July - August 2020
ISSN: 0193-4120 Page No. 6136 - 6143

Research on GRP Scheme for Compressible Inviscid Hydrodynamics Equations Based on Nonlinear Algebraic Equations

Huijing Zhan[1,*]
[1]College of Mathematics and Physics, Fujian University of Technology, Fuzhou, Fujian, China, 350118

Article Info
Volume 83
Page Number: 6136 - 6143
Publication Issue:
July - August 2020

Article History
Article Received: *25 April 2020*
Revised: *29 May 2020*
Accepted: *20 June 2020*
Publication: *28 August 2020*

Abstract

With the support of intelligent technology, fluid mechanics has gradually become an important means and method to solve practical engineering problems. Fluid mechanics based on nonlinear algebraic equations can directly solve and operate nonlinear algebraic equations, so as to solve the basic laws of fluid motion. Based on this, this paper first analyses the numerical methods of computational fluid dynamics, introduces the finite volume method, diffusion equation, viscosity and compressibility of fluid. Secondly, the flow of compressible fluid is studied. Finally, the GRP scheme for solving compressible inviscid hydrodynamic equations based on nonlinear equations is studied and analysed.

Keywords: *GRP Scheme, Compressible Inviscid Hydrodynamics Equations, Nonlinear Algebraic Equations;*

1. Introduction

With the rapid iteration and development of computer information technology represented by big data and artificial intelligence, it has been widely used in various fields and achieved fruitful results. Fluid mechanics based on computer technology can directly solve and operate nonlinear algebraic equations, so as to solve the basic laws of fluid motion. With the rapid progress of fluid mechanics under the support of intelligent technology, it has been widely used in many fields as shown in Figure 1, and has become an important means and method to solve practical engineering problems.

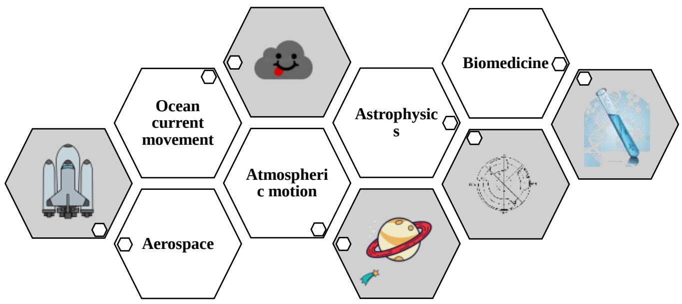


Figure1. Theapplicationfields of fluid mechanics.

Fluid is essentially composed of molecules, and these molecules have a much larger gap between each other than the size of the molecules themselves[1]. Based on these molecular gap scales, we can get the statistical average of the microscopic physical quantities of the fluid molecules as the macroscopic physical quantities of the fluid. At a scale much larger than the intermolecular gap scale, a mass of fluid material, which contains a large number of fluid molecules, can be used for macro physical quantity statistics. In addition, the scale of

Published by: The Mattingley Publishing Co., Inc.

TEST Engineering & Management
July - August 2020
ISSN: 0193-4120 Page No. 6136 - 6143

the fluid mass is much smaller than the macro scale of the problems in fluid mechanics, so that it can be regarded as a point, called a particle, and the fluid is composed of these continuously distributed particles.

Fluid mechanics studies the macro motion of fluid. The continuum model considers that the matter is continuously distributed in the whole space occupied by the material without any gap. The macroscopic physical quantity of fluid is a continuous function of space point and time. When the scale of the problem studied is much larger than the scale of molecular structure and molecular gap, the fluid can be regarded as a continuous medium.

1.1. Motion of fluid

Fluid is not a closed system, and it does not care about the whole movement of the system. It has an extremely large number of particles, and the movement between particles will not be consistent. The fluid can be divided into many fluid clusters on a larger scale than particles. The motion of particles in these fluid clusters will not be completely consistent, but it can be considered that the inconsistency has the rule of linear distribution.

