Application of Finite Volume Method for Solving Two-dimensional Navier Stokes Equations

Jian Ni, Xifei Wei
College of Information and Electronic Engineering
Hebei University of Engineering
Handan, China
nijianlaoshi@126.com, weixifei123@163.com

Journal of Digital
Information Management

ABSTRACT: *The finite volume method combines the advantage of the finite element method and the finite difference method, while overcoming their shortcomings. This paper describes the development of the finite volume method and the process of using finite volume method for fluid calculations. Finally, this paper introduces the process of using improved finite volume method for solving the flow equations of NS.*

Categories and Subject Descriptors:
G.1.8 [Partial Differential Equations]: Finite Element Methods; **G.1 [Numerical Analysis]**

General Terms:
Numerical Analysis, Finite Volume Method, Finite Element Methods

Keywords: Finite Volume Method, NS Equations, Riemann

Received: 12 May 2013, Revised 28 June 2013, Accepted 1 July 2013

1. Introduction

Finite volume method (FVM) is used to solve two-dimensional Euler equations for the first time in 1971 by the McDonald, and it is used to calculate the constant incompressible flows through the SIMPLE algorithm by Patankar [1] in 1972,and it begins to be used calculated airflow by Jameson in 1977. With the popularity of unstructured grids and energy algorithm, the realization of FVM has greatly improved and enhanced, so that FVM has entered a period of rapid development. Meanwhile, FVM combined with other numerical methods such as finite element method, Upwind format, Runge Kutta method and some new developed methods including MUSCL [2], Roe [3], TVD [4], ENO [5] and other, which results in a series of innovative and effective ways. The finite volume method which calculates the area into a number of discrete rules or irregularly shaped cells is similar to the finite element method. In the flow calculations, first calculating the control volume through the boundary along the normal of each input and output flow, momentum flux, and then balancing calculations separately for each control volume and momentum of the water, and finally getting the end of time period of the control body average water depth and flow velocity.

The finite volume method of generating the discrete equations method is very simple, and can be regarded as a weighted surplus method is finite element equations, and gets the differential equation through the equations to function $\delta W = 1$. But equation of the physical meaning completely different. First, the integral region is associated with a node in the control volume; Second, the physical meaning of integral equation is to control the volume of the flux balance.For example one-dimensional steady-state convection-diffusion equation finite volume method is the starting point of the discrete equations for the following equation:

$$\int_v \frac{d}{dx}(\rho u \varphi)dV = \int_v \frac{d}{dx}(\Gamma \frac{d\varphi}{dx})dV \qquad (1)$$

Appearing on the left side expresses through the control volume V to flow, equality right representation by using the control volume V diffusion quantity. The equation is changed to:

$$\int_v \left[\frac{d}{dx}(\rho u \varphi) - \int_v \frac{d}{dx}(\Gamma \frac{d\varphi}{dx})\right] dV = 0 \qquad (2)$$

In (2) interprets as the sum of the volume of flow and diffusion is zero in the steady state, that is, flux balance.

Therefore, the finite volume method is the derivation of the discrete equations by controlling the volume of the integral equation as a starting point, this and finite difference method is directly from differential equation is totally different. In addition, the finite volume method to obtain the discrete equations, the physical representation on a control volume flux balance equations, each has a clear physical meaning, which is the finite volume method is more competitive than the finite difference method places. From the integral method of selecting the domain seems finite volume method is the weighted residual method in the sub-domain method, the unknown solution from the approximate method seems finite volume method is using a local approximation of the discrete method. In short, subdomain method with discrete finite volume method is the basic way of thinking.

Grid algorithm, boundary treatment for fluid flow simulation is very important. The plane grid, for example, the grid can be divided into structured and unstructured mesh grid. Structured grid is also known as FDM [6, 7] grid, that is difference grid, unstructured grid FEM mesh is also known that the finite element mesh.

