

PAPER • OPEN ACCESS

Initialization Method of Nozzle Flow Field Based on One-Dimensional Isentropic Theory and the Study of Its Application

To cite this article: HaiLong Zhao *et al* 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **608** 012012

View the [article online](#) for updates and enhancements.



IOP | ebooks™

Bringing you innovative digital publishing with leading voices to create your essential collection of books in STEM research.

Start exploring the collection - download the first chapter of every title for free.

Initialization Method of Nozzle Flow Field Based on One-Dimensional Isentropic Theory and the Study of Its Application

HaiLong Zhao¹, Ke Peng^{1*, a}, WeiHua Zhang^{1*, b}, YuZhu Fan² and JinLiang Jie¹

¹College of Aerospace Science and Engineering, National University of Defense Technology, No.109 DeYa Street, ChangSha, HuNan, P.R. China

²Unit 63629, Jiuquan Satellite Launch Center

*Corresponding author: ^apengke.nudt@nudt.edu.cn; ^bzhangweihua@nudt.edu.cn

Abstract. The two-dimensional axisymmetric nozzle is initialized by two methods which are the hybrid initialization method of the commercial computational fluid dynamics software Fluent and the initialization program based on the one-dimensional isentropic theory. The first-order upwind format is combined with the SA model to solve the two-dimensional axisymmetric nozzle under the same boundary conditions. The convergence process is obtained in the process of numerical calculation, and the influence of two different initialization methods on the convergence process is analysed. The results show that the initialization method based on the one-dimensional isentropic theory effectively improves the convergence of the nozzle numerical model calculation process, accelerates the convergence speed and improves the calculation efficiency.

1. Introduction

The nozzle is an important part of the aircraft for energy conversion. It can transform the internal energy of high temperature and high pressure gas into kinetic energy, generate reaction force, and provide continuous and stable power for the aircraft. Therefore, it is of great significance to study the flow field of the nozzle. At present, numerical simulation, theoretical analysis, experimental research and other methods are often used in the study of flow field problems. These methods are studied from different perspectives and complement each other. For the nozzle flow field research, compared with the experimental research method, the numerical simulation method is more economical and convenient. However, when we use numerical simulation method to study the nozzle flow field problem, it is possible that the process of convergence speed of the nozzle numerical simulation is too slow and sometimes it can be diverging because of the complexity of the nozzle flow field and the defects of the numerical calculation method.

Therefore, it is of great significance to solve the problems of slow convergence and easy dispersion in numerical simulation of nozzle flow field. As the most simplified flow model, one-dimensional isentropic flow model can give analytical solution or semi-analytical and semi-numerical results, which is helpful to understand the actual three-dimensional flow law. The initialization program of numerical calculation of nozzle is written by using the one-dimensional isentropic theory, and the numerical simulation calculation of nozzle flow field is initialized by using this initialization problem. The results of one-dimensional isentropic flow are taken as the initial



conditions of numerical calculation of nozzle, which is helpful to improve the convergence in numerical calculation of nozzle.

2. Initialize Method

2.1 One-dimensional Isentropic theory of Nozzle

For the one-dimensional flow in the nozzle, the sectional area of the nozzle is set as A , which is function of the axial distance X (the inlet center as the origin point) of the nozzle. The parameters inside the nozzle only change with the change of X . The continuity equation, momentum equation and energy equation of one-dimensional isentropic flow are given below, as shown in (1)-(3).

$$\text{continuity equation:} \quad \rho_1 V_1 A_1 = \rho_2 V_2 A_2 \quad (1)$$

$$\text{momentum equation:} \quad p_1 A_1 + \rho_1 V_1^2 A_1 + \int_{A_1}^{A_2} p dA = p_2 A_2 + \rho_2 V_2^2 A_2 \quad (2)$$

$$\text{energy equation:} \quad h_1 + \frac{V_1^2}{2} = h_2 + \frac{V_2^2}{2} \quad (3)$$

In order to give an analytical solution, the complete gas equation (4) and the complete gas enthalpy equation (5) need to be supplemented[1].

