Numerical Investigation of Propellant Flow and Finite Element Analysis of Wall Structure for a Bi-propellant Thruster, Compared to Proposed Analytical Results

Maziar Shafaee and Abbas Elkaie
Faculty of New Science and Technology, University of Tehran, North Kargar Street 143951374, Tehran, Iran
Email: mshafaee@ut.ac.ir, a.elkaie@ut.ac.ir

Hamed Fallah
Department of Mechanical Engineering, Iran University of Science and Technology, narmak 1684613114, Tehran, Iran
Email: fallah.hamed8@gmail.com

Jebraiel Bayramlu
Department of Mechanical Engineering, Bonab Islamic Azad University, Bonab 5551785176, Bonab, Iran
Email: J.bayramlu@gmail.com

***Abstract*—This paper reviews the design process of a bi-propellant space propulsion thruster to output 80N nominal thrust. The thrust chamber design problem of a space propulsion system is a complex and time consuming process. This is mainly due to performance constraints and high temperature flow requirements. These facts along with the specific geometry of the thrust chamber, make the fluid computations and structure analysis so difficult, particularly in terms of the thermal and force stresses. CFD and FEA are useful methods that helped to overcome these difficulties. Therefore, the thruster was successfully designed using ideal rocket equations and the design was successfully confirmed using CFD and FEA.**

***Index Terms*—space propulsion system, bi-propellant, thrust chamber, CFD, FEA**

I. Introduction

Liquid propulsion systems are the most popular form of space propulsion when relatively high specific impulse and thrust are required [1]. As telecommunication satellites become larger and longer-lived as a result of the launch capability provided by more efficient launch vehicles, integrated liquid bi-propellant propulsion systems will replace solid propellant motor/liquid monopropellant thruster propulsion systems for apogee and on-orbit maneuvers [2].

Design process of liquid propulsion system is still a challenging and labor-intensive process. The performance of a liquid propulsion system depends greatly on different variables such as the chamber pressure and oxidizer-to-fuel mass ratio. In the preliminary design of a liquid propulsion system, these last design parameters are determined by system analysis that considers the design requirements and constraints [3]. The qualitative or subjective decision-making for the conceptual phase of design of space propulsion system are used traditionally [4].

Designing the thrust chamber as the main part of bi-propellant engines requires to identify the specific engine operation parameters. The main objective is to produce the desired thrust and increase the specific impulse by improving energetic parameters of engine. Selecting more energetic propellant combinations, increasing working pressure and choosing an optimal oxidizer-to-fuel ratio are common methods to increase the specific impulse; all of them require new strategies to increase chamber tolerance to the increased heat created during combustion and this will need to the practical solutions to predict and calculate exact critical values (like maximum temperature) in the thrust chamber.

The present computational research developed and validated a propulsion system design strategy for liquid propulsion systems to satisfy the required thrust under performance and structural constraints. The results obtained show that the proposed method provides an effective way to obtain fluid variables and structure deflections and stresses using both Computational Fluid Dynamic (CFD) and Finite Element Analysis (FEA) methods along with analytical computations in order to design thrust chamber for bi-propellant thrusters. Thus, the paper reviews the design process of a bi-propellant thruster. This thruster is designed to use MMH&N2O4 propellants while providing a minimum thrust of 80N [5].

II. Design Process

Cylindrical combustion chamber with a flat injector is used in Liquid fuel engines as the first choice. The main advantage of such configuration is the proper use of

Manuscript received July 15, 2015; revised December 17, 2015.

doi: 10.18178/ijmerr.5.2.133-137

chamber's total volume and capacity [5]. Cylindrical combustion chambers satisfactorily provide suitable combustion, performance, stability of operation and reliable internal cooling. Geometry of a thrust chamber consist of a combustion chamber and a divergent-convergent nozzle that is shown in Fig. 1. In order to design and produce such combustion chamber, the combustion chamber volume, length of cylindrical part, chamber's diameter and the nozzle inlet length must be determined.