2. The finite volume method for computational fluid dynamics

Use the finite volume method for fluid calculation, first ,we must understand domain mesh. Grid finite volume method can be structured, it can be unstructured. In the finite element method and finite volume method, the grid layout of the division and the nodes are independent, the node is not necessarily a lattice vertex. For two-dimensional problem, there are three basic node layout. One is the lattice sub-center type (cell-centered, CC). Another way is to check the vertex-type (cell-vertice, CV). The third way is a hybrid, is to mix up the first two methods. Second, we must define the control volume, it is the mass and momentum balance calculations for the basic unit, corresponding to different nodes in the layout of the control volume chosen as follows: (1) Center for the grid layout node type (CC) mode, the control body to take the grid itself, and do not overlap. (2) For the CV mode, the control body may be defined as three types: control volume is a grid, along the sides of the control volume method to the integral flux, the flux from the two vertices determined by trapezoidal rule; Control volume around a vertex from the lattice by a group composed of some of the control volume covering each other; CV-centric approach to a vertex from the intersection to the vertex centroid of each grid line segment by combining into a control volume . CV mode is mainly used in the present high precision constant gas flow and viscous flow calculations, still less applied to shallow water flow calculations. (3)In hybrid layout, with water depth and velocity components of the node node-centered control volume staggered overlap, its purpose is not easy to achieve in the calculation of pressure conditions. It is mainly used for SIMPLE-based algorithm, the calculation is seldom used in shallow water. Finally, wu must provide the distribution of physical quantities in the control of the body. The finite volume method separates the subdivision of grid and node layout,bringing the geometric flexibility. FVM allows control of the body is inconsistent with the lattice, bringing flexibility to define the discrete flow.

The finite volume method can be the same as the finite element method, the rules applicable to any small grid, and focus on the control body approximation, with the conservation of nature,and like character method which has the characteristic features of wind-based. We can say that the finite volume method reflects the finite element geometric properties, characteristics method and finite difference method the accuracy of the efficiency and nature conservation. The disadvantage is that, the finite volume method is more trouble than on a rectangular grid finite difference method trouble in the irregular grid computing viscous term, and also can not like the finite element that use weak solution and adjoint operator concept so that the two derivative of order reduction. So it is now mainly used for the Euler flow, shallow water and high Reynolds number Navier-Stokes [8, 9] flow.

3. Solving the NS equations

Anastasia [10] who in 1997 proposes a complete conservation of the vector form, as the finite volume method for shallow water wave model problem, the equation is written as:

$$U_t + \Delta . F = 0$$

or

$$U_t + E(U)_x + G(U)_y = H(U) \qquad (3)$$

The equation written in integral form is:

$$\frac{\partial}{\partial t}\iint_{\Omega} U d\,\Omega + \int_{\partial\Omega} F.\,nds = 0 \qquad (4)$$

In (4), $n = (n_x, n_y)$ is the solution of the boundary of the normal vector field, Ω is the solution domain, $\partial\Omega$ is the boundary outward normal vector.We consider (3) For the case of homogeneous equations, and even if (3) of the source term $H(U) = 0$, we have (3) to read as follows in the form of

$$\frac{\partial U}{\partial t} + \nabla F = 0 \qquad (5)$$

In the control unit on the above (5) can be integrated

$$\int_{Vt}\left[\frac{\partial U}{\partial x} + \nabla F\right] dV = 0$$

Supposing U_t is the average of configuration in V_t center, the above equation becomes

$$\frac{\partial \overline{U_t}}{\partial x}\nabla V_t + \oint_{\partial V_t} F.\,nds = 0 \qquad (6)$$

In (6) ∇V_t is the size of V_t and ∂V_t is the boundaries of V_t, $\overline{U} = \frac{1}{|\nabla V_t|}\int_V U dxdy$ is the average of U in the unit, is a boundary ∂V_t outside the unit normal vector, $F = (f, g)$, V are given solution area. Assuming the V is part of the solution for Delaunay type no structure triangle mesh, control volume can take as triangular element, the average per unit of conserved variables configured in the center of the unit.

