

Development of Two-Dimensional Convergent-Divergent Nozzle Performance Rapid Analysis Project

Yaxiong YANG^{1,a,*}, Eri Qitai^{2,b}, Qiang Wang^{3,c} and Xijuan ZHU^{4,d}

^{1,2,3} School of Energy and Power Engineering, Beihang University, Beijing 100191, China

⁴ Science and Technology on Optical Radiation Laboratory, Beijing 100854, China

^ayyx502@buaa.edu.cn, ^beriqitai@sina.com

Keywords: two-dimensional convergent-divergent nozzle; performance analysis; non-isentropic flow; radially divergent.

Abstract. A rapid nozzle performance analysis methodology and project for two-dimensional convergent-divergent nozzle have been developed. A one-dimensional isentropic flow model is employed to calculate the parameters of nozzle internal flow field. And the skin friction coefficient is used to correct the non-isentropic flow effect. Method of integral around the nozzle surface is used to compute the thrust. The divergence factor is applied to correct the nozzle thrust loss which because that the nozzle exit flow is radially divergent. Calculating the nozzle thrust coefficient in three different working conditions, including over/perfect/under expansion conditions, by using nozzle performance analysis project and numerical simulation respectively. The results of the project, numerical simulation and experiment data have a good agreement in different working conditions, and the error of thrust coefficient is less than 2%. The calculation accuracy is contented to the requirements of calculations.

Introduction

In the aero engine, the main function of nozzle is to make the gas from the gas generator continue to expand which causes the engine to produce an anti - acting thrust force. In the process of nozzle design, a large number of performance calculations must be completed because of the iterative design with aircraft and engine. Using the traditional numerical simulation method to complete all design work will take a great time. In this paper, the rapid calculation method of two-dimensional convergent-divergent nozzle performance is studied and the project has been developed which can greatly improve the efficiency of the design work. The project has higher accuracy and can be used as a tool for the initial design stage.

Rapid calculation method of nozzle performance

The rapid calculation of two-dimensional convergent-divergent nozzle performance is carried out by three steps: Firstly, the flow field parameters of the nozzle can be quickly calculated by using one-dimensional model, and the influence of the non-isentropic flow is modified by the wall friction coefficient. Then the thrust calculation formula is improved and the divergence factor is applied to correct the nozzle thrust loss which because that the nozzle exit flow is radially divergent. Finally, the thrust coefficient of the nozzle is calculated.

Ideal flow field parameters calculation

The nozzle is divided into several nodes along the axial direction, and the flow field parameters are calculated on the one-dimensional nodes. Assuming that the flow in nozzle is one-dimensional isentropic flow, the equation (1) can be derived from the continuous equation.

$$\left(\frac{A_1}{A_2}\right) - \left(\frac{P_{T2}}{P_{T1}}\right) \sqrt{\frac{T_{T1}}{T_{T2}}} \frac{Ma_2}{Ma_1} \left[\frac{1 + \frac{g-1}{2} Ma_2^2}{1 + \frac{g-1}{2} Ma_1^2} \right]^{\frac{g+1}{2(g-1)}} = 0 \quad (1)$$

Using equation (1) as an iterative equation, Mach number of each node in the whole flow field is obtained by Newton iteration method. Because the total temperature and total pressure is constant, the other parameters in the flow field can be obtained from the ideal gas state equation and the relationship between the stagnation parameters and the static parameters.

Correction of non-isentropic flow effect

In order to make the calculation result more close to the actual working condition of the nozzle and improve the calculation accuracy, the wall friction coefficient is introduced to correct the deviation between calculation and actual situation caused by the wall friction.

Wall friction coefficient C_f is defined as the ratio of wall shear stress to the gas dynamic head. In this paper, the following empirical formula is used to calculate it:

$$C_f \approx \frac{0.455}{\Omega^2 \ln^2 \left(\frac{0.06 \operatorname{Re}_L}{\Omega} \frac{m_e}{2 m_w} \sqrt{\frac{T_e}{T_w}} \right)} \quad (2)$$

In the equation, subscript e expresses the flow field parameters obtained under the ideal condition, and the other unknown parameters are calculated by the formula (3) ~ (8):

$$a = \sqrt{\frac{g-1}{2} Ma_e^2 \frac{T_e}{T_w}} \quad (3)$$

$$b = \frac{T_{aw}}{T_w} - 1 \quad (4)$$

$$T_{aw} = T_e \left(1 + \sqrt[3]{\operatorname{Pr}} \frac{g-1}{2} Ma_e^2 \right) \quad (5)$$

$$\operatorname{Re}_L = \frac{r_e U_e L}{m_e} \quad (6)$$

$$L = \frac{1}{\sqrt{p}} \sum_{n=2}^N \frac{S_n}{\sqrt{A_{n-1}} + \sqrt{A_n}} + L_0 \quad (7)$$

$$m(T) \approx m_s \left(\frac{T}{T_s} \right)^{3/2} \frac{T_s + C_s}{T + C_s} \quad (8)$$

