another important measure is the continuity of the slope of spike so that if there is no continuity of curve slope, the risk of flow separation will increase.

In order to evaluate the functionality of the designed nozzles for each exit Mach number, the numerical simulation in the outer diameter of the outlet nozzle was taken to be 5.5 cm and in design pressure ratio for each Mach number (isentropic pressure ratio for output pressure of 1 atm.). Numerical simulation parameters are given in Table 2.

Using Figures 6, 7, and 8 and comparing the resulting Mach contours of nozzle both resulted in a design code and a numerical solution for the ideal pressure ratio of nozzle; in addition, good consistence was observed between Mach number distributions in

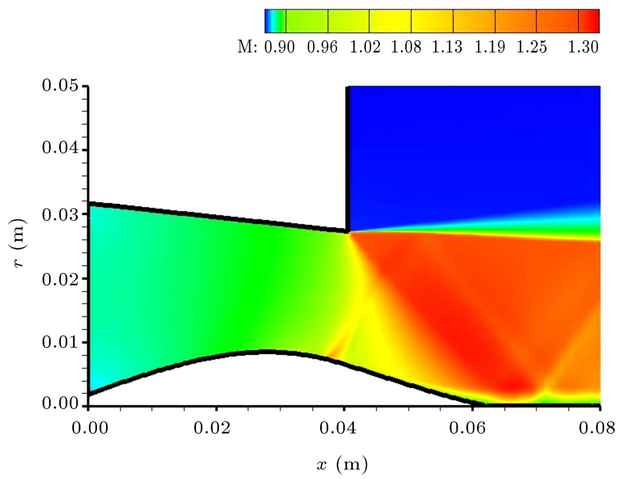

Figure 6. Distribution of Mach number resulted in numerical analysis in exit Mach 1.3.

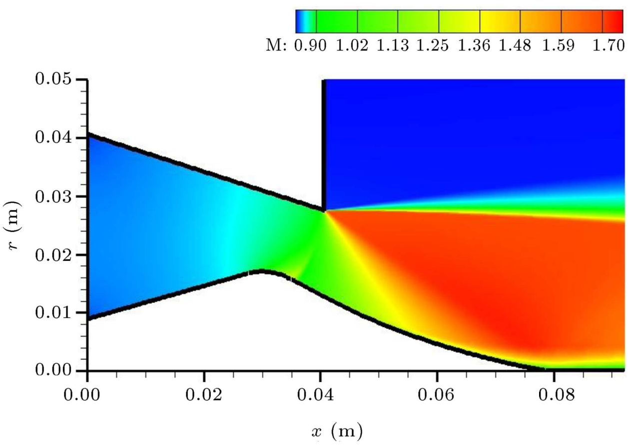

Figure 7. Distribution of Mach number resulted in numerical analysis in exit Mach 1.7.

Table 2. Numerical simulation parameters.

Physical parameter	Type/model
Number of cells in studied network	Course: 29000 medium: 252500 fine: 100500
Y^+ Criteria	$Y^+ < 1$
Turbulence	K-omega, SST
Solver	Density based
Analysis condition	Axisymmetric, steady
Density	Ideal gas
Specific heat coefficient	Piecewise polynomial
Viscosity	Sutherland 3 coefficient method
Solving formulation	Implicit
Type flux	AUSM
Solving order	Second order

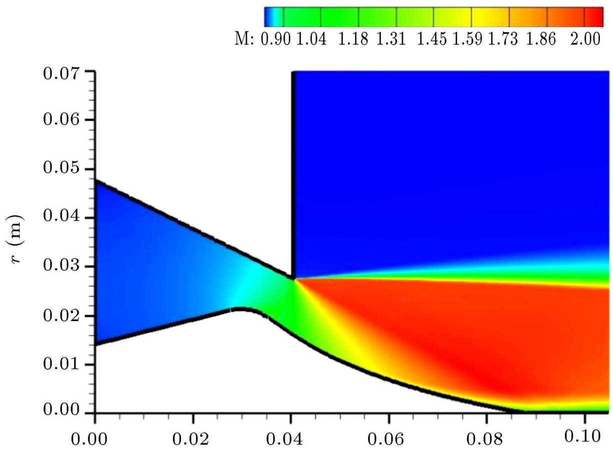

Figure 8. Distribution of Mach number resulted in numerical analysis in exit Mach 2.

the spike. The Mach number in the last characteristic line was consistent with designed Mach numbers for each nozzle.