3.1 Mesh Generation and Space Discretization

Here is the solution region V profile is divided into many small non-overlapping unit $\{V_j, j = 1, K, N\}$, V_j is the polygon Any subdivision within the region for two small units V_j and $V_i\,(j_1 \neq j_2)$, if this two elements intersect, or a total or a total of boundary vertices, the ΔV_j is the control of body volume.

Spatial discretization process includes the following three aspects:

(1) Reconstruction: According to the control volume on the average values of $\overline{U}_t$, obtains $U(x, t)$ in the control volume boundary integral point approximation.

(2) Using appropriate quadrature formula, usually Gaussian formula, to approximate (6) where the control volume boundary integral term, (6) in the dispersion curve of the integral term as follows:

$$\oint_{\partial V_t} F.\,nds = \sum_{j=1}^{3} F_{ij}\,\Delta l_{ij} \qquad (7)$$

In (7) l_{ij} indictes the j-th edge of triangular element V_t, Δl_{ij} represents the length of l_{ij}, F_{ij} is the value of l_{ij} by the side of the stream function.

(3) Using monotone flux (single equation) or the Riemann solver (for equations) constitute the numerical flux F^*, to approximate the physical quantities F_{ij} in (7). In the continuous case, by the Godunox thinking, at the interface can be problem as the direction of the local one-dimensional Riemann problem, therefore, the value stream can be obtained by proximating Riemann solver, the interface on both sides of the function value can be considered different, that jump, and thus calculating the value of the stream function must take into account the conditions of this interruption, that is:

$$F_{ij} = F^*((U_L)_{ij}, (U_R)_{ij}) \qquad (8)$$

F^* is an approximate Riemann solver, $(U_L)_{ij}$ and $(U_R)_{ij}$ are U in l_{ij} counterclockwise left, on the right side of the approximate value, The according to (i) step (that is reconstruction step), obtains U in U_{ij} approximation from the control volume ΔV_{ij}, and $(U_R)_{ij}$ expresses V_j has common borders from neighboring control volume ∂V_{ij} obtained within V_i point U in the same approximation. Choose different Riemann solvers and $(U_L)_{ij}$, $(U_R)_{ij}$ of different reconstruction, we can get a variety of different methods.

3.2 Discrete Time

Solution of ordinary differential equations on time mainly in two ways: first is the Lax-wendroff type of approach, that the Taylor expansion of the time, and then use the time derivative of the original PDE into a spatial derivative, then the discrete derivative of the space, but this multi-dimensional problem is too complex and time-discrete method; the second is to use ODE solver such as Runge-Kutta method, finite volume method which is frequently used time-discrete method, and stability for discontinuous solutions that combine a non-discrete space linear stability.

A second-order Runge-Kutta scheme is:

$$U^{(1)} = U^{(n)} + \Delta t L(U^n)$$

$$U^{n+1} = \frac{U^n}{2} + \frac{1}{2}(U^{(1)} + \Delta t L(U^{(1)}))$$

Another third-order Runge-Kutta scheme is:

$$U^{(1)} = U^{(n)} + \Delta t L(U^n)$$

$$U^{(2)} = \frac{3}{4}U^n + \frac{1}{4}(U^{(1)} + \Delta t L(U^{(1)}))$$

$$U^{n+1} = \frac{1}{3}U^n + \frac{2}{3}(U^{(2)} + \Delta t L(U^{(2)}))$$

L is the space of discrete operator.