$$\text{complete gas equation:} \quad p = \rho R T \quad (4)$$

$$\text{complete gas enthalpy equation:} \quad h = c_p T \quad (5)$$

The analytical solution can be obtained from the equation (1)-(3). Firstly, the Mach number in the nozzle is only related to the area ratio, and the relational expression is (6).

$$\left(\frac{A}{A^*} \right)^2 = \frac{1}{Ma^2} \left[\frac{2}{\gamma + 1} \left(1 + \frac{\gamma - 1}{2} Ma^2 \right) \right]^{\frac{\gamma + 1}{\gamma - 1}} \quad (6)$$

For a given nozzle, its cross-sectional area A is a function of X , so the relation between the Mach number and X can be obtained from equation (6). After the Mach number is solved, the changes of pressure, density, temperature and others are functions of Mach number, and the equation (7)-(9) can be obtained[2].

$$\frac{P}{P_0} = \left(1 + \frac{\gamma - 1}{2} Ma^2 \right)^{-\frac{\gamma}{\gamma - 1}} \quad (7)$$

$$\frac{\rho}{\rho_0} = \left(1 + \frac{\gamma - 1}{2} Ma^2 \right)^{-\frac{1}{\gamma - 1}} \quad (8)$$

$$\frac{T}{T_0} = \left(1 + \frac{\gamma - 1}{2} Ma^2 \right)^{-1} \quad (9)$$

Among them, P_0 ; T_0 ; ρ_0 ; γ ; A^* are the total pressure at the inlet of the nozzle, the total temperature, the density, the specific heat ratio of the gas, and the area of the throat of nozzle. According to the above conditions, the physical parameters in the nozzle can be determined once the nozzle shape, fluid parameters, and inlet conditions are determined.

2.2 Initialization Program Based on One-dimensional Isentropic theory

According to the theory of 1.1, when the gas parameter conditions of the nozzle inlet and the shape of the nozzle are determined, the flow parameters inside the entire nozzle can be obtained. The physical parameter values solved in the one-dimensional isentropic theory are used as the initial values of the nozzle simulation calculation, and the physical parameters obtained by the one-dimensional steady flow theory are analyzed as initial conditions to improve the numerical calculation of the nozzle flow field.

First, after meshing the nozzle, the commercial computational fluid dynamics software Fluent is called to generate the initial IP file, and the coordinate information of all the grid points of the nozzle can be obtained; then a file containing the nozzle generatrix coordinate points and the inlet the gas parameters in the program. The calling program calculates the area of each grid node to obtain the area ratio of the position, and then combines the inlet gas parameters into the formula of equation (1)-(9) to obtain the physical parameters of the mesh node. After looping through all the grid points, an initial nozzle flow field can be obtained, and a new IP file is output, which is imported into Fluent as the initial flow field for numerical simulation calculation, The block diagram is shown in figure 1.

