Research on 3D Hydrodynamic Model Based on Adaptive Mesh

Ming Wang\(^1,2\)*, Huili Dong\(^3\) and Fangxiu Zhang\(^1,2\)

\(^1\) Yellow River Institute of Hydraulic Research, Zhengzhou 450003, China
\(^2\) Key Laboratory of Yellow River Sediment Research of MWR, Zhengzhou 450003, China
\(^3\) Henan Technical College of Construction, Zhengzhou 450064, China

Email: wangming198261@163.com

Abstract. In this paper, a three-dimensional hydrodynamic model based on adaptive mesh was established. The cubic mesh based on octree structure was adopted to solve the three-dimensional Navier-Stokes equation by the step-by-step pressure decomposition method, which realized the automatically encryption and coarsening of the grid. The calculation of three-dimensional square cavity flow showed that the calculation results of this model are in perfect agreement with those of Chen, and the calculation accuracy is high. Through the calculation of three-dimensional flow around a cylinder, the model can automatically encrypt the grid near the complex boundary, and can refine and coarsen the grid reasonably through the vorticity field. Meanwhile, it can also reduce the calculation amount and improve the calculation speed while maintaining the calculation accuracy.

Keywords. Adaptive mesh, distributed projection operator method, Poisson equation, complex boundary.

1. Introduction

Currently, in the calculation of 3D unsteady flow with complex boundary, the body fitted coordinates [1-2] and unstructured meshes [3] are preferred for the property of fitting the calculation boundary well. However, their mesh generation is relatively difficult and takes more time. Since the flow pattern of unsteady flow varies with time and the layout density of grid is closely related to the flow pattern, the local grid will be too dense or too thin when the grid cannot be adjusted accordingly with the variation of flow pattern. Too dense grid leads to a waste of the calculation resources, and too thin grid leads to calculation accuracy affected. Adaptive mesh can automatically control the mesh density according to the variation of computational variables with time without needing mesh manually completely, so it will save working time to a great extent.

2. Numerical Method

2.1. Control Equation

In this model, the incompressible continuity equation and Navier-Stokes equation are expressed as follows:
\[\nabla \cdot \mathbf{u} = 0 \]
\[
\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla \left( \frac{p}{\rho} + gz \right) + \nu \nabla^2 \mathbf{u} \tag{1}\]

where, \( \mathbf{u}, p, \rho, g, \) and \( \nu = \mu / \rho \) are the velocity, pressure, density, gravity acceleration and kinematic viscosity coefficient of the fluid, respectively, and \( \mu \) is the dynamic viscosity coefficient of the fluid.

2.2. Discrete Solution

The time discretization adopts the split step projection method [4-5], and the temporary velocity field \( \mathbf{u}^{**} \) can be obtained from the following equation:

\[
\frac{\mathbf{u}^{**} - \mathbf{u}^n}{\Delta t} = \left[ - (\mathbf{u} \cdot \nabla) \mathbf{u} + \nu \nabla^2 \mathbf{u} \right]^{n+1/2} \tag{2}\]

Then the new velocity field \( \mathbf{u}^{n+1} \) and fractional step pressure field \( p^{n+1/2} \) are obtained by using projection operator \( \mathbf{u}^{**} \). This projection method relies on Hodge decomposition of velocity

\[
\mathbf{u}^{**} = \mathbf{u} + \nabla \phi \tag{3}\]

Here, in the calculation area \( W \), \( \mathbf{u} \) can satisfy \( \nabla \cdot \mathbf{u} = 0 \), and on the boundary, it can satisfy \( \mathbf{u} \cdot \mathbf{n} = 0 \). Calculate the divergence on both sides of the above equation, and the Poisson equation can be obtained and expressed as follows.

\[
\nabla^2 \phi = \nabla \cdot \mathbf{u}^{**} \tag{4}\]

It is defined on the boundary as:

\[
\frac{\partial \phi}{\partial n} = \mathbf{u}^{**} \cdot \mathbf{n}, \quad (x, y) \in \partial \Omega \tag{5}\]

Thus, the velocity field can be defined as:

\[
\mathbf{u} = \mathbf{u}^{**} - \nabla \phi \tag{6}\]

where \( f \) can be obtained by solving Poisson equation.

2.3. Introduction to Grid Hierarchy

The discretization process of the computational region is to discretize the finite volume space of the cube into octree [6-7]. The basic principle of hierarchical grid framework is illustrated by taking two-dimensional spatial discretization as an example, and the corresponding tree structure of two-dimensional spatial discretization is quad-tree structure, as shown in Figure 1. The computational region is generally composed of one or more root cells, which are subdivided recursively from the root cell to generate children cells of different generations, and each cell can be equally divided into four subunits. The root cell is the basis of tree structure, while leaf cell is the cell without sub unit. The definition of element layer is as follows: Assume the root element is the zero layer, and the number of layers will increase by one when adding a group of sub units.
Figure 1. Quad-tree discretization and its corresponding tree structure diagram.