3.3 Structure finite volume method format

Here we use a Roe-MUSCL format of the format structure finite volume method, we use Roe Approximate Riemann solution approach in order to calculate $(F.\,n)_j$, we set U_j^+ and U_j^- which are the values of U which plants on both sides of the triangle, then:

$$(F.\,n)_j = \frac{1}{2}[F(U_j^+).\,n_j + F(U_j^-).\,n_j - |A|(U_j^+ - U_j^-)] \quad (9)$$

Here A is the numerical flux Jacobian matrix, A is defined as follows:

$$A = \frac{\partial(F.\,n)}{\partial U} = \begin{bmatrix} 0 & n_x & n_y \\ (c^2 - u^2)n_x - uvn_y & 2un_x - vn_y & un_y \\ (c^2 - u^2)n_y - uvn_x & vn_x & un_x + 2vn_y \end{bmatrix}$$

The specific expression of F and n_j into $F(U)\,.\,n$ can be calculated following the formula:

$$F(U)\,.\,n = E\,.\,n_x + G\,.\,n_y = \begin{bmatrix} uh\,.\,n_x \\ (u^2h + (\frac{1}{2})gh^2)\,.\,n_x + uvh\,.\,n_y \\ uvh\,.\,n_x + (v^2h + (\frac{1}{2})gh^2)\,.\,n_y \end{bmatrix}$$

Therefore, the following problem as long as the calculation of $|A|(U_j^+ - U_j^-)]$ and the estimated $U_j^+, U_j^-, |A|(U_j^+ - U_j^-)]$ are the last to be:

$$|A|(U_j^+ - U_j^-) = \alpha_1\,.\,un_x + vn_y + c.\begin{bmatrix} 1 \\ u + cn_x \\ v + cn_y \end{bmatrix} + \alpha_2$$

$$\alpha_3 \cdot |un_x + vn_y - c| \cdot \begin{bmatrix} 1 \\ u - cn_x \\ v - cn_y \end{bmatrix} \qquad (10)$$

In (10):

$$\alpha_1 = \frac{1}{2}\Delta h + \frac{1}{2c_u}[\Delta(uh)\, n_x + \Delta(vh)\, n_y - (u_a n_x + v_a n_y)\,\Delta h]$$

$$\alpha_2 = \frac{1}{c_u}\,[(\Delta(vh) - v_a \Delta h)\, n_x - (\Delta(uh) - u_a \Delta h)\, n_y]$$

$$\alpha_3 = \frac{1}{2}\Delta h - \frac{1}{2c_u}[\Delta(uh)\, n_x + \Delta(vh)\, n_y - (u_a n_x + u_a n_y)\,\Delta h]$$

For the U_j^+, U_j^-, because they are the practice of first-order accuracy, it is very simple, we directly adjacent to the current edge Value as the center of the triangle U_j^+, the current U-value as the center of the triangle U_j^-.

4. Boundary conditions

Actually shallow water flow problems have boundary exists, which belongs to mixed initial boundary value problem, so we must give early boundary conditions when we solving it. Definite solution of the equation of the problem need to give some initial and boundary conditions due to the error of the initial conditions time will soon decay, so that people given the initial conditions do not have to be precise.

4.1 The general principles of the boundary treatment

Boundary conditions can be divided into two categories: First, setting the boundary point part or all the value of the dependent variable, or the relationship between the dependent variable to reflect the impact of the computational domain Ministry, which is referred to as the physical boundary conditions (often referred to as the boundary conditions); Second, in order to determine the additional mathematical conditions for the remainder of the boundary points given dependent variable, reflecting the impact of computational domain, which is referred to as the numerical boundary conditions.

The provisions of the physical and numerical boundary conditions have a certain degree of arbitrariness. Former electes from the measured data to be a part of the provisions. The same point format within the select the latter, as there is considerable freedom. Typically, the inner point of variables uses conservation variables, the physical boundary conditions uses the original variables and the numerical boundary condition uses these two variables or characteristics.

The basic requirements of the boundary treatment is:

(1) The computational problems posed mathematically and physically reasonable.

(2) Minimal impact on accuracy and stability of numerical solution of the interior point. We mainly study one-dimensional boundary with the exception of a few special requirements on the two-dimensional boundary. Explicit calculation, the boundary value only spread to the adjacent grid at each time step, implicit calculation is more sensitive to boundary treatment, related to the algorithm (to keep the equations easy to accurately solving), and the impact each time spread to the global step.