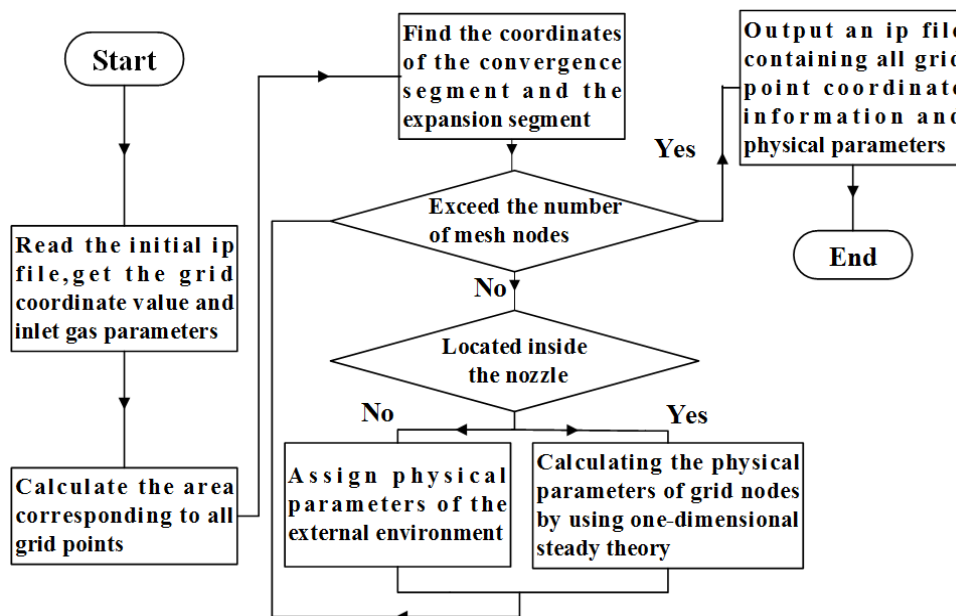


Figure 1. Initialization block diagram based on one-dimensional isentropic theory

3. Research Object and Method

3.1 Computational model and meshing

In this paper, a two-dimensional axisymmetric nozzle is taken as the research object, and the convergence process under the same boundary conditions but different initial conditions are mainly analyzed. The geometry is shown in figure 2 and the geometric parameters are shown in Table 1.

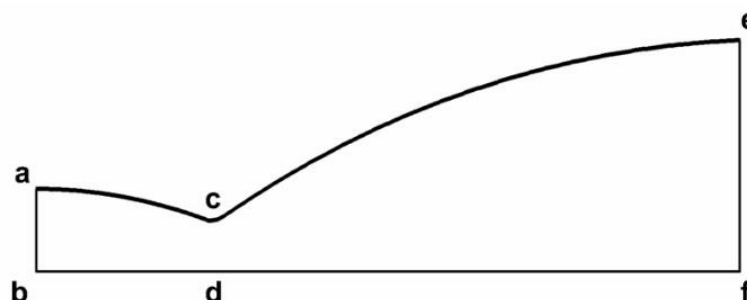
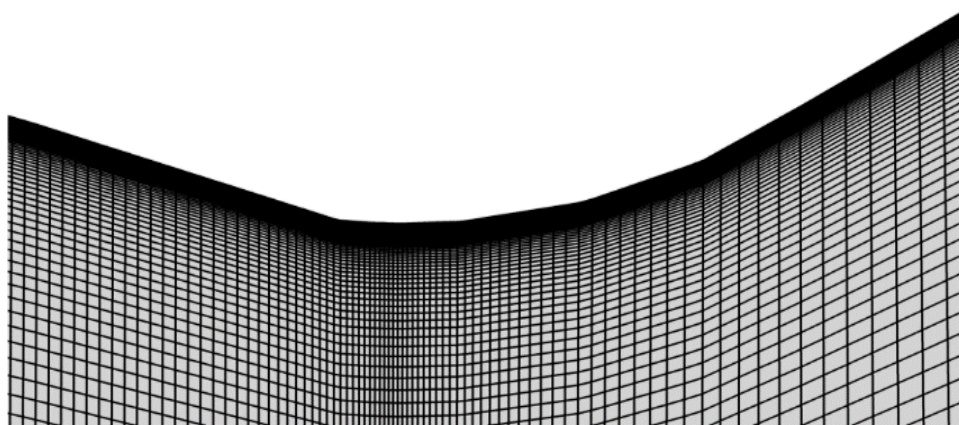


Figure 2. Geometric model of the nozzle**Table 1.** Nozzle geometry basic parameter

Parameter	Value
Nozzle inlet radius(ab)	0.054(m)
Laryngeal radius(cd)	0.0335(m)
Nozzle outlet radius(ef)	0.15(m)
Nozzle length(bf)	0.465(m)
Expansion ratio	20

The mesh division of some areas of the two-dimensional axisymmetric nozzle is shown in figure 3. In order to reduce the influence of the grid on the numerical simulation calculation, a structured grid is used to encrypt the wall, throat, entrance and exit to ensure that the y^+ of the first layer is closer to 1 [3][4], the total number of grids is 1000×200 .

