Discretization Analysis of Fluid Mechanical Flow Field Grid

Yu Sheng*

Science and Technology on Particle Transport and Separation Laboratory, Tianjin 300180, China

*urinspiration@163.com

Abstract. The design basis of the performance of special industrial fluid machinery is to obtain the distribution of air flow parameters through the solution of flow field. In the numerical solution of flow field, the discretization of computational domain grid is the key factor affecting the calculation results. In this paper, the two-dimensional axisymmetric flow field analysis model were adopted. By establishing the software interface and automatic execution script file, the grid discreteness analysis process was established. The Richardson extrapolation method was used to infer the accurate solution of the flow field, and the grid independence and convergence analysis and the grid spacing analysis of the first layer of the wall were carried out. The results showed that the numerical results gradually approached the exact solution with the increase of the total number of grids, but the correlation between the numerical results and the total number of grids decreased in this process; By changing the normal spacing of the first layer of grid on each wall and making mathematical statistical analysis, the sensitive area of grid spacing could be determined. The method established in this paper could significantly improve the overall efficiency of the fluid mechanical mesh discreteness analysis and flow field analysis, and recommended the configuration of the optimal mesh discretization mode for the subsequent simulation calculation, which was conducive to further improve the calculation accuracy and efficiency of the flow field.

Keywords: Flow field; Numerical calculation; Mesh discreteness; Calculation accuracy; Computational efficiency

1. Introduction

With the progress of computer technology, CFD numerical simulation software has become an indispensable tool for flow field analysis and performance design of fluid machinery in special industrial field. Among them, the reasonable design and high-quality generation of computational domain grid is the prerequisite of CFD calculation and one of the most important decisive factors affecting CFD calculation results.

In the fluid machinery under a working state, the rotor speed is high, and there are slits inside, and the flow field distribution is extremely complex, which makes it difficult to choose the grid size. On the one hand, if the grid requirements of minimum space are met, the number of overall grids will increase sharply; On the other hand, if the mesh size is increased, the mesh quality will be reduced, and the local mesh is too sparse, which will reduce the accuracy of the calculation results. Therefore, in order to improve the credibility of CFD simulation results and ensure high computational efficiency, fine and reasonable meshing for the computational domain is an essential part of CFD work.

In the field of CFD, researchers carried out a lot of research in grid discreteness analysis. Freitas [1] pointed out that the convergence of the grid is that the convergence solution of numerical calculation
should be independent of the number of grids. Roaches [2] proposed the concept of grid convergence index and established the relationship between grid convergence index and grid convergence error, convergence accuracy and safety factor. Wang Yongda [3] believed that the quality of computational grid is the key factor affecting the results of hypersonic aerodynamic thermal numerical simulation, which is reflected in two aspects: the object surface grid and the grid distribution in the boundary layer. Kang Shun et al. [4] found through a two-dimensional example that Richard error estimation method can be used to correct the grid error, but the accuracy of the correction result is closely related to the number of grids used in the calculation.

In this paper, the two-dimensional axisymmetric flow field of fluid machinery was adopted as the research object. The software interface and automatic execution script file were built, the mesh discreteness analysis process was established, and the mesh discreteness analysis of flow field was carried out. In order to further improve the accuracy of flow field calculation, the optimal grid discrete configuration was found.

2. Physical problem description

2.1 CFD simulation error

CFD research includes the process of model establishment and calculation method selection. The model includes physical model, grid discretization in computational domain, partial differential equations and initial boundary value conditions characterizing the conservation of mass, momentum and energy. The calculation methods include finite difference method, finite volume method, time marching method, TVD scheme, high-order scheme and so on. It can be said that the flow field presentation process based on CFD is an effective way to characterize and describe the flow problems in the real world, but uncertainty and errors will be introduced in all links of its implementation[5]. Some errors are inevitable, such as physical model error and machine rounding error, while some errors can be avoided, such as coding error and use error. Therefore, identifying and quantifying those avoidable errors and effectively improving the model accuracy is an extremely important content in CFD research, which needs to be paid enough attention.

2.2 Grid Discreteness

In CFD research, grid discretization research is an effective means to verify numerical computation, which is also the focus of this paper. Based on the characteristics of large span of component structure and high rotor speed in fluid machinery in special industrial fields, the meshing method has a significant influence on the flow field calculation results. In this case, how to determine the grid scale to minimize the grid discretion error and thus improve the reliability of grid partition is the main content of this paper.

Richardson's extrapolation [6] assumes that the discrete solution of an equation is equal to the sum of its exact solution and the power series of the discrete interval. This method can be used to estimate the discrete errors caused by different grids. First, the convergence of several grid results with different grid numbers needs to be analyzed. If only two different grids are calculated, the degree of influence of the grids on the calculation results can be seen, but it is not sufficient for astringency judgment. For convergence analysis, at least three different grids must be calculated. When using this method for analysis, it is required that the simulated values of different grid densities converge monotonously, i.e. with the change of the grid, the simulated values change uniformly.

2.3 Model and Grid Dividing

A two-dimensional axisymmetric model for the calculation of fluid-mechanical flow field is shown in Figure 1. In the equipment of a working state, the fluid flows from the inlet to the area between the inner wall and the outer wall. Under the action of gravity and driven by the rotating inner wall, a top-down swirl flow is formed and then flows from the outlet to the inner cylinder. The flow field space studied in this paper is a slit area between the inner and outer wall.
The grid partition method for the flow field area is shown in Figure 2. The flow field was divided into 7 grid blocks, each of which was spatially discretized by Cartesian grid. Aiming at the three position grids a, b and c, the influence of spacing reduction on the vorticity was investigated. In addition, the influence of normal spacing of the first grid on the vorticity was investigated by changing the dimensions of the first grid on the inner wall and the outer wall respectively.

Figure 2. Partition of the flow field

3. Flow field calculation method

3.1 Hydrodynamic equations

In fluid machinery, fluid flow follows the continuity equation and momentum equations in two-dimensional cylindrical coordinate system. As shown in equations (1) to (4).

Continuity equation: \[
\frac{1}{r} \frac{\partial (r \rho v_r)}{\partial r} + \frac{\partial (\rho v_z)}{\partial z} = 0
\]  

Momentum equations:

\[
\rho \left( v_r \frac{\partial v_r}{\partial r} + v_z \frac{\partial v_r}{\partial z} - \frac{v_z^2}{r} \right) = - \frac{\partial p}{\partial r} + \mu \left( \nabla^2 v_r \right)
\]

\[
\rho \left( v_r \frac{\partial v_z}{\partial r} + v_z \frac{\partial v_z}{\partial z} - \frac{v_r^2}{r} \right) = - \frac{\partial p}{\partial z} + \mu \left( \nabla^2 - \frac{1}{r^2} \right) v_r
\]

\[
\rho \left( v_r \frac{\partial v_\theta}{\partial r} + v_z \frac{\partial v_\theta}{\partial z} + \frac{v_r v_\theta}{r} \right) = \mu \left( \nabla^2 - \frac{1}{r^2} \right) v_\theta
\]

Where: \( r \) - radial coordinate, m; \( z \) - axial coordinate, m; \( \rho \) - density, kg/m\(^3\); \( p \) - pressure, Pa; \( \mu \) - power viscosity coefficient, kg/m\( \cdot \)s; \( v_r \) - radial velocity of gas, m/s; \( v_\theta \) - angular velocity of gas, m/s; \( v_z \) - axial velocity of gas, m/s.

3.2 Initial and boundary value conditions

In this paper, the problem was solved in a steady way without defining the initial conditions.

Boundary conditions were defined as follows:

The top, bottom and the outer wall were defined as static wall surfaces; The drainage plate and inner wall were defined as rotating wall; There was no relative sliding between the fluid and the wall;
Inlet was defined as flow inlet; Outlet was defined as pressure outlet; The axis was defined as the axis of symmetry; All temperatures were defined as constant temperature conditions.

3.3 Numerical calculation method
The pressure-based solver and steady-state calculation method were used in the numerical calculation of flow field, and the calculation model was laminar flow model. When the finite volume method was used to discretize the mass conservation equation and momentum conservation equation, the coupled algorithm was used for pressure velocity coupling, and the second-order discrete scheme was used for pressure and momentum. The appropriate sub relaxation factor was selected for iterative calculation.

4. Process construction
In this paper, the grid discreteness analysis process was established, as shown in Figure 3. In the grid parameterized batch generation module, automatic example generation module and automatic example upload module, automatic execution script files were established to associate and map relevant parameters, so as to realize batch modification of parameters, file generation, automatic iterative calculation and reading of calculation results. Different modules were overlapped and connected through software interfaces. Using this process, the automatic operation process from parameter setting, case generation, parallel example calculation to result reading was completed.

![Figure 3. Analysis process of grid discreteness](image)

5. Result Analysis

5.1 Grid independence and grid convergence analysis
By changing the number of nodes at flow field a, b and c, five groups of grids with different numbers were formed. The grid discreteness analysis was carried out according to the grid setting in Table 1. Figure 4 and figure 5 show the analysis results of grid independence and grid convergence respectively. As seen from Figure 4, with the increase of the total number of grids, the vorticity value of the whole field decreases continuously. At the same time, when the total number of grids increases to a certain number, the variation range of vorticity value decreases gradually. As seen from Fig. 5, with the decrease of the average grid spacing, the vorticity value of the whole field gradually approaches a fixed value, that is, the inferred exact solution when the grid spacing approaches 0 inferred on the basis of the five groups of grid numerical calculation results. It shows that with the decrease of discrete interval, the discrete error is also decreasing, so that the numerical solution of the equation gradually approaches the real solution of the differential equation. Under the dual requirements of calculation accuracy and calculation efficiency, there was an optimal configuration of the total number of grids. As shown in the Fig.4 and Fig.5, it is quite appropriate that the total number of grids is between 75000 to 80000. Under this condition, the numerical solution error is only about 1.5%.

| Table 1. Numbers of nodes of grids |
|------------------------------------|
| Mesh 1  | Mesh 2  | Mesh 3  | Mesh 4  | Mesh 5  |
| a        | b        | c        | a        | b        | c        |
| 80       | 40       | 15       | 90       | 45       | 20       |
| 100      | 50       | 25       | 110      | 55       | 30       |
| 120      | 60       | 35       |          |          |          |
5.2 Analysis of grid spacing of the first layer of wall

Based on the grid script file, the grid sizes of the first layer of inner wall and outer wall were changed to form five sets of grids respectively, and the influence of the normal spacing of the first layer of wall grid on the whole field vorticity was investigated. The obtained results realize mathematical statistical analysis, give the upper and lower bounds, mean and median, as shown in Fig. 6 and Fig. 7, and give the variance and standard deviation, as shown in Table 2. As seen from Fig. 6 and Fig. 7, if the normal spacing of the first layer of grid on the inner wall and outer wall is changed within a certain range, the vorticity values in the whole field are between 10.50 and 10.57, and there is no significant difference; By comparing the data in Table 2, it can be found that the normal spacing of the first layer of grid on the inner wall is more sensitive to the vorticity than that on the outer wall. This is because the strong shear force between the rotating inner wall and the fluid makes the boundary layer flow complex. Therefore, the grid spacing of the inner wall is a key concern in the grid configuration.

Table 2. Vorticity fluctuation corresponding to grid spacing variations

| Change of grid spacing of the first layer of inner wall | Change of grid spacing of the first layer of outer wall |
|--------------------------------------------------------|-------------------------------------------------------|
| Variance of vorticity                                  | 7.2315 × 10^{-4}                                       |
| Standard deviation of vorticity                        | 2.6891 × 10^{-2}                                       |
|                                                       | 6.7534 × 10^{-4}                                       |
|                                                       | 2.5987 × 10^{-2}                                       |

6. Conclusion

(1) In this paper, a grid discreteness analysis process was built for special industrial fluid machinery, which realized the automatic operation process from parameter setting, case generation, parallel example calculation to result reading, and improves the efficiency of grid discreteness analysis.
(2) In this kind of fluid mechanical flow field, with the increase of the total number of grids, the vorticity gradually converges to the exact solution of the differential equation. Compared with the outer wall, the normal spacing of the first layer of grid on the inner wall is a more sensitive factor for the output of the flow field, which should be paid more attention to.

(3) Through the analysis of grid independence, convergence and the first layer grid spacing on the wall, this paper provides the direction of grid discrete configuration for the subsequent simulation calculation of fluid mechanical flow field, which is helpful to further improve the calculation accuracy and efficiency of flow field.

References
[1] Freitas C J. The issue of numerical uncertainty[J].Applied Mathematical Modeling, 2002,26(2):237-248.
[2] Roache P J. Error bars for CFD[C]. 41st Aerospace Sciences Meeting and Exhibit. AIAA.2003:48.
[3] Wang Yongda. Study on grid independence of hypersonic aerodynamic thermal numerical simulation of typical shape [D]. Sichuan: Southwest University of science and technology, April 2020.
[4] Kang Shun, Shi Lei, Dai Liping, et al. Error analysis and grid convergence of CFD simulation, Academic conference of Chinese society of Engineering Thermophysics.
[5] Deng Xiaogang, Zong Wengang, Zhang laiping. Verification and validation in computational fluid dynamics [J]. Progress in mechanics, 2007, (37): 10-15.
[6] Shi Lei, Kang Shun, Guo ran, Grid independence of CFD simulation of transonic rotor. Academic conference paper of Chinese society of Engineering Thermophysics.