4.2 Moving boundary treatment

Moving boundary is the level of the computational domain boundaries of the water and the anhydrous region, which is different from the free surface between the water vapor in the causes, mechanisms and treatment. This is also the water unique air. One-dimensional examples have the evolution of dam-break flood on the dry riverbed and anhydrous nullah water-filled; two-dimensional examples have the accumulator water, estuaries and coastline of the slot and burst flooding of floodplains, lakes and reservoirs in the astronomical tide and storm surges, advance and retreat. The relocation of the land and sea borders is due to the inside of the water level is higher than the outside ground, and shrink due to the inside of the water level is lower than the same side of the ground. Water depth near the border are usually small, and the boundary exists to the flow rate, which is different from the land border.

Moving boundary numerical simulation of the difficulties include:

(l) Uniform flow along the moving boundary method is different from the open channel flow, unsteady and non-uniformity, commonly used Manning Mo resistance formula in the form it is difficult to apply (water depth tends to zero friction unlimited increase , rough great change). Even the water does not meet the assumption of shallow water flow, peak moving speed is difficult to determine.

(2) The water depth is very small, demanding discrete format, the numerical solution does not produce false vibration. To ensure that water depth is always greater than zero.

(3) Moving boundary as the calculation of the singular point, and point calculation to match. There are two types of processing techniques in the application: First, to track the exact location of the moving boundary; is only concerned with moving boundary where the grid.

Tracking the moving boundary we can divide into two kinds of cases. Such as the inside of the ground below the outside ground, at the beginning of the calculation period, the moving boundary can be regarded as the solid wall, and after calculating the water level at the end of the hours we can through the water level in epitaxial or use of the water balance equation to determine the moving boundary position. On the contrary it may be assumed near the depth of water before the peak for a small value, and its calculation of flow rate (assumed to be uniform flow) as the relocation of the peak line speed. In both cases, the moving boundary as the boundary of the computational domain there depth is set to zero or a small value. The grid is divided into dry, submerged and semi-flooded three categories. Semi-submerged grid can be further partitioned into dry and flooded two parts, the inside of the moving boundary lattice of local deformation. Or adaptability encryption nearby grid. In addition, a simple track to handle only a fixed grid, moving the boundary to coincide with the grid lines, the calculation results according to each time period to determine the boundary move as leap.

If you do not track the moving boundary, the put may be the largest regional fixed computational domain, regardless of water are the same anhydrous calculated. Commonly used two treatments:

(1) In the one-dimensional treatment assumes that all grid there is a narrow deep groove can be calculated by one-dimensional compound channel flow. Groove deep enough to always have water, but also due to the slit very narrow, the resulting water conservation error is very small, and can allow.

(2) Assuming that all the lattice at least there is a very small depth of water (such as l mm, may be regarded as soil moisture, water conservation and the resulting error can also be allowed) to other lattice the same calculation. Of course, if a grid and all its adjacent grid equal the water depth in this lower limit can be regarded as the land and do not have to calculate. Where the lattice of the moving boundary must be adjacent to a land grid do not have to determine its specific location. The numerical solution of the lattice average of the meaning of this singular point of the moving boundary integral processing and disappeared.

Whether or not to track the moving boundary, usually take a number of processing techniques, including point:

(1) To adjust the roughness the water depth decreases when the roughness is usually increased, there are many observation experiments. Numerical simulation of reference, but not necessarily strictly related data. Reference oceanographic research, the processing of numerical tests using depth less than l m, the roughness increases in multiples of equal depth or the reciprocal of the square. Roughness increase helps prevent calculate the flow rate soared, and thus is conducive to stability, but when the water depth very hour to prevent too much friction, as well as cause the flow rate of the reverse and instability.

(2) Point format to prevent the numerical solution of false vibration. Early simple processing techniques such as: omitted to control the inertia in the equation and (or) The slope of the friction term in order to avoid nonlinear instability, of course, will bring the error (such as dam-break flood peak movement by inertia effects large). Near the moving boundary to take full upwind scheme. Limit the calculation of water depth greater than a small value, no calculation of flow velocity changes its sign in the case of unidirectional flow, the provisions of the flow rate at a depth of less than a small value is zero. The best treatment is monotonous format, you can avoid the small depth due to false vibration becomes negative. High-performance format is not only conducive to dealing with intermittent, but also helps to deal with the small water depth and moving boundary problems.