The fluid at the initial moment is divided into many micro clusters, each of which is numbered and identified by a . Then a mathematical model is established for the physical quantities of each fluid micro cluster:

$$f = f(a, t) \tag{1}$$

1.2. Motion of fluid

For the irrotational motion of inviscid compressible fluid, the overall fluid field is shown in Figure 2, and the potential function of irrotational motion:

$$\frac{D\boldsymbol{v}}{Dt} = \boldsymbol{F}_b - \frac{1}{\rho}\nabla p \tag{2}$$

$$\frac{\partial \phi}{\partial t} + \frac{|\boldsymbol{v}|^2}{2} + \pi + P = C(t) \tag{3}$$

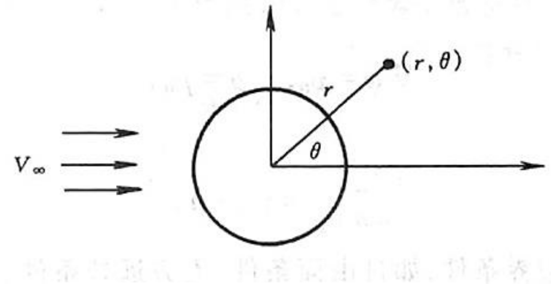
The fixed wall shown in Fig. 2 has:

$$\frac{\partial \phi}{\partial r} = 0 \tag{4}$$

At the infinity shown in Figure 2 has:

$$\frac{\partial \phi}{\partial r} = V_\infty \cos\theta \tag{5}$$

For the plane irrotational motion of inviscid compressible fluid, it can be solved by solving the analytic function of complex variable function.



Uniform flow around a cylinder

Figure2. Non rotational motion of inviscid compressible fluid.

With the continuous development of basic theory and computer technology, fluid calculation has made great progress. A series of methods and theories in fluid mechanics, such as finite volume method, finite difference method and finite element method, have been applied continuously. In addition, with the deepening of the influence of relativity on the whole physical system, many interdisciplinary research fields are emerging, and relativistic fluid mechanics is playing an increasingly important role in many related disciplines such as physics. Based on the above background, it is of great theoretical and practical value to study the GRP scheme for compressible non viscous fluid dynamics equations based on nonlinear algebraic equations.

2. Numerical methods of computational fluid dynamics

2.1. Finite volume method in computational fluid dynamics

For a conserved physical quantity in any control body, the source in the control volume is equal to the sum of the change rate of the physical quantity in the control body and the net flux passing through the boundary of the control body. The total flux passing through the boundary of the control body consists of convection and diffusion. If φ is the conserved

Published by: The Mattingley Publishing Co., Inc.

quantity of unit mass fluid, the general scalar convection diffusion equation can be expressed as follows:

$$\frac{d}{dt}\left(\rho V\varphi\right) + \sum_{faces}(C\varphi - \Gamma A\frac{\partial\varphi}{\partial n}) = SV \qquad (6)$$

$$C = \rho u_n A \qquad (7)$$

The finite volume method directly discretizes the above formula, as shown in Figure 3. In the case of only considering the steady-state problem, the first term on the left of equation 6 is zero.

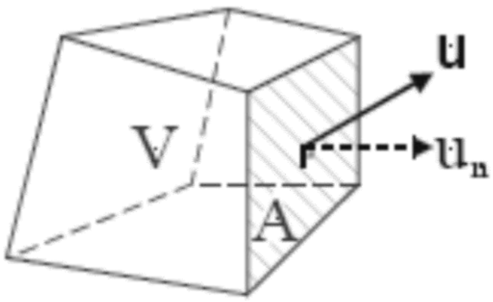


Figure3. Schematic diagram of finite volume method in fluid mechanics.

2.1.1. FVM computing grid

The finite volume method defines the geometry of the fluid field solution domain and divides the solution domain into computational grids, that is, a group of non-overlapping finite bodies or elements. The integral equation is discretized based on the above-mentioned elements, that is, approximated by node values. The discrete equation is solved numerically. The computational grid can be structured grid or unstructured grid, Cartesian grid. The commonly used grid forms include the storage method based on cell centre and the storage mode based on cell vertex, as shown in figure 4 below, and not all variables must be stored in the same location.