Another important parameter in the aerospike nozzle is the parallelism of flow to the axis of the nozzle and the uniformity of the outlet angle of the nozzle into the ideal pressure ratio of the nozzle. If the flow is not parallel, the thrust vector has a lateral component and the efficiency of Nozzle reduces. Figure 9 shows the distribution of outlet angle at exit nozzle leading to the tip of the spike. The range of changes of angle on the negative side is of maximum 0.01 degrees and, at best, it is 0.003 degrees, which is quite perfect. These deviations are higher when the nozzle operates in higher or lower ambient pressure. The minimal amount of exit flow angle deviation at the nozzle outlet represents the validity of functional design of nozzle geometry at designed atmospheric pressure and enables the nozzle to generate maximum thrust by exhausting flow parallel to the axis of symmetry.

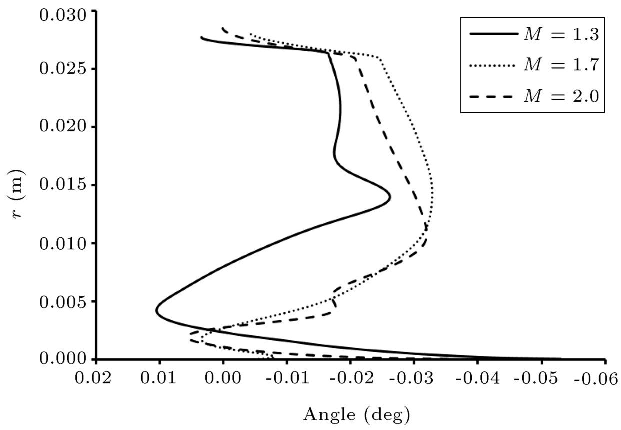

Figure 9. Distribution of outlet angle at exit nozzle leading to the tip of the spike for Mach numbers 1.3, 1.7 and 2.0.

Another important parameter in the design of the aerospike nozzle is continuity of spike curve; if it meets its criteria, the risk of flow separation may reduce, because, in the supersonic flow, lack of continuity may cause a sudden change of a slope of surface and make local compressive waves. Figure 10 shows the distribution of spike curve slope along the axis of symmetry. Using the figure, it is known that there are established continuity conditions for each point of spike (there are no sharp edge or breaks in the curve).

The last parameter evaluated in this study is the nozzle efficiency using a number of analytical data. The efficiency of a nozzle was achieved by Eq. (4) [2]:

$$\eta_{\text{Nozzle}} = \frac{\text{actual exit velocit } y^2}{\text{ideal exit velocit } y^2}. \tag{4}$$

For a designed nozzle contour, the amount of thrust of nozzle and the efficiency in design condition are shown in Table 3.

Finally, Figure 11 shows a comparison among the profiles of designed nozzles in the presented method

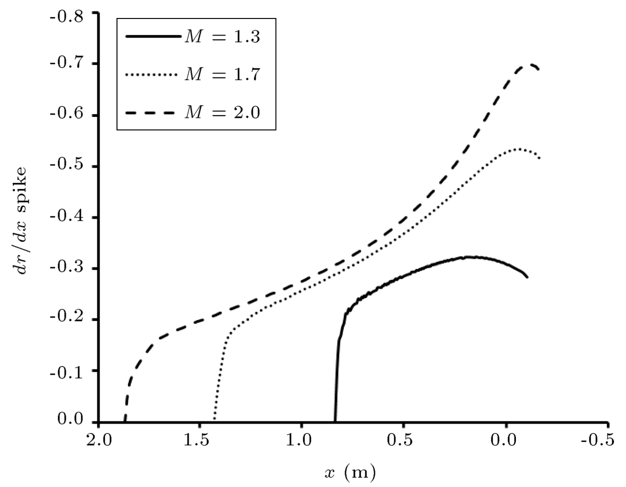

Figure 10. Distribution of slope of spike curve for Mach numbers 1.3, 1.7, and 2.0.