**Figure 3.** Nozzle partial area grid

3.2 Numerical calculation parameter setting

In this paper, the SA turbulence model is selected to solve the problem. It is a RANS model with only a small amount of computation for solving a modified eddy viscosity transport equation. In the modified form, the eddy viscosity in the near-wall region is easier to solve and is often used to simulate the inclusion of wall jets [5]. The problem is that the simulation results are better in the boundary layer flow of the inverse pressure gradient. For the discrete format of various parameters, the first-order upwind format is selected. The first-order upwind format has the characteristics of high stability and fast calculation speed, which is beneficial to improve the calculation efficiency. For the selection of the initialization method, one method is to use the hybrid initialization method provided by the commercial computational fluid dynamics software Fluent to initialize, and the other method is to initialize using the initialization program which is written based on the one-dimensional isentropic theory, and compare the convergence process of the two methods. In the numerical calculation, the Coulomb number also has a great influence on the stability and convergence of the calculation, so the Coulomb number in all numerical calculation is set to 0.5.

For this physical model, as shown in figure 2. The nozzle inlet ab is set as a pressure inlet condition, the nozzle outlet ef is set as a pressure outlet condition, the wall surface ace is set as a wall condition, and bdf is set as a symmetry axis condition. In this paper, three pressure inlet conditions are calculated, which are 8MPa, 10MPa and 12MPa, and the total temperature is set to 3568.923K. The

pressure and temperature of the pressure outlet condition are set to 101325Pa and 300k.

The density of the gas is calculated using the ideal gas model. The constant pressure heat ratio of the gas is set to 3724.088k, and the thermal conductivity is solved by the kinetic-theory. The gas viscosity coefficient is solved by the Sutherland's law of three coefficients, as shown in equation (10).

$$\mu = \mu_0 \left(\frac{T}{T_0} \right)^{3/2} \frac{T_0 + T_s}{T + T_s} \quad (10)$$

In equation (10), the reference temperature T_0 is taken as 273.15k; T_s is the Sutherland constant, it is related to the nature of the gas. μ_0 is the reference viscosity coefficient when the temperature is T_0 .

4. Calculation Results and Analysis

First, the flow field pressure and velocity distribution after initialization using two initialization methods are given, when the inlet pressure is 8MPa, as shown in figure 4-figure 5. The physical parameters obtained by the one-dimensional isentropic theory are only related to the area ratio and are a function of the change with X; and the hybrid initialization method is based on the given boundary conditions to pre-iterate the Laplace equation, as shown in equation (11). The initial distribution of velocity and pressure in the calculation region is solved, and other physical quantities (temperature, turbulence) are automatically obtained from the average interpolation of the flow field.

$$\begin{aligned} \nabla^2 \varphi &= 0 \\ V &= \nabla \varphi \\ \nabla^2 p &= 0 \end{aligned} \quad (11)$$

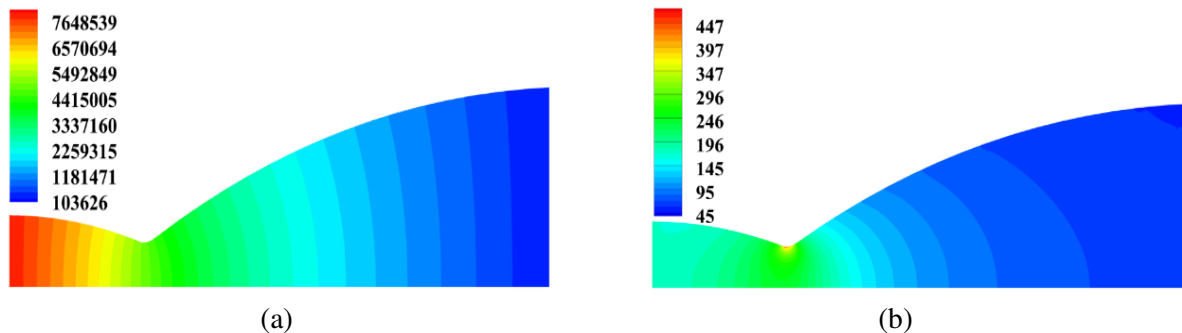


Figure 4. Flow field pressure and velocity distribution after hybrid initialization