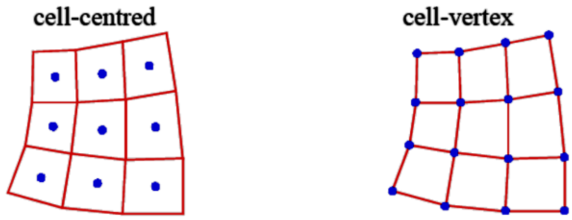


Figure 4. Grid form of FVM computing grid pair.

2.1.2. Diffusion equation

The One-dimensional Steady-state convection diffusion equation is mainly based on the simplification of the problem analysis and the manual calculation of the discrete equation. The one-dimensional diffusion equation can be directly extended to two-dimensional or three-dimensional. In fact, the flux is discretized along the coordinate direction, i.e. along the *i, j, k* lines respectively. Many important theoretical problems are one-dimensional.

For one-dimensional control volume, the conservation relationship of physical quantity is flux e-flux w = source, as shown in Figure 5, where flux is the transport rate across the cell surface. If φ is the transport volume per unit mass, thenthe total flux is the sum of convection flux and diffusion flux.For the pure diffusion problem, there are the following differential equations and integral equations:

$$\frac{d}{dx}\left(-kA\frac{dT}{dx}\right) = 0 \qquad (8)$$

$$\left[-kA\frac{dT}{dx}\right]_w^e = 0 \qquad (9)$$

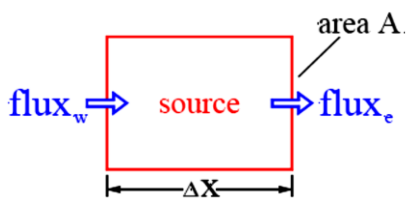


Figure 5. Schematic diagram of one-dimensional convection diffusion process.

In addition, the calculation of flux on boundary interface is shown in the following formula, and the boundary conditions are shown in Fig. 6.

$$\frac{T_1 - T_{B1}}{\Delta x/2} - \frac{T_2 - T_1}{\Delta x} = 0 \qquad (10)$$

$$\frac{T_2 - T_1}{\Delta x} - \frac{T_3 - T_2}{\Delta x} = 0 \qquad (11)$$

$$\frac{T_3 - T_2}{\Delta x} - \frac{T_4 - T_3}{\Delta x} = 0 \qquad (12)$$

$$\frac{T_5 - T_4}{\Delta x} - \frac{T_{B2} - T_5}{\Delta x/2} = 0 \qquad (13)$$

Figure 6. Boundary conditions for calculating flux at boundary.

2.2. Compressibility of fluid

The compressibility of fluid is that the volume decreases with the increase of pressure at a certain temperature. At a certain temperature, the higher the pressure is, the smaller the fluid volume compressibility coefficient is; with the increase of pressure, the compressibility of fluid decreases. The smaller the bulk modulus of fluid is, the more compressible the fluid is. Generally speaking, the compressibility of liquid fluid is much smaller than that of fluid. In addition, under a certain pressure, the volume of fluid increases with the increase of temperature, and at a certain pressure, the higher the temperature, the smaller the expansion coefficient of fluid, and with the increase of temperature, the expansibility of fluid decreases.

2.3. Viscosity of fluid

The characteristic of tangential resistance produced by relative motion between fluid layers is the performance of fluid viscosity. With the increase of temperature, the viscosity of fluid increases while the viscosity of liquid decreases. The ratio of dynamic viscosity to density is called kinematic viscosity. Ideal fluids have no viscosity. The viscosity of a real fluid exists whether it is at rest or in dynamic state. Viscosity makes the fluid have the ability to resist shear deformation and hinder fluid. To overcome the viscous resistance and maintain the fluid will inevitably lead to energy consumption.