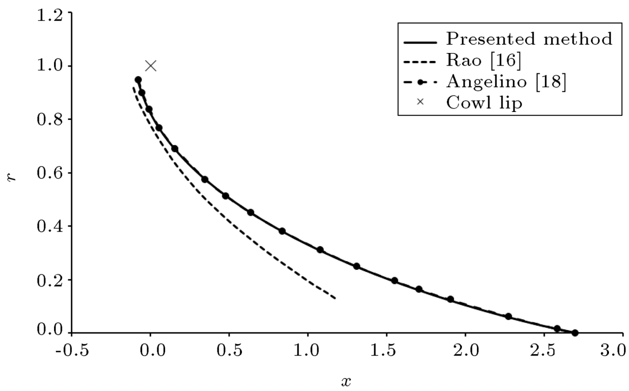

Figure 11. Comparison among spike contours in design Mach number 2.89 from [16,18] and suggested method $\gamma = 1/23$.

Table 3. Thrust and efficiency for designed nozzles.

Nozzle exit Mach number	Thrust (N)	Efficiency (%)
1.3	547.11	99.3
1.7	929.97	96.7
2.0	1287.21	96.7

and other resources associated with aerospike nozzle. According to this figure, the contour resulted in the suggested method has more compliance with analytical method presented in [18]; however, in the input region, a minor difference can be seen. This means that the proposed method in [18] predicts more input area. Due to the form of the method, the mentioned reference is used through estimating method by analytical equations and does not use the method of characteristics; besides, there is no analysis of these differences. The value of the difference in less Mach numbers was observed better (Figure 12). Compared with the method presented in [16], there are many differences. Firstly, due to differences in calculation methods, spike

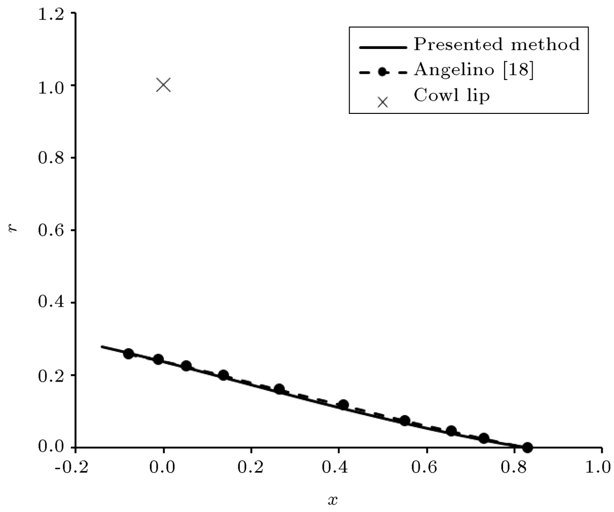

Figure 12. Comparison among spike contours in design Mach number 1.3 from [18] and suggested method $\gamma = 1/4$.

cuts off, and considering the slope of characteristic lines as constant makes the characteristic lines longer than true state; thus, leaner spike is obtained as compared to the other two methods; in addition, the inlet area of flow is also predicted to be more than true value.

The effect of viscosity virtually makes the spike thicker than the ideal condition and can be determined as displacement thickness in each station on spike. Two-phase flow, which occurs in solid propellant rockets, changes the specific heat ratio, and density of flow can be considered using CFD results in MOC output, in which both of mentioned deviations are out of scope of this study.