When calculating the flux of each triangular element to control the inflow and outflow of the body, we need to calculate the state of three adjacent triangular elements with this triangular element. In the triangular control volume in the calculation of boundary flux due to the triangular control volume adjacent to the border on the edge, so we want to set the boundary conditions of the equation. The boundary conditions are often in the following ways:

4.1.1 Land boundary

The boundary conditions of land is known as a closed boundary and uses no-slip conditions, and thinks that the water depth in the boundary normal direction does not change and the normal direction of flow velocities in the boundary derivative is zero. It need to meet the following conditions:

$$U^+ = -U^-, \; h^+ = h^-$$

We simply use slip boundary condition of no penetration.

4.1.2 Open boundary

Usually intercepting part water forms the field, and determines as the boundary of the upstream and downstream is for open border. Open boundary is generally divided into the open boundary of the subcritical flow and supercritical flow. It is a boundary Riemann problem to be solved when the unit side of the computational domain boundaries or physical boundaries consistent boundary for U^+ already know, and U^- is unknown, we must, according to the local fluid state type and determined the physical boundary conditions type to select the desired characteristics to solve it. In order to simulate the flow of constant inflow and outflow, this boundary can also be used to constant boundary conditions, h, u and v is taken as the initial value.

5. Conclusions

The finite volume method can be the same as the finite element method, the rules applicable to any small grid, and focus on the control body approximation, with the conservation of nature, and like character method which has the characteristic features of wind-based. We can say that the finite volume method reflects the finite element geometric properties, characteristics method and finite difference method the accuracy of the efficiency and nature conservation. The disadvantage is that, the finite volume method is more trouble than on a rectangular grid finite difference method trouble in the irregular grid computing viscous term, and also can not like the finite element that use weak solution and adjoint operator concept so that the two derivative of order reduction. So it is now mainly used for the Euler flow, shallow water and high Reynolds number Navier-Stokes flow.

6. Acknowledgment

The authors wish to thank the part of Handan Natural Science Foundation. The authors are highly thankful.

References

[1] Patankar, S. V. (2006). Heat transfer and fluid flow numerical methods.

[2] Van Leer, B. (1979). Towards the ultimate conservative difference schemes V. Asecond- order sequel to Godunov's method. *Journal of Comput. Phys.*, (32) 101-136.

[3] Roe, P. L. (1981). Approxiamate Riemann solvers, parameter vectors,and difference schemes. J*ournal of Comput. Phys*, (43) 357-372.

[4] Harten, A. (1983). A high resolution scheme for the computation of weak solutions of hyperbolic conservation laws. *Journal of Comput. Phys.*, (49) 357-393.

[5] Harten, A., Engquist, B.,Osher, A., Chakravarthy S. (1987).Uniformliy high order essentially non-oscillatory schemes. *Journal Comput.Phys.*, (71) 231-303.

[6] Alfredo Bermudez. (1998). Upwind schemes for the two-dimensional shallow water equations with variable depth usingunstructured meshes. *Comput Methods Appl. Mech. Engrg*.

[7] Chen, Y., Falconer, R. A. (1994). Modified forms of the third-order convection second-order diffusion scheme for the advection-diffusion equation. *Advances in Water Resources*, (17) 147-170

[8] Weiyan Tan. (1998). Calculation of shallow water dynamics eleven application of the finite volume method. Tsinghua University Press.

[9] Mengping Zhang. (1998). Numerical methods for shallow water wave problem research and application of [D]. PhD thesis University of Science and Technology of China.

[10] Anastasion, K.,Chan, C. T. (1997). Solution of the 2D shallow water equations using the finite volume method on unstructured triangular meshes. *International Journal for Number Methods in Fluids*, (24) 1225-1245.