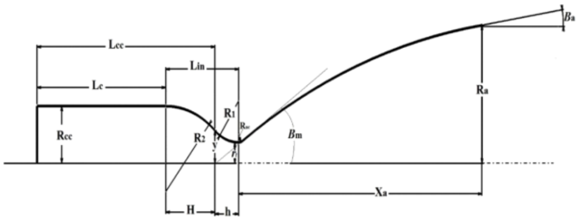

Figure 1. Geometry of a sample thrust chamber [5].

A. *Geometrical Calculations and Combustion Chamber Profile.*

A typical combustion chamber profile, is shown in Fig. 1. After calculating the circular area at each section (nozzle exit, throat and combustion chamber), by using dynamic gas calculations (24-31), other dimensions can be calculated by using (1-18) [5].

$$R_1 = d_t \tag{1}$$

$$R_2 = \rho R_c \tag{2}$$

where d_t is the throat diameter, R_{cc} is the combusion chamber radius and R_1 and R_2 are shown in Fig. 1. ρ is a constants value that will be change with respect to combustion chamber pressure. Combustion chamber volume can be calculated by using the method of equivalent combustion length L^* (3-12) [5]:

$$V_{cc} = L^*.A_t \tag{3}$$

$$L_{cc} = \frac{V_{cc}}{A_c} \tag{4}$$

$$\overline{A_{cc}} = \frac{V_{cc}}{A_c} \tag{5}$$

$$L_{in} = 0.5 d_t \sqrt{(2+\rho\sqrt{\overline{A_{cc}}})^2 - [(\rho-1)\sqrt{\overline{A_{cc}}}+3]^2} \tag{6}$$

$$h = \frac{2L_{in}}{2+\rho\sqrt{\overline{A_{cc}}}} \tag{7}$$

$$H = L_{in} - h \tag{8}$$

$$y = d_t[\frac{h}{L_{in}}\sqrt{\overline{A_{cc}}} + \frac{H}{L_{in}}] \tag{9}$$

$$\bar{y} = \frac{y}{d_t} \tag{10}$$

$$\Delta V_{in} = A_t L_{in}\{[(2\overline{A_{cc}} + \bar{y}^2)\frac{H}{3L_{in}}] + [(\bar{y}^2 + \bar{y} + 4)\frac{h}{6L_{in}}]\} \tag{11}$$

$$L_c = \frac{V_{cc} - \Delta V_{in}}{A_c} \tag{12}$$

Recent variables are shown in Fig. 1.

B. *Nozzle*

The nozzle will increase the velocity and decrease the pressure of combustion products. Cone-shape, Delaval-shape (Trimmed bell) and Bell-shape are common nozzle configurations. Most of space thruster nozzles are bell-shaped convergent-divergent nozzles [1] and the main variables are calculated according to (13-16) [5].

$$r_1 = (0.1-0.2)d_t \tag{13}$$

$$r_{in} = r_t + r_1(1-\cos\beta_m) \tag{14}$$

$$\lambda = 0.5(1+\cos\beta_a) \tag{15}$$

$$\beta_a = (12^o - 18^o) \tag{16}$$

where r_1, r_2, r_t and β_m are shown in Fig. 2. λ is the efficiency of the nozzle and β_a is the cone half angle. The length of divergence part in a conical nozzle is calculated according to (17) [5]:

$$L_n = \frac{d_e - d_t}{2\tan\beta_a} \tag{17}$$

where L_n is the length of nozzle, d_e is the diameter of nozzle and d_t is the throat diameter. The cone half-angle in conical nozzles is about 12^o-18^o.

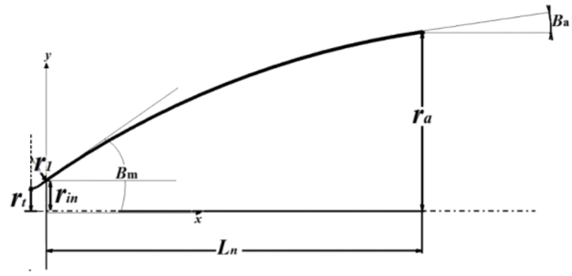

Figure 2. Bell nozzle configuration [5].