In equations (3) ~ (8), the nozzle wall is arranged as adiabatic wall, so $T_w = T_e$; T_{aw} is the wall recovery temperature, Pr is the gas Prandtl number, taken to be 0.7; μ_s , T_s and C_s is to meet the physical parameters of the three coefficients Sutherland viscosity law, according to the actual conditions, respectively, the value taken to be 1.716×10^{-5} , 273 and 111; L is effective length, its expression is equation (7) and it can be determined by the inlet Reynolds number, and the inlet Reynolds number value taken to be 1.0×10^{-7} during calculation. S_n is the side wall area between the n-1 node and the n node.

After the friction coefficient of the wall surface was calculated, different adiabatic friction tube flow models are used to calculate the corrected internal flow field parameters of horizontal section and convergent-divergent section of nozzle.

For horizontal section, the one-dimensional equal area adiabatic friction tube flow model is taken as calculation model. Using equation (9) as the iteration equation and Newton method as the iterative method calculates the Mach number of the horizontal segment at each node.

$$\frac{Ma_2^2 - Ma_1^2}{kMa_1^2 Ma_2^2} + \frac{k+1}{2k} \ln \left[\frac{Ma_1^2 \left(1 + \frac{k-1}{2} Ma_2^2 \right)}{Ma_2^2 \left(1 + \frac{k-1}{2} Ma_1^2 \right)} \right] - 4C_f \frac{L}{D} = 0 \quad (9)$$

In equation (9), L is the axial length of the horizontal section, and D is the inside diameter of horizontal section.

For convergent-divergent section, taken divergent section for example, the one-dimensional variable area adiabatic friction tube flow model is the calculation model. The calculation equation is equation (10).

$$f(Ma, x) = \frac{Ma \left(1 + \frac{k-1}{2} Ma^2 \right) kMa^2 C_f - 2 \tan d}{1 - Ma^2 R_{in} + x \tan d} \quad (10)$$

In equation (10), R_{in} indicates the radius of the nozzle's expansion section entrance (throat radius), δ is the divergent angle. Using the classical Runge-Kutta method to solve the equation.

Improvement on calculation method of nozzle thrust

After the corrected internal flow field parameters was calculated, the calculation method of the nozzle thrust is improved, as shown in equation (11):

$$F_G = - \iint_7 V_x r (\mathbf{V} \cdot \hat{n}) dA - \iint_7 (P - P_0) \hat{n}_x dA - \sum_{i=1}^{N_i} \iint_i V_x r (\mathbf{V} \cdot \hat{n}) dA - \sum_{i=1}^{N_i} \iint_i (P - P_0) \hat{n}_x dA - \sum_{i=1}^{N_i} \iint_i 0.5 C_f r V_x^2 dA \quad (11)$$

The improved thrust calculation equation is more complicated in form, but the simple one-dimensional flow model can be used in the calculation process which can greatly improve the calculation speed.

In the actual working process of the engine nozzle, the discharge direction is not completely along the axis line of the nozzle. There will be a radial divergence phenomenon which causes the thrust loss and the calculated thrust is less than the actual thrust. The equation (12) is used to correct the thrust loss.

$$F_G' = C_A F_G \quad (12)$$

In the equation (12), the correction coefficient C_A is also called the divergent coefficient which considers the loss caused by the radial divergence of the flow line source flow and is used to correct the one-dimensional source slow thrust. The calculation equation is shown in the equation (13), δ is the divergent angle.

$$C_A = \frac{1 + \cos d}{2} \quad (13)$$

Project development and accuracy verification

Based on the study of rapid calculation method for the performance of two-dimensional convergent-divergent nozzle, we develop a project which is corresponding to the method. In order to

verify the accuracy of the project, the thrust coefficient of the nozzle is calculated by the project and the numerical simulation method separately. The nozzle geometric parameters are as follows:

The design NPR of this nozzle is 2.97, the diameter of throat is 2.74cm, the aspect ratio is 1.47, the exit diameter of nozzle is 2.98cm, the area ratio is 1.09, the convergent axial length is 5.78cm, the divergent axial length is 5.78cm, the convergent angle is 22.33° , the divergent angle is 1.21° . The geometric sketch and the mesh figure are shown in Fig.1 and Fig.2, the grid number is about 890000.