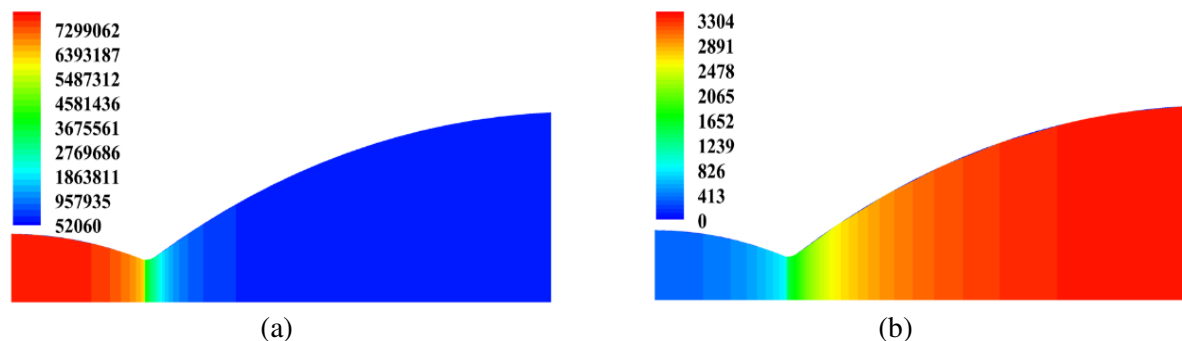


Figure 5. Flow field pressure and velocity distribution after initialization based on isentropic theory

It can be seen from figure 4 and figure 5 that the pressure fields obtained by the two initial methods are similar, but there is a big difference in the velocity field. It can be seen that the velocity field of the hybrid initialization is a certain area as the “source point”. Diffusion to the surrounding, the speed gradually decreases; and the velocity field using the one-dimensional isentropic theory is still only a function related to the area ration. In the contraction section of the nozzle, the gas velocity is subsonic, in the expansion section of the nozzle, the gas velocity is supersonic. The continuous

residual variable process of two different initialization methods under three inlet pressure conditions is analyzed. As shown in figure 6-figure 8.