3. Calculation Case

3.1. Three Dimensional Square Cavity Flow
The calculation area of the three-dimensional square cavity flow is 1 m × 1 m × 1 m with Reynolds number Re=100, and boundary conditions for calculation is: when z=0, u=1m/s, v=0, and w=0; x=0, x=1 m, y=0, y=1 m, and z=1 m all have no slip boundary conditions with u=0, v=0, and w=0.

The distribution of velocity u along the z direction at x=0.45 m and y=0.5 m is shown in figure 2. The results indicated that the calculation results of this model are in perfect agreement with those of Chen [8], which proved that the model proposed in this paper has high calculation accuracy.

As can be seen from figure 3, the calculation grid will adjust constantly with the variation of flow field. Under the initial condition, there is only one grid in the whole calculation area. With the development of the calculation, the velocity of the flow develops from the bottom to the top. The grid is densified at the larger vorticity and near the solid wall boundary along with the development of the velocity field.
The 7th International Conference on Environmental Science and Civil Engineering
IOP Conf. Series: Earth and Environmental Science 719 (2021) 042077
doi:10.1088/1755-1315/719/4/042077

(a) t=0 s

(b) t=0.4 s

(c) t=0.8 s

(d) t=8.0 s

Figure 3. Variation of grid and velocity field with time.

The velocity field of various sections after the calculation becomes stable is shown in figure 4. The results indicated that the velocity in the middle section is larger than that in the two sides due to the influence of the side wall.

Figure 4. Velocity field of different profiles.

3.2. Three-Dimensional Flow Around a Cylinder
The calculation area is 4 m × 1 m × 1 m water tank, the radius of the cylinder is 0.062 m, and the axis of the cylinder is 1.0 m away from the water inlet and horizontal centered. The Reynolds number (\( Re_d = uD/n \)) based on the diameter and inflow velocity is 100. On the left side of the calculation area is inflow boundary conditions, with velocity of 1 m/s, and the right side is continuous outflow condition. The bottom and left and right-side walls adopt the non-slip boundary conditions, and the top adopts the slip boundary conditions.

As can be seen from figure 5, the grid around the cylinder is sparse at the initial time, and the grid layer is 6. At \( t=0.1 \) s, the grid is very dense with layer of 8. The model can automatically encrypt the grid near the complex boundary, and the speed of encryption is very fast. The number of grid layers is only increased by 2, while the mesh density has already been 16 times that of the original.
It can be seen from figure 6 that there is a perfect correspondence between the vorticity field and the grid density. The grid density is high at the place with large vorticity, and small at the place with small vorticity. Before the vortex street behind the cylinder is formed, the grid behind the cylinder is very sparse. After the vortex street is formed, all vortices are covered by the grid with higher density all the time as the vortex moves downward. The density is relatively small in the area outside the vortex, which shows that the model can automatically refine the grid at the large vorticity and coarsen the grid at the small vorticity.

Figure 6. Variation of surface grid and vorticity field as time changes.

4. Conclusion
In this paper, a three-dimensional hydrodynamic model based on adaptive mesh is established and the three-dimensional Navier Stokes equations are solved by the step-by-step projection method. The calculation of three-dimensional square cavity flow showed that the calculation results of this model are in perfect agreement with those of Chen, and the calculation accuracy is high. Through the calculation of three-dimensional flow around a cylinder, the model can automatically encrypt the grid near the complex boundary, and can refine and coarsen the grid reasonably through the vorticity field, meanwhile, it can also reduce the calculation amount and improve the calculation speed while maintaining the calculation accuracy.
Acknowledgments
This study is supported by the National key basic research and development plan (2017YFC0405505); Funding was also provided by the NSFC funded project (42041006, 51679102).

Reference
[1] Fang H W and Wang G Q 2000 Three-dimensional mathematical model of suspended-sediment transport Journal of Hydraulic Engineering 126(8) 578-592.
[2] Xia Y F 2002 Research and Application of Three Dimensional Numerical Model of Flow and Sediment in Tidal River Hehai University.
[3] Wang G and Ye Z Y 2003 3D unstructured hybrid mesh generation and N-S equation solution Journal of Aeronautics 24(5) 385-391.
[4] Domingo P and Benazzouz T 2017 Direct numerical simulation and modeling of a nonequilibrium turbulent plasma Aiaa Journal 38(1) 73-78.
[5] Deb P, Kiem A S and Willgoose G 2019 A linked surface water-groundwater modelling approach to more realistically simulate rainfall-runoff non-stationarity in semi-arid regions Journal of Hydrology 575 273-291.
[6] Zhang Y, Horowitz H, Wang J, Xie Z, Kuss J and Soerensen A L 2019 A coupled global atmosphere-ocean model for air-sea exchange of mercury: insights into wet deposition and atmospheric redox chemistry Environmental Science & Technology 53(9) 5052-5061.
[7] Phaksopa J and Sojisuporn P 2020 Storm surge in the gulf of Thailand generated by typhoon Linda in 1997 using Princeton ocean model (POM) Kasetsart Journal-Natural Science 40(5) 260-268.
[8] Chen C J, Ho K S and Cheng W C 1982 The Finite Analytic Method Iowa (Iowa: The University of Iowa).