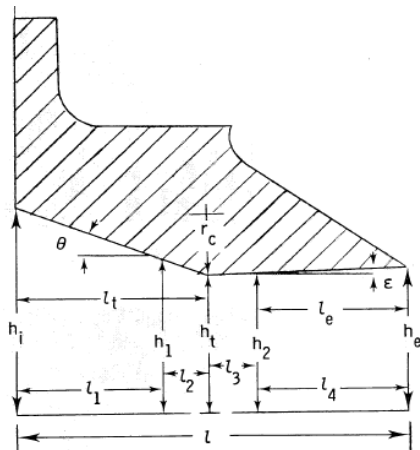


Fig. 1 The geometry of the 2D-CD nozzle

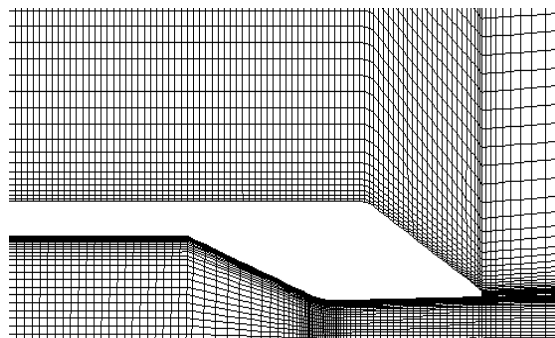


Fig. 2 The mesh of the 2D-CD nozzle

The boundary conditions are given in the reference of experimental conditions. In this paper we use the density based, coupled implicit solver, RNG k-ε turbulence model to conduct the numerical simulation.

Results and Discussion

The comparison of the results calculated by the project and the numerical simulation with the experiment data is shown in Table 1.

Table 1 The thrust coefficient comparison at different NPR

C_F \ NPR	Experiment	CFD	Project	Error
2.45	0.97970	0.98554	0.99730	1.80%
2.92	0.98622	0.98997	0.99978	1.37%
3.41	0.98801	0.98979	0.99862	1.07%
3.92	0.98741	0.98761	0.99550	0.82%
5.84	0.97264	0.97368	0.97931	0.69%

In order to show the results of the program and the numerical simulation are in agreement with the experimental data directly, Fig3 gives a comparison of the results of the three results at different NPR.

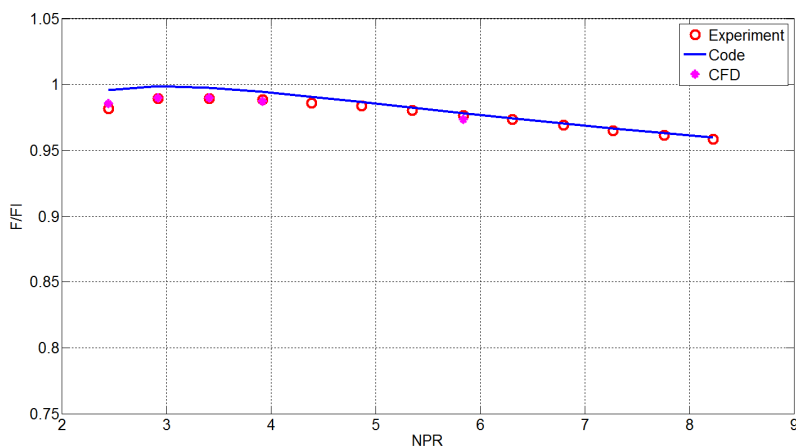


Fig. 3 The thrust coefficient of the 2D-CD nozzle at different NPR

It can be seen from Fig.3 and Table 1, the calculation error of project is very small when the flow condition in nozzle is close to the perfect expansion or under expansion. When the flow condition in nozzle is over expansion, because of the shock wave, the one dimension prediction project can not capture the position of the shock wave accurately; the accuracy of the project is decreased.

The internal static-pressure distributions along upper flap and sidewall at NPR=2.95 are shown in Fig.4 and Fig.5.