4. Conclusion

The current study introduced the design method of the aerospike nozzle while assuming a non-viscous flow, thermally prefect gas without any limitation and certain error sources; in particular, based on the Method Of Characteristics (MOC), three nozzles geometries for Mach numbers 1.3, 1.7, and 2.0 were designed through the numerical code formulation. The introduced method has the capability to create geometry in the desired exit Mach for each type of operating gas in very simple and low computational cost. In this method, to create a concave and flat geometry for spike and also to reach the tip of the spike to the symmetry line, a reduction factor for terminal angle was obtained on the surface of spike for each Mach number so that, by increasing the exit Mach number, due to the reduction of slope of each characteristic line and widening of expansion fan in the nozzle cowl, the reduction factor used for the terminal angle is close to number 1 and, thus, the resulting error can be removed. Due to the reduction of spike angle in non-ideal conditions, this factor creates poor local density waves on the low design Mach numbers. To validate the design code for nozzles, three 2-D axisymmetric numerical simulations were performed for the given Mach numbers 1.3, 1.7, and 2.0, where the constant output diameter of nozzle is equal to 55 mm and certain isentropic ratio of design for creating output pressure is equal to 1. The results included Mach number distribution, the output angle parallel to the axis of symmetry, and the created thrusts which all represent very good consistency with the results of analysis and numerical code developed for nozzle design. In addition, the investigation of the slope of the spike indicates the continuity of spike curve in order to prevent the flow separation risk. Two common methods for aerospike nozzle design in [16,18] were proposed. A comparison of the proposed method in the current study with those mentioned in references was conducted. In comparison with the [16], due to fixed slope of the characteristic lines, deviation from actual conditions, and fixed nozzle length in this method, the presented method in this study is more consistent with the real condition; moreover, there is no limitation in nozzle length. In addition, in the method introduced in [18], there is considerable overlapping in results; however, the prediction of the area of inlet in this reference is not appropriately accurate, and it is an area greater than the actual value.

Nomenclature

r	Distance from axis of symmetry
x	Axial distance
θ	Flow velocity angle from axis of symmetry
μ	Characteristic line angle from axis of symmetry
ν	Prandtl-Meyer function
M	Mach number
γ	Heat capacity ratio

References

1. Anderson, J.D., *Modern Compressible Flow*, 3rd Edn., pp. 397-407, McGraw-Hill, New York, USA (1990).
2. Faro, I.D.V., *Handbook of Supersonic Aerodynamics*, 1st Edn., Johns Hopkins university press, USA (1964).
3. Griffith, A. and Rolls-Royce., *Jet Propulsion Nozzle*, Patent, Docket No. US 2928235 A, USA (1954).
4. Rao, G.V.R. "Recent developments in rocket nozzle configurations", *ARS Journal*, **31**(11), pp. 1488-1494 (1961).
5. Tomita, T., Tamura, H., and Takahashi, M. "An experimental evaluation of plug nozzle flow fields", *32nd Propulsion Conference and Exhibit*, Lake Buena Vista (1996).
6. Tomita, T., Takahashi, M., Onodera, T., and Tamura, H. "Visualization of shock wave interaction on the surface of aerospike nozzles", *34th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit*, OH (1998).
7. Vuillamy, D., Duthoit, V., and Berry, W. "European investigation of clustered plug nozzles", *35th Joint Propulsion Conference and Exhibit*, Los Angeles (1999).
8. Sule, W.P. and Mueller, T.J. "Annular truncated plug nozzle flow field and base pressure characteristics", *Journal of Spacecraft and Rockets*, **10**(11), pp. 689-695 (1973).
9. Kapilavai, D., Tapee, J., Sullivan, J., Merkle, C.L., Wayman, T.R., and Conners, T.R. "Experimental testing and numerical simulations of shrouded plug nozzle flow fields", *45th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*, Denver (2009).
10. Junwei, L., Yu, L., Unfei, L., Changhui, Yibai, W., and Ningfei, W. "Experimental and numerical study on two dimensional plug nozzle", *46th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*, Nashville (2010).
11. Tomita, T., Takahashi, M., and Tamura, H. "Flowfield of clustered plug nozzles", *33rd Joint Propulsion Conference and Exhibit*, WA (1997).
12. Chutkey, K., Vasudevan, B., and Balakrishnan, N. "Flow field analysis of linear plug nozzle", *Journal of Spacecrafts and Rockets*, **49**(6), pp. 1109-1119 (2012).
13. Ito, T., Fujii, K., and Hayashi, A.K. "Computations of the axisymmetric plug nozzle flow fields: Flow structures and thrust performance", *Journal of Propulsion and Power*, **18**(2), pp. 254-260 (2002).
14. Rommel, T., Hagemann, G., Schley, C.A., Krülle, G., and Manski, D. "Plug nozzle flowfield analysis", *Journal of Propulsion and Power*, **13**(5), pp. 629-634 (1997).
15. Ruf, J.H. and McConnaughey, P. "A numerical analysis of a three dimensional aerospike", *33rd Joint Propulsion Conference and Exhibit*, Huntsville (1997).
16. Rao, G.V.R. "Spike nozzle contour for optimum thrust", *Journal of Planetary and Space Science*, **4**, pp. 92-101 (1961).
17. Greer, H. "Rapid method for plug nozzle design", *ARS Journal*, **31**(4), pp. 560-561 (1960).
18. Angelino, G. "Approximate method for plug nozzle design", *AIAA Journal*, **2**(10), pp. 1834-1835 (1964).
19. Gearold, R.J., Doyle Thompson, H., and Hoffman, J.D. "Design of maximum thrust plug nozzles with variable inlet geometry", *Journal of Computers & Fluids*, **2**(2), pp. 173-190 (1947).
20. Wuye, D., Yu, L., Xianchen, C., and Haibin, T. "Aerospike nozzle performance study and its contour optimization", *37th Joint Propulsion Conference and Exhibit*, Reston (2001).