Conical nozzle with a 15^o half angle, due to suitable compromise between weight, length and operation, is known as a standard nozzle. Bell nozzles of the same length have better performance compare to cone nozzles with a 15^o cone angle [1]. Bell nozzles analysis require complex calculations using computational fluid dynamic method based on the methods of characteristics [6]. However, in order to design the nozzle profile for bell nozzles, parabolic approximation can be used with a good precision [7]. This approach offers good answers, therefore the characteristics method used rarely to design the nozzle. The bell nozzles provide more efficiency compared to conical ones [8]. Using a parabolic approximation, for the bell profile of the nozzle, provide near optimum design to produce maximum thrust, which is proposed by Rao [9]. The configuration that used the parabolic approximation, is shown in Fig. 1. Equations (18-23) can be used to design the bell nozzle parabolic profile.

$$y = ax^2 + bx + c$$
$$y|_{x=0} = r_{in} \quad \rightarrow c = r_{in} \tag{18}$$

$$\left.\frac{dy}{dx}\right|_{x=0} = \tan\beta_m \quad \rightarrow b = \tan\beta_m \tag{19}$$

$$y\big|_{x=L_n} = r_a = r_{in} + \tan\beta_m . L_n + aL_n^{\,2} \tag{20}$$

$$\left.\frac{dy}{dx}\right|_{x=L_n} = \tan\beta_a = \tan\beta_m + 2aL_n \tag{21}$$

The constant value of a and the nozzle length L_n, can be calculated by using (20-21). Finally, the nozzle length (L_n) and nozzle contour will be calculated according to (22-23).

$$L_n = 2\frac{r_a - r_{in}}{\tan\beta_a + \tan\beta_m} \tag{22}$$

$$r = (\frac{1}{4}\frac{\tan^2\beta_a - \tan^2\beta_m}{r_a - r_{in}})x^2 + (\tan\beta_m)x + r_{in} \tag{23}$$

III. Gas Dynamic Calculations and Thruster Sizing

Thruster design and sizing consisted of several considerations; ambient pressure, chamber pressure, desired thrust and propellants combination are some of them. Exit velocity and exit pressure, determine the amount of thrust that the thrust chamber would produce. Chamber pressure affected the combustion characteristics and the required strengths of the thruster (wall thickness) and components such as valves and piping. Sizing started with the ideal rocket equations (24-31) [1]:

$$\dot{m} = \frac{T}{I_{sp}} \tag{24}$$

$$T_t = \frac{T_c}{(1+\frac{k-1}{2})} \tag{25}$$

$$P_t = P_c(1+\frac{k-1}{2})^{\frac{-k}{k-1}} \tag{26}$$

$$M_e = \sqrt{\frac{2}{k-1}[(\frac{Pe}{Pt})^{1-k/k} - 1]} \tag{27}$$

$$T_e = T_t(1+\frac{k-1}{2}M_e^{\,2})^{-1} \tag{28}$$

$$\frac{A_y}{A_x} = \frac{M_y}{M_x}\sqrt{[\frac{1+[k-1/2]M_y^{\,2}}{1+[k-1/2]M_x^{\,2}}]^{\frac{k+1}{k-1}}} \rightarrow$$

$$A_e = A_t(\frac{k+1}{2})^{\frac{-(k+1)}{2(k-1)}}\frac{(1+\frac{k-1}{2}M_e^{\,2})^{\frac{k+1}{2(k-1)}}}{M_e} \tag{29}$$

$$V_e = \sqrt{\frac{2k}{k-1}\frac{R'T_c}{\mathrm{M}}[1-(\frac{p_t}{p_c})^{\frac{k-1}{k}}]} \quad or \quad V_e = M_e\sqrt{kRT_e} \tag{30}$$

$$T = \dot{m}V_e + (P_e - P_a)A_e \tag{31}$$

where m^o is the total mass flow rate, T is the engine thrust, I_{sp} is the specific impulse and P_i, T_i, M_i, A_i and V_i are pressure, temperature, mach number, area and velocity in each section, respectively.

IV. Results and Discussion

Most calculations to formulate the design problem of space propulsion system related to the design of the thrust chamber. This includes gas dynamic equations to calculate flow parameters and geometric equations to calculate the geometric parameters of the thrust chamber. The equations used in this section were extracted from different sources and some are based on experimental relationships. Thus, validation of these equations is necessary. The results of a real space propulsion system were considered as an evaluation index. The real propulsion system chosen was produced by Kaiser Marquardt (KM) [1]. Mission requirements and type of propellant were selected as defined in the KM catalog to integrate the initial conditions of the design.