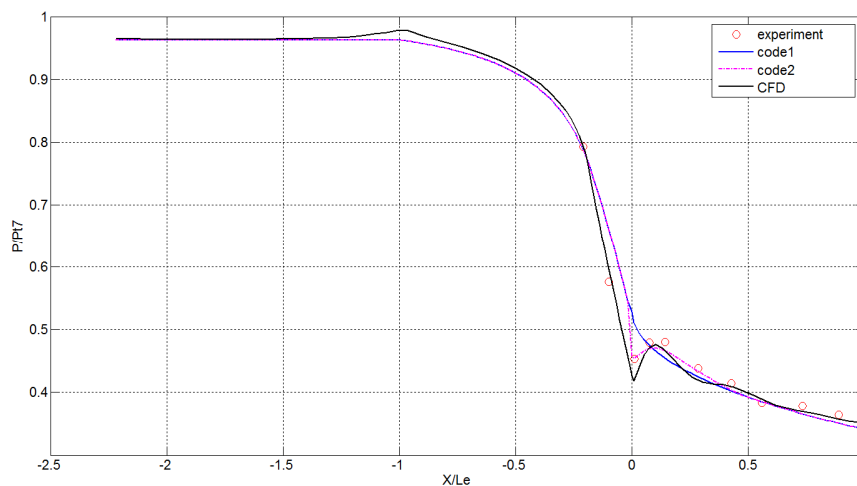


Fig. 4 The internal static-pressure distributions along upper flap at NPR=2.95

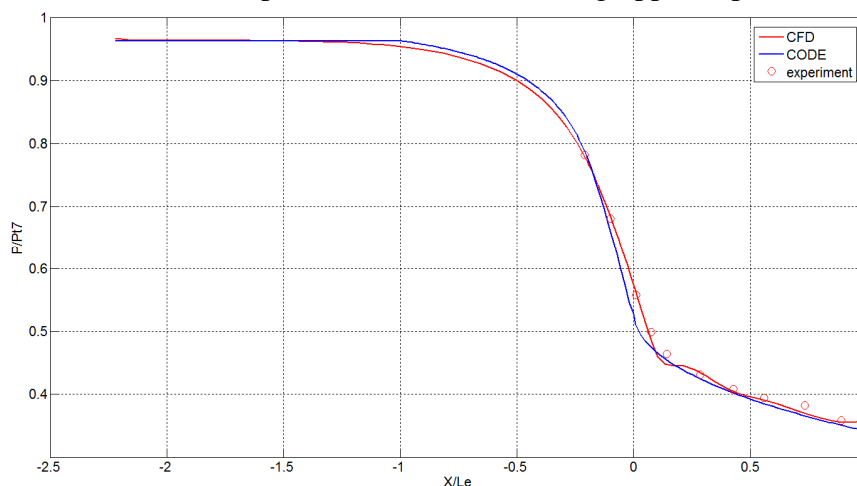


Fig. 5 The internal static-pressure distributions along sidewall at NPR=2.95

By comparison of static-pressure distributions curves in Fig.4 and Fig.5, it is shown that the calculation model in this paper is feasible, and the corrected pressure distributions along the wall are more close to the actual situation.

Conclusions

In this paper, a rapid calculation method for the performance of two-dimensional convergent-divergent nozzle is studied. A fast calculation project is developed to calculate the flow parameters and the performance parameters of the nozzle. The following conclusions are obtained by comparing with the numerical simulation results and experiment data:

- (1) In the case of the over expansion and the heavy under expansion, the thrust loss is larger and the thrust coefficient is lower. In the near perfect expansion condition, the thrust loss is smaller, and the thrust coefficient is higher.
- (2) Under different working conditions, the result of nozzle performance parameters calculated by the project can be consistent with the numerical simulation and the experiment data. The max error is less than 2%.

(3) The project developed in this paper has the characteristics of fast calculation speed and high accuracy. As long as the nozzle geometry and working parameters are known, the thrust coefficient of the nozzle can be rapidly and accurately calculated. The project is very suitable to obtain the reference value of performance parameters rapidly during the nozzle initial design stage and other issues.

References

- [1] Crag A. Hunter, An Approximate Theoretical Method for Modeling the Static Thrust Performance of Non-axisymmetric Two-Dimensional Convergent-Divergent Nozzles[R].NASA-CR-195050,1995
- [2] Paul J. Barnhart. NPAC-Nozzle Performance Analysis Code[R].NASA-CR-204129,1997
- [3] A. Matesanz, Aerodynamics Performance Prediction of Thrust-Vectoring Nozzles[J]. Journal of Propulsion and Power,1998,Vol. 14, No. 2:241-246
- [4] Jinshan Pan, Peng Shan, et al. Fundamentals of Gasdynamics[M].Beijing:National Defense Industry Press,2012. "In Chinese"
- [5] Frank M. White, Viscous Fluid Flow[M]. New York: McGraw-Hill,1974.637-648
- [6] John R. Carlson, Computational Prediction of Isolated Performance of an Axisymmetric Nozzle at Mach Number 0.90[R]. NASA-TM-4506, 1994
- [7] Mary L. Mason, The Effect of Throat Contouring on Two-Dimensional Converging-Diverging Nozzles at Static Conditions[R].NASA Technical Paper 1704, 1980