The tangential stress acting on the fluid layer is directly proportional to the velocity gradient between two adjacent layers. When the fluid fluids, any two adjacent layers of fluid are resistant to each other, and the resisting force is shear force, also known as internal friction force, viscous force and viscous friction force.

2.3.1. Continuum hypothesis of fluid

The assumption of continuous medium of fluid mainly includes infinitesimal volume, the size of fluid particles is much larger than the distance between molecules, and the distance between particles is not greater than the distance between molecules, that is, there is no gap between particles. Fluid is a continuous medium composed of numerous fluid particles with continuous distribution. The Euler equilibrium differential equation is as follows:

$$\left(f_x + f_y + f_z\right) - \frac{1}{\rho}\left(\frac{\partial p}{\partial x} + \frac{\partial p}{\partial y} + \frac{\partial p}{\partial z}\right) = 0 \qquad (14)$$

2.3.2. Methods of fluid field research

Lagrange method and Eulerian method are two important means and methods to study the fluid field. The Euler method analyses the velocity field and expresses the change rate of the physical quantity of fluid particles with time as two parts: the local change rate caused by instability and the migration change rate caused by inhomogeneity[2]. In the process of fluid, the surface of the system is usually constantly deformed, and the quality of the fluid is determined. The position of the fluid system changes with the movement.

3. Fluid of compressible fluid

When the velocity of air fluid is far less than the speed of sound, the change of density can still be ignored. When the velocity of fluid is close to or even exceeds the speed of sound, if the fluid is disturbed, it will inevitably cause great pressure changes, so that the density and temperature will also change significantly, and the fluid state of fluid

July - August 2020
ISSN: 0193-4120 Page No. 6136 - 6143

will have a fundamental change. At this time, the influence of compressibility must be considered.

Due to the non-uniform density in the fluid field of compressible fluid, the movement of compressible fluid has some special properties compared with that of incompressible fluid. The propagation velocity of sound is a parameter or a standard to judge the influence of fluid compressibility on fluid. The greater the compressibility of fluid is, the smaller the speed of sound is. If the fluid is disturbed, the disturbance will travel in the form of waves in the fluid; if the disturbance is small, then the propagation speed is certain, and this speed is the speed of sound propagation - sound speed.

3.1. Sound speed and Mach number

A long straight pipe with A cross-sectional area is filled with static fluid. When the piston moves to the right at a small speed dv, the fluid near the right side of the piston is pressurized, and the pressure increases dp. The weak pressure disturbance generated will propagate to the right according to this, as shown in Figure 7.

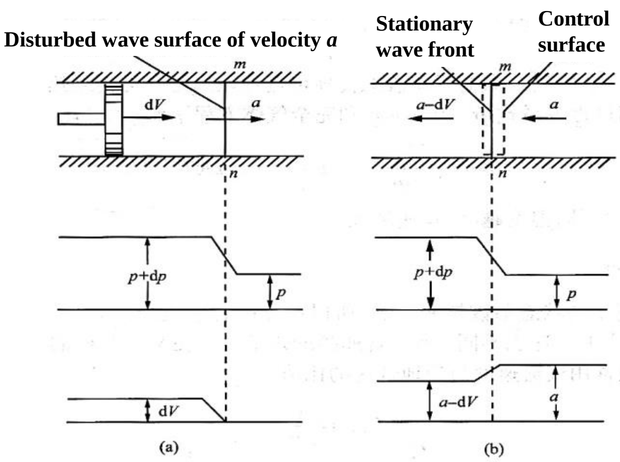


Figure7. Schematic diagram of straight pipe filled with static fluid.

The pressure, density and temperature of the undisturbed fluid in front of the compression wave are p, ρ and T respectively. After the wave, the disturbed fluid moves to the right with the same small velocity V as that of the piston. When the pressure increases to $p + \mathrm{d}p$, the density and temperature also increase to $\rho + \mathrm{d}\rho$ and $T + dT$.