21. Besnard, E., Chen, H.H., Mueller, T., and Garvey, J. "Design, manufacturing and test of a plug nozzle rocket engine", *38th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*, Indianapolis (2002).

22. Besnard, E. and Garvey, J. "Aerospike engines for nanosat and small launch vehicles (NLV/SLV)", *Space Conference and Exhibit*, Ca. (2004).

23. Zebbiche, T. and Youbi, Z. "Supersonic plug nozzle design and comparison to the minimum length nozzle configuration", *IJASS Journal*, **7**(1), pp. 27-42 (2006).

24. Kraiko, A.N. and Tillyayeva, N.I. "Contouring spike nozzles and determining the optimal direction of their primary flows", *Journal of Fluid Dynamics*, **42**(2), pp. 321-329 (2007).

25. Wang, C., Liu, Y., and Qin, L.Z. "Aerospike nozzle contour design and its performance validation", *Journal of Acta Astronautica*, **64**(11,12), pp. 1264-1275 (2009).

26. Chutkey, K., Vasudevan, B., and Balakrishnan, N. "Analysis of annular plug nozzle flowfield", *Journal of Spacecraft and Rockets*, **51**(2), pp. 478-490 (2014).

Biographies

Mehran Masdari is an Assistant Professor at University of Technology. He holds BSc, Eng, MSc, and PhD degrees in Aerodynamics from Sharif University of Technology. He published more than 50 scientific papers and 30 text books in English and Persian. He has 15 years of job experience both in industry and academic fields. Various aerospace courses and student projects were conducted by him. His research interests include transitional flows, applied aerodynamics, experimental fluid dynamics, Particle Image Velocimetry (PIV), bluff body wakes, turbulent boundary layer, wind engineering, mechanical engineering, fluid mechanics, aerodynamics, turbomachinery, turbulence, wind turbine, Vertical Axis Wind Turbine (VAWT), micro air vehicles, fluid mechanics, aerospace engineering, fluid dynamics, wind tunnel testing, neural network, and data processing.

Mojtaba Tahani was born in 1984 in Semnan. He received BS, MS, and PhD degrees in Mechanical Engineering (fluid mechanics) from Semnan University, Iran University of Science and Technology, Iran in 2006, 2008, and 2012, respectively. He has been a member of Iran's National Talents Foundation since 2009 and he has been an Assistant Professor at the Aerospace Department of Faculty of New Sciences and Technologies of University of Tehran since 2012. He is an editorial board member of "International Journal of Energy Science and Engineering" and "American Journal of Renewable and Sustainable Energy" journals. Dr. Tahani was invited to the renewable energy researcher program in 2016 to Mie University, Japan. He has executed several energy projects among which the power production from water distribution network can be named. He has recorded several inventions and also has published numerous research papers in the field of numerical simulations and also optimization of different engineering problems. His research interests are energy, turbulence, physics and astronomy, computational fluid dynamics and optimization.

Fata Mohammadi Fard is a PhD student of Aerospace Engineering in University of Tehran. He is holding BSc Degree in Fluid Mechanical Engineering from Yazd university and MSc in Space Engineering from Malek-e-Ashtar University of Technology. He is interested in high-speed flow regimes and working on experimental and numerical calculations of supersonic flows.