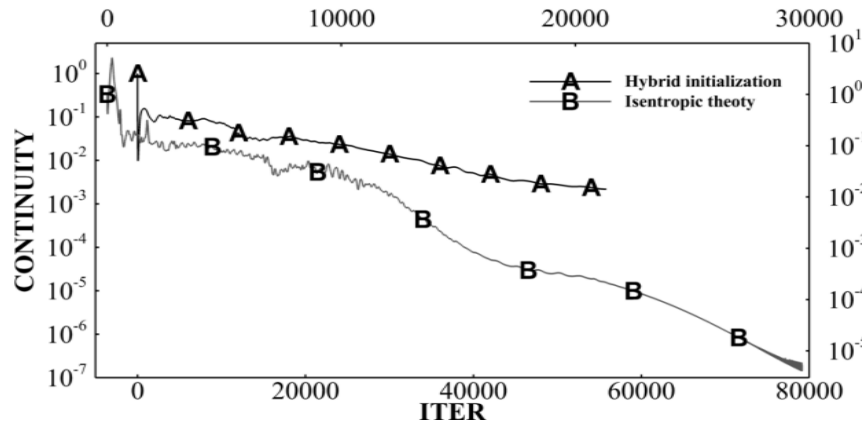


Figure 6. 8MPa continuous residual

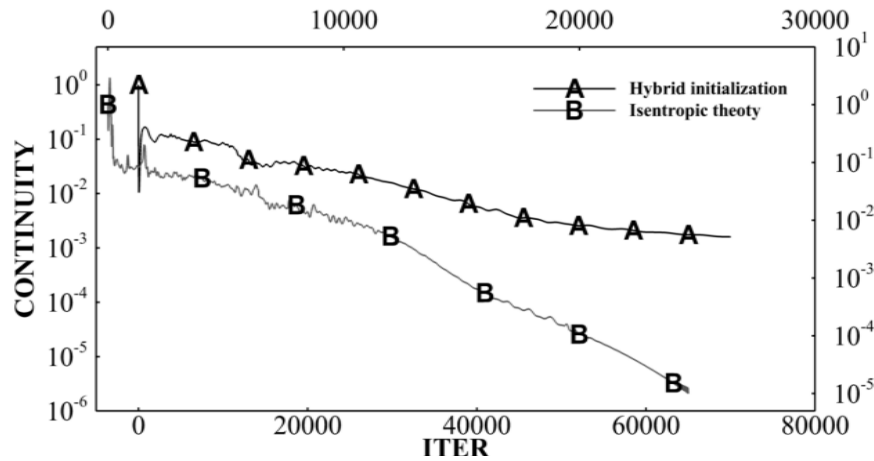


Figure 7. 10MPa continuous residual

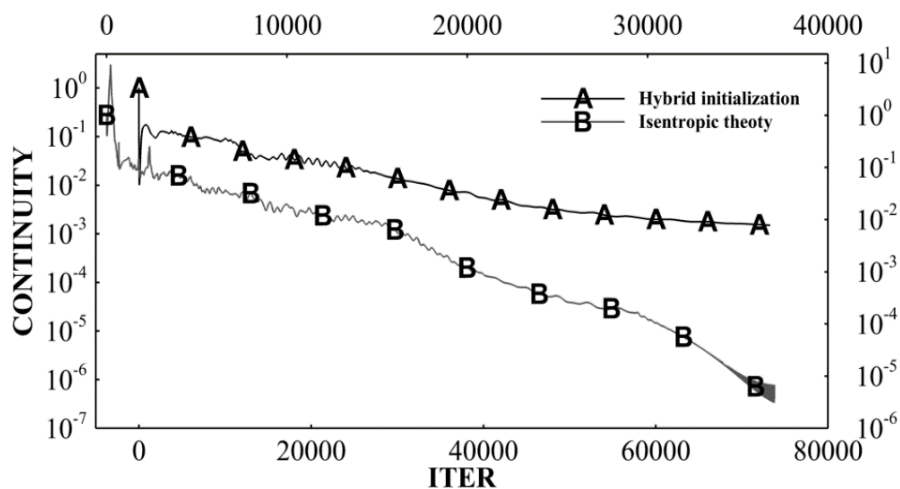


Figure 8. 12MPa continuous residual

From figure 6- figure 8, it can be seen that the numerical calculation process initialized using the one-dimensional isentropic theory is significantly accelerated, and the value that can finally converge

is reduced by nearly three orders of magnitude compared to the use of hybrid initialization. As the inlet pressure continues to increase, the number of convergence steps of the calculation process using the hybrid initialization method increases from 60000 steps to nearly 80000 steps, while the calculation process using the one-dimensional isentropic theory method has a convergence step of about 30000 steps. There is no significant improvement in the number of steps. It can be seen from figure 4- figure 5 that the initialization method based on the one-dimensional isentropic theory method actually accelerates the convergence speed of the nozzle numerical simulation calculation process, effectively reduces the calculation time and improve the efficiency of the nozzle numerical calculation.

The final calculation results obtained by the two methods are compared below. The inlet pressure condition is 8 MPa, and the calculated pressure, velocity field and streamline diagram are shown in figure 9-figure 10.