Taking the left and right sides of the wave surface as the control surface, the volume of the control volume in the control plane is zero. The mass conservation principle is applied to the control body. The fluid rate in and out of the control surface per unit time is equal:

$$\rho a A = (\rho + d\rho)(a - dV)A \tag{15}$$

The compression wave is very thin and the friction force acting on the wave is negligible. According to the momentum theorem, along the fluid direction, the rate of change of momentum of a fluid is equal to the sum of the pressures acting on the fluid:

$$(a\rho A dt)\frac{[-(a-dV)-(-a)]}{dt} = [(p+dp)-p]A \tag{16}$$

The propagation velocity of the weak disturbance wave is as follows:

$$a = \sqrt{\frac{dp}{d\rho}} \tag{17}$$

The above formula is a general expression of sound speed, which is suitable for any continuous medium. Therefore, the value of sound speed reflects the compressibility of fluid. The weak disturbance wave propagates rapidly, which can be approximately regarded as a reversible adiabatic process, i.e. isentropic process. For an ideal fluid, the higher the temperature, the greater the speed of sound.

Mach number reflects the ratio of inertial force to elastic force, which is the standard to judge the influence of fluid compressibility on fluid[3]. According to the Mach number, the fluid of compressible fluid can be divided into subsonic fluid, transonic fluid, supersonic fluid and hypersonic fluid. For the fluid of compressible fluid, the thermodynamic state changes with the change of density. Therefore, the state equation and process equation in thermodynamics must be considered together to solve the fluid problem.

3.2. One dimensional steady isentropic fluid of compressible fluid

Continuity equation:

Published by: The Mattingley Publishing Co., Inc.

July - August 2020
ISSN: 0193-4120 Page No. 6136 - 6143

$$\left.\begin{array}{l}\rho_1 V_1 A_1 = \rho_2 V_2 A_2 \\ \rho V A = const\end{array}\right\}(18)$$

Its differential form is as follows:

$$VAd\rho + \rho AdV + \rho VdA = 0 \quad (19)$$

3.2.1. One dimensional fluid equation of compressible fluid

The Euler motion differential equation of ideal fluid in one-dimensional fluid is as follows:

$$\rho\frac{dV}{dt} = \rho f_x - \frac{dp}{dx}(20)$$

The sum of the changes of pressure energy and kinetic energy along the total fluid direction of an ideal fluid is zero, that is to say, the sum of the two energies does not change along the total fluid direction.

3.2.2. One dimensional energy equation of compressible fluid

Combined with isentropic fluid and integral equation along the fluid tube, the energy equation (Bernoulli equation for compressible fluid) of one-dimensional steady isentropic fluid of ideal fluid is obtained:

$$\frac{k}{k-1}\frac{p}{\rho} + \frac{V^2}{2} = const(20)$$

The existence of friction only transforms the mechanical energy consumed in resisting friction into heat energy, which is added to the fluid again, thus increasing the entropy in the fluid. So, in adiabatic fluid, the total energy does not change. In one-dimensional steady isentropic fluid of fluid, the sum of pressure potential energy, kinetic energy and internal energy per unit mass of fluid flowing through any effective section remains unchanged.

3.3. Isentropic flow of compressible fluid in nozzle

The flow of compressed fluid in nozzle is actually a kind of fluid flow in variable cross-section pipe. The law of velocity and pressure with cross-section is related to the relationship between fluid parameters and cross-section. The relationship between density change rate and velocity change rate, combined with the definition of motion equation and sound speed, is as follows:

$$VdV = -\frac{dp}{\rho} = -\frac{dp}{d\rho}\frac{d\rho}{\rho} = -a^2\frac{d\rho}{\rho}(21)$$

That is to say, in subsonic flow, the relative change of density is less than that of velocity, while in supersonic flow, the relative change of density is greater than that of velocity. This difference leads to the essential difference between subsonic and supersonic velocity in the relationship between velocity and channel cross-section shape.