The KM propulsion system considered an 80N thrust attitude control thruster. Since the outside ambient is considered to be vacuum, the outlet pressure is assumed to be zero. The platform utilizes a conventional MMH/NTO bi-propellant propulsion system to provide the necessary delta-V for transfer orbit maneuvers or station keeping.

TABLE I. Validating the Governing Equations Using KM Model

Design variable	KM design data [1]	Calculated results	Error (%)
I_{sp} *(s)*	290	288.3681	0.56
m^o *(gr/s)*	28.12	28.3	0.64
d_c *(mm)*	27.686	30.6	10.5
d_t *(mm)*	10.8458	10.9	0.49
d_e *(mm)*	76.6572	77.1	0.57
T (N)	80.0947	82.8413	3.4

The governing equations were validated against the KM real case design data. The results are shown in Table I. A comparison of results shows that the governing equations were able to calculate performance and geometric parameters adequately.

A. CFD-FEA Results

The FEA analysis indicates that the maximum deflection of a 2.8mm thick wall, would be 1.06×10^{-4}mm. This deflection would occur towards the exit nozzle area and less deflection would occur where the thruster is fix supported in the beginning (Fig. 3). To consider whether this deflection is acceptable, the stress values on the combustion chamber were examined.

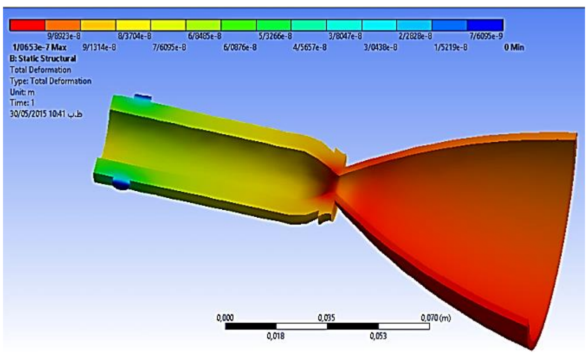

Figure 3. FEA numerical calculation of the deflection the thruster will experience shows no dangerous deflections or strain concentration points.

The next FEA model that was produced during the analysis was the stress model. Fig. 4 shows maximum stress locations and various stress concentration throughout the model. One noticeable feature is that the location of the maximum stress of 1.6×10^6 pa occurs in the combustion chamber and throat, which is well below the yield stress of niobium, even when greatly heated.

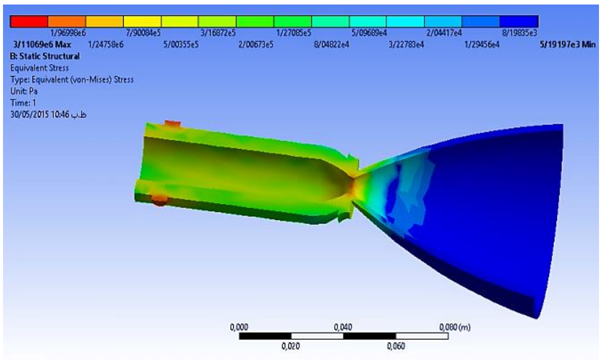

Figure 4. FEA numerical calculation of the stresses the thruster will experience show a large margin of safety for 80N thruster.

CFD was used to check the calculated velocity of the flow at the thruster outlet. A numerical analysis was performed on the proposed bi-propellant thruster. The introduced gas to the model, had an initial pressure and temperature of 12bar and 2600°K respectively. The outlet of the thruster had a final temperature of 1100°K and a final pressure of 0bar.