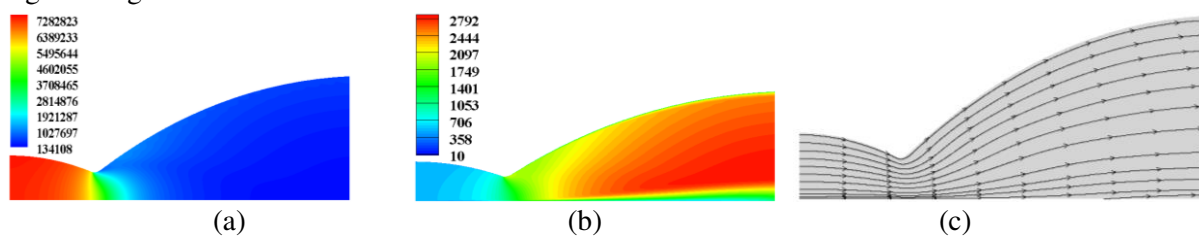


Figure 9. After hybrid initialization, pressure, velocity, streamline results

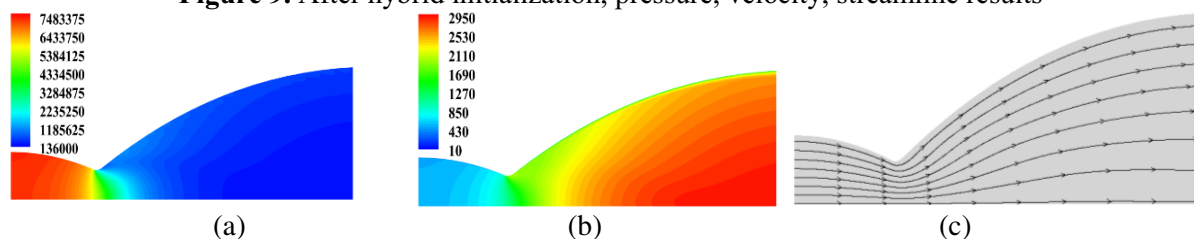


Figure 10. After one-dimensional steady initialization, pressure, velocity, streamline results

Comparing the final results obtained by the two initialization methods, we can see that there are some differences in the numerical results of the two final results, but the difference is not large, and the pressure, velocity, and streamline distribution are basically the same. The results obtained are reasonable. Since the iterative calculation is more sensitive to the initial value. Therefore, the initial conditions are well specified and have a great influence on the convergence of the calculation process. Before the iterative calculation, giving an initial value close to the calculated results is beneficial to improve the convergence of the iterative calculation. Conversely, if the initial value is far from the final value, it may cause convergence difficulties. Comparing figure 10 with 5, we can see that the initial velocity field based on the one-dimensional isentropic theory is closer to the final calculated velocity field, that is, the initial field obtained by the one-dimensional isentropic theory is better than the hybrid initialization method. Therefore, the initialization method based on the one-dimensional isentropic theory has a good effect on improving the convergence of the nozzle numerical simulation calculation process.

5. Conclusion

Comparing the two initialization methods, we can see that the one dimensional isentropic theory is used to initialize the Laval nozzle. Under the same conditions, it is better than the Fluent's own hybrid initialization method, which effectively improves the convergence and stability in the calculation process. It also improves the computational efficiency and has a more stable calculation process. Therefore, the numerical simulation of the problem of nozzles, if combined with the one-dimensional isentropic theory, can greatly improve the calculation efficiency and improve the convergence and stability of the calculation process, which is of great significance for the numerical simulation of the

nozzle.

The numerical calculation of the nozzle is a complicated process, and its convergence is often not only affected by the initial conditions, but also closely related to various factors such as model simplification, meshing quality, discrete method, solution method and boundary conditions[6][7]. This paper mainly studies the influence of initial conditions on the convergence, so some simplifications are made, and there are still some shortcomings. The factors affecting the convergence of nozzle numerical calculations will be studied in the later stage.

References

- [1] HaiFeng H,FuTing B,Qiang C,Yang L. Flow separation and aeroelastic coupling analysis in overexpanded rocket nozzles..*Journal of Solid Rocket Technology* 2011(6)
- [2] QingWei W,Bo L,RuGen W.Numerical Simulation of Two-dimensional Nozzle Using Fluid Injection for Throat Area Control..*Aeronautica Et Astronautica Sinica*.2009(2)
- [3] Ping W,ChangPing L,BaiSong C.Laval Nozzle Flow Analysis Based on CFD Computation. *Aeronautical Computing Technique*.2009(2)
- [4] WenXiang Z,JinQuan H,RenZhi Z.Improvement of Laval nozzle calculation model and simulative verification in aero-engine performance calculation..*Journal of Aerospace Power* 2009(11)
- [5] PengFei L,MinYi X,FeiFei W..*Proficient in CFD Engineering Simulation and Case Combat* (Beijing: Posts and Telecom Press) p123
- [6] WenZhi L,NaiRen Z,ChunLin-Z,YongZhong Z.Contour design and numerical calculation of certain solid propellant rocket motor nozzle..*Journal of Engineering Design* 2006(02)
- [7] A.A.Verevkin,Yu.M.Tsirkunov.Flow of a dispersed phase in the laval nozzle and in the test section of a two-phase hypersonic shock tunnel *Journal of applied mechanics and technical physics*,2008,5(5):789-798