3.3.1. Relationship between fluid parameters and cross section

The relationship between the change rate of sectional area and the rate of change of velocity can be obtained by the differential equation of continuity equation:

$$\frac{\mathrm{d}A}{A} = Ma^2\frac{\mathrm{d}V}{V} - \frac{\mathrm{d}V}{V} = (Ma^2 - 1)\frac{\mathrm{d}V}{V}(22)$$

When the pressure decreases, the cross-sectional area of the channel decreases with the increase of the air velocity, which is called subsonic nozzle; when the pressure increases, the cross-sectional area of the channel increases with the decrease of the air velocity, which is called subsonic diffuser[4]. This phenomenon is similar to that of incompressible fluid. When the pressure decreases, the cross-sectional area of the passage increases with the increase of the flow velocity, which is the supersonic nozzle. This is because when the pressure drops, the density of the supersonic fluid decreases sharply and the volume increases rapidly. At this time, the cross-sectional area of the channel must be enlarged to make the rapidly expanding accelerating air flow pass through.

3.3.2. One dimensional steady motion differential equation of fluid

One dimensional steady motion differential equation of compressible fluid with friction and heat insulation:

$$VdV + \frac{dp}{\rho} + \lambda\frac{V^2}{2}\frac{dx}{d} = 0 \quad (23)$$

dρ/ρ is replaced by continuity equation:

$$\left(\frac{V^2}{k} - \frac{p}{\rho}\right)\frac{dV}{V} - \frac{p}{\rho}\frac{dA}{A} + \lambda\frac{V^2}{2}\frac{dx}{d} = 0(24)$$

It can be seen from the above formula that the critical section of compressible fluid in the divergent

Published by: The Mattingley Publishing Co., Inc.

July - August 2020
ISSN: 0193-4120 Page No. 6136 - 6143

nozzle is not at the minimum section due to the influence of friction.

4. GRP scheme for compressible inviscid hydrodynamic equations

4.1. GRP scheme for one dimensional relativistic hydrodynamics equations

The numerical solution obtained by GRP scheme is in good agreement with the exact solution, and the rarefaction wave and shock wave are also well resolved. There is a difference between the solutions of the relativistic shock tube problem and the corresponding non relativistic shock tube problem, which is mainly caused by the nonlinear superposition of velocity and Lorentz contraction. In the non-relativistic case, the profile of the sparse wave is close to a straight line, while in relativity, the nonlinear superposition of the velocity results in the curvilinear curve of the sparse wave part.

To solve partial differential equations by difference method is to obtain the approximate valueofon some discrete nodesof one region. The basic idea of the difference method is to mesh the solution area, discretize the partial differential equation on the grid nodes, and derive the difference equation that the approximate value of the exact solution satisfies on the grid node. Finally, the approximate value of the exact solution on the discrete node can be obtained by solving the difference equation, which is usually a linear equation system. Divide g into grid area, as shown in Figure 8, The difference quotient expression of the second partial derivative is obtained

$$\frac{\partial^2 u}{\partial x^2}(i,j) = \frac{1}{\hbar_1^2} + [u(i+1,j) - 2u(i,j) + u(i-1,j)] + 0(\hbar_1^2)(25)$$

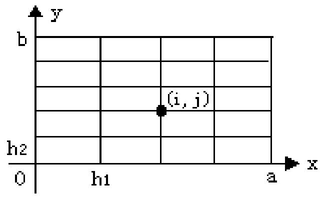


Figure 8. Grid area diagram.

Based on the Riemannian invariants and shock relations of the one-dimensional RHD equations, the cases in which the left and right nonlinear waves are located on both sides of the element interface can be distinguished. In the case of transonic rarefaction wave, the time derivative at the initial discontinuity can be calculated. In the case of only linear wave, the calculation is simple and direct. The GRP scheme for one-dimensional relativistic fluid dynamics equations has high accuracy, so it can be used to solve one-dimensional relativistic fluid dynamics problems effectively.