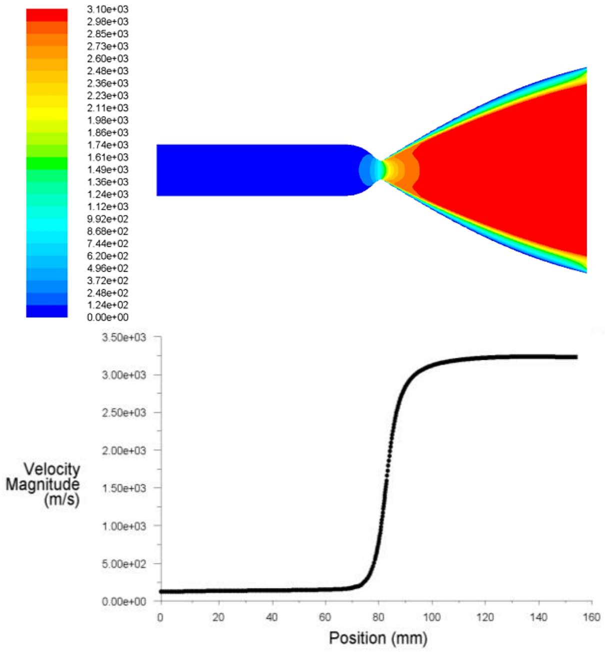

Figure 5. Velocity profile of the thruster using CFD results that show similarities to analytical results and indicate no unusual flow phenomena.

Fig. 5 shows the velocity contour inside the thruster. The result of the CFD model shows that the exit velocity reached a value of 3100m/s with an average velocity of 350m/s inside the combustion chamber. However, it is important to know that the CFD code used in this study does not take into consideration the combustion that is occurring inside the chamber that would cause both temperature and velocity rise as a result of the chemical reaction. Knowing the exact pressure and velocity would require extensive research and program to determine the effects of the combustion on the gas particles inside the thruster. However, for this experiment the CFD model was used to give a rough estimate to the exit velocity. Therefore, a more precise model was not crucial for the experiment. From the CFD model, the exit velocity of 3100 m/s is very similar to the calculated velocity that is discussed in greater depth in the analytical based calculations.

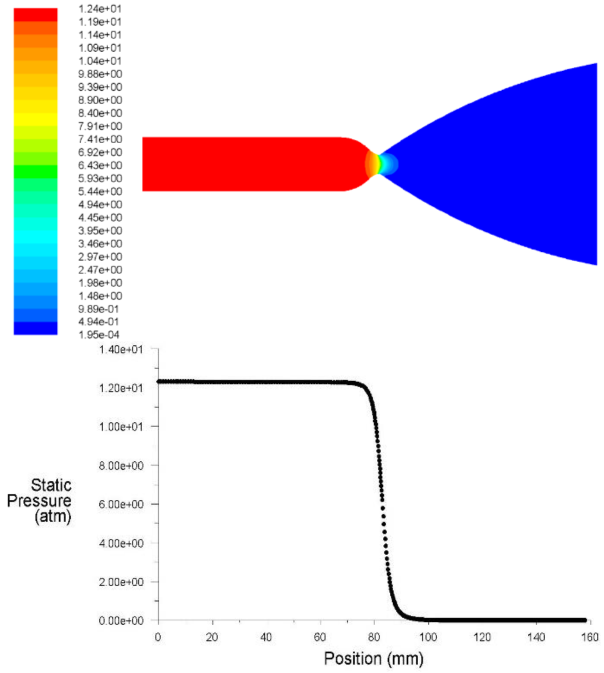

Figure 6. Pressure profile of the thruster using CFD results that show appropriate pressure decrease through the nozzle.

The CFD analysis also modeled the pressure decrease inside the thruster as the flow exited the nozzle. Fig. 6 illustrates the static pressure contour. The result of the CFD model shows that the exit static pressure reached ambient pressure with an average static pressure inside the combustion chamber having 12.36bar.

TABLE II. TEMPERATURE VALUES BY USING CFD CALCULATIONS COMPARED TO ANALYTICAL RESULTS

	Temperature (°K)		
Section	Analytical results	CFD results	Error (%)
Inlet	2696	2600	3.56
Throat	2532	2408	4.89
Exit	1118	1053	5.8

Table II- Table IV show the deviation values between analytical results and the results that are obtained from CFD simulations. The error values indicates that the governing analytical equations could calculate and predict the flow variables properly in comparison with CFD simulations. Moreover, the results that are provided in Table I- Table IV, indicates that the governing analytical equations are capable to calculate the geometrical design variables, flow parameters and also performance quantities, properly compared with KM real bi-propellant thruster.

TABLE III. VELOCITY VALUES BY USING CFD CALCULATIONS COMPARED TO ANALYTICAL RESULTS

Velocity (m/s)			
Section	Analytical results	CFD results	Error (%)
Inlet	0	0	0
Throat	1069.41	1106	3.4
Exit	3067.5	3100	1.06

TABLE IV. PRESSURE VALUES BY USING CFD CALCULATIONS COMPARED TO ANALYTICAL RESULTS

Pressure (bar)			
Section	Analytical results [5]	CFD results	Error (%)
Inlet	12.36	12.13	1.86
Throat	7.15	6.96	2.65
Exit	0.0059	0.0057	3.39

V. CONCLUSION

Comparing the CFD results to analytical ones concluded that the results of computer models reasonably agreed with analytical calculations; having a maximum 6 % difference between the numerical and analytical values. The velocities of the inlet and outlet of the nozzle were analytically calculated by knowing the geometry, temperature, and pressure that would enter and exit the nozzle. Thus, in order to design the nozzle profile for bell nozzles, parabolic approximation can be used with a good precision. This approach offers good answers, therefore the characteristics method used rarely to design the nozzle.

REFERENCES

[1]. G. P. Sutton and O. Biblarz, *Rocket Propulsion Elements*, 7st ed. John Wiley & Sons, 2001.

[2]. S. D. Rosenberg, "Liquid bi-propellant engines for large telecommunication sattelites," *Acta Astronautica*, vol. 9, pp. 663-669, 1982.

[3]. L. Sangbok, R. Tae-Seong, K. Ja-Ye and K. Kuisoon, "Design optimization of liquid rocket engine using a genetic algorithm," in *Proc. 63rd International Astronautical Congress*, 2012.

[4]. A. Butt, C. G. Popp, H. M. Pitts, and D. J. Sharp, "NASA ares i launch vehicle roll and reaction control systems design status," *AIAA*, 2008.

[5]. A. Elkaie, "Optimal design of a space control system components with objective function to minimize the mass," M.S. thesis, Faculty of New Scienc and Technology, University of Tehran, 2014.

[6]. H. M. Frey and G. R. Nickerson, "TDK-two dimensional kinetic reference program," Developed for NASA Contract NAS9-10391. Irvine, CA: Dynamic Science, 1971.

[7]. R. W. Humble, G. N. Henry, and W. J. Larson, *Space Propulsion Analysis and Design*, McGraw Hill Inc. 1995.

[8]. D. K. Huzel and D. H. Huang, *Modern Engineering for Design of Liquid Propellant Rocket Engines*, AIAA, 1992.

[9]. G. V. R. Rao, "Exhaust nozzle contour for optimum thrust," *Journal of Jet Propulsion*, vol. 28, pp. 377-382, 1958.

Maziar Shafaee received B.S. degree in Mechanical Engineering from Tabriz University in 2000. He also received M.S. and PhD degrees from University of Tehran in 2002 and 2011 respectively. Mr. Shafaee is currently work as an assistant professor, in Faculty of New Sciences and Technologies at University of Tehran. His research interests include numerical and experimental study of spray and atomization systems.

Abbas Elkaie received B.S. degree in Mechanical Engineering from Babol Noshirvani University of Technology in 2012. He received M.S. degrees in Aerospace Engineering from University of Tehran in 2014. He is a Ph.D. candidate in University of Tehran from 2014. Mr. Elkaie is currently working on his Ph.D. thesis on the engineering systems optimization at Faculty of New Sciences and Technologies in University of Tehran. His research interests include design optimization of engineering systems using MDO, evolutionary, gradient-based, hybrid and also multi-objective optimization techniques.

Hamed Fallah received B.S. degree in Mechanical Engineering from Babol Noshirvani University of Technology in 2012. He is a master student in Iran University of Science and Technology from 2013. Mr. Fallah is currently working on his M.S. thesis on the engineering systems optimization at Iran University of Science and Technology. His research interests include numerical investigation and simulation and design optimization of engineering systems.