ON THE RESEARCH OF FLOW AROUND OBSTACLE USING THE VISCOUS CARTESIAN GRID TECHNIQUE

by

Yan-Hua LIU [*] ***and Hao GU***

College of Mechanical and Electrical Engineering,
Hohai University, Changzhou, China

Short paper
DOI: 10.2298/TSCI1205488L

A new 2-D viscous Cartesian grid is proposed in current research. It is a combination of the existent body-fitted grid and Cartesian grid technology. On the interface of the two different type of grid, a fined triangular mesh is used to connect the two grids. Tests with flow around the cylinder and aerofoil NACA0012 show that the proposed scheme is easy for implement with high accuracy.

Key words: *viscous Cartesian grid, multigrid, hybrid grid*

Introduction

The multiphase flow due to the cavitation in hydraulic machinery is one of the most important directions for the study of off-design running status of hydraulic component. The numerical simulation method is recently used to prove the cavitation model and predict the beginning position of the cavitation. The cavitation in hydraulic machinery is closely related to the pressure and velocity distribution. When the cavitation occurs, the whole system often runs in the off-design point and the tilt angle of running blade is relatively big. At this moment, in spite of an accurate theoretical model of cavitation, a computational mesh with good orthogonality, reasonable distribution and certain density also plays a crucial role in precise prediction of cavitation. When the cavitation occurs, the structural and flow characteristic determines that: a viscous boundary layer should be contained, and during the calculation, the local mesh should be dynamically. Hence, it is a key point to automatically create the viscous mesh around the physical boundary and dispose the remained mesh in the flowpass. Basically, there are two types of meshing method, *i. e.* structured mesh and unstructured mesh. When cavitation is studied, it will spend a lot of time to create a structured mesh. Furthermore, there is not a standard logic block for all the running status. For a certain runner or blade angle, it could even be hard to find this one. On the other hand, the number of meshes is usually huge and not convenient to be modified. An alternative scheme is to use adaptive Cartesian grid technology. It originated from 80-s of last century [1], and achieved a great development. Now, it is applied to all kinds of simulation of compressible flow [2, 3], such as Aftosmis's team simulate the 3-D flow around aircraft [4]. However, most of current applications focus on the inviscid flow, and pay much attention to the capture of shock wave. Few researches have been done on the viscous adaptive Cartesian grid technology. Although a lot of progress has been made to deal with this problem, the auto generation of viscous grid around the complex geometry still remained to be

* Corresponding author; e-mail: liuyanhua@zju.edu.cn

solved. The viscous grid highly depends on the geometry and tendency, so geometrical information should be included to generate an auto Cartesian grid. Actually, most code of the existent adaptive Cartesian grid is to treat relationship between grid and physical boundary. Here, a new technology is proposed to solve above problem. The auto Cartesian grid is no longer used to generate the mesh near the physical boundary. Based on the characteristic of boundary flow, body-fitted mesh is introduced into this procedure, incorporated with the existent Cartesian grid.

Theory

Gridding technique

The key of traditional Cartesian grid technology is the treatment the intersection between the faces of Cartesian cells and the geometry surface, construction of volume grid and data structure. Due to the hybrid mesh of the body-fitted viscous grid adjacent to physical surface, the Cartesian grid in the far-field and the unstructured grid connected to these two kinds of mesh. The intersection problem no longer exists. The only problems are the construction of volume mesh and data structure. The data structure is also a very important issue to the Cartesian grid technique. There are three types: Octree (Quadtree) [5], block mesh structure and unstructured mesh. Octree is the most wildly used data structure, which take the low-level cell as the parent and high-level cell as the son. Each low-level cell has 8 (or 4) high-level refined cells. This structure is consistent with the nature of geometric refinement of Cartesian cells, and it is convenient to describe the relationship of cells in different levels. The disadvantage is that it needs to traverse the tree to take the refinement. The block mesh structure categorizes different level mesh into block. It needs to mark the grid and the code is also more complicated. The body-fitted grid is also a wildly-used technique in computational fluid dynamic. There are two kinds of body-fitted technique: differential equation and surface growth. The former transforms the uniform grid in computational space into the orthogonal grid in the physical space, and the later constructs the prism mesh from the physical surface. When the single-body problem is solved, there is not so much difference between the two. But for multi-body the surface growth method has its superiority. For this reason, we choose the second scheme. For the intermediate area between the Cartesian and the viscous mesh, the triangle mesh is adopted to couple these two. Given the boundary point, Delaunay algorithm is the common one. This algorithm is stable and has smooth transition, which means a good quality mesh will be obtained.

Multigrid technique

Multigrid is an efficient numerical technique to speed up the convergence procedure. Its principle is to do an iterative calculation on the meshes in different scale in order to eliminate the numerical error. While the Cartesian grid technique has a nature of the multigrid structure, it is suitable to use multigrid technique to solve this problem. The major procedure is concerned restriction operator and interpolation operator. The former is to restrict the residual error from the fine mesh to the coarse one, while the latter is to interpolate the solution from the coarse mesh to the fine one. For 1-D situation, the linear interpolation operator and fully weighted restriction operator or direct mapping is preferred:

$$\mathrm{I}_h^{2h}\mathbf{v}^h = \mathbf{v}^{2h}; \quad \mathbf{v}_j^