4.2. GRP scheme for two dimensional relativistic hydrodynamics equations

As a generalization of the second order direct Euler type GRP scheme for one-dimensional RHD in two-dimensional RHD equations, two-dimensional GRP scheme is similar to one-dimensional, but their derivation is different[5]. The Riemann invariants corresponding to nonlinear waves of high dimensional split type RHD equations are nonlinear dependent on the changing tangential velocity. The accuracy and validity of the GRP scheme for two-dimensional relativistic fluid dynamics equations can also be well guaranteed, which lays the foundation for the GRP scheme to be extended to the three-dimensional RHD equations.

4.3. Finite time stabilization of neural networks based on discontinuous excitation function

Based on the solution of the initial value problem of one-dimensional RHD equations, the numerical convergence rate of the third-order GRP scheme for one-dimensional relativistic fluid dynamics equations and its ability to capture shock waves and other discontinuities can be effectively verified. Using uniform grid, WENO reconstruction based on feature decomposition is used in the reconstruction of the third-order GRP scheme[6]. Based on the verification calculation, it is concluded that the third-order scheme not only approximates the exact solution slightly better than the second-order scheme in the sparse wave region, but also has less numerical error in the middle low-density region than the second-order scheme, and the numerical results of the third-order scheme are closer to the

Published by: The Mattingley Publishing Co., Inc.

July - August 2020
ISSN: 0193-4120 Page No. 6136 - 6143

reference solution in the middle density disturbance region.

The construction process of the third-order direct Euler type GRP scheme based on one-dimensional RHD equations is not only more complicated for the nonlinear introduction of the equation, but also the characteristic variables involved in the construction process of the third-order GRP scheme for the one-dimensional relativistic fluid dynamics equations need to be given by numerical integration. The accuracy of the third-order GRP scheme for one-dimensional relativistic fluid dynamics equations is demonstrated based on an example. The results obtained by the third-order GRP scheme are better than those of the second-order scheme in the smooth region, but the resolution is slightly lower than that of the second-order scheme near strong discontinuities.

5. Conclusion

In summary, With the deepening of the influence of relativity on the whole physical system, many interdisciplinary research fields are constantly emerging. Relativistic fluid mechanics is playing an increasingly important role in many related disciplines such as physics. Fluid mechanics based on computer technology can directly solve and operate nonlinear algebraic equations, so as to solve the basic laws of fluid motion. With the support of intelligent technology, fluid mechanics has been applied in many fields, and has become an important means and method to solve practical engineering problems.

In this paper, the finite volume method, FEV and diffusion equation are introduced by analyzing the numerical methods of computational fluid dynamics. Through the study of compressible fluid flow, the one-dimensional flow equation, energy equation and isentropic flow of compressible fluid are analyzed. Finally, the GRP schemes for compressible inviscid hydrodynamics equations are solved, including the GRP scheme for one-dimensional relativistic fluid dynamics equations, the GRP scheme for two-dimensional relativistic fluid dynamics equations and the third-order GRP scheme for one-dimensional relativistic fluid dynamics equations. Numerical calculations are given to compare the numerical errors and applicability of each scheme.

References

[1] J. Q. Li, Z. F. Sun, Remark on the generalized riemann problem method for compressible fluid flows. J. Comput. Phys., 2017, (2):796–808.
[2] E. Han, J. Q. Li, H. Z. Tang, An adaptive GRP scheme for compressible fluid flows. J. Comput. Phys., 2010, 22(5):1448–1466.
[3] P. He, H. Z. Tang, An adaptive moving mesh method for two-dimensional relativistic hydrodynamics. Commun. Comput. Phys., 2012, 11:114–146.
[4] H. Z. Tang, T. G. Liu, A note on the conservative schemes for the Euler equations. J. Comput. Phys., 2006, 218:451–459.
[5] J. Q. Li, Q. B. Li, K. Xu, Comparison of the generalized riemann solver and the gas-kinetic scheme for inviscid compressible flow simulations. J. Comput. Phys., 2011,230(12):5080–5099.
[6] W. Q. Zhang, A. I. Fadyen. A relativistic adaptive mesh refinement hydrodynamics code. Astrophys. J. Suppl., 2016, 164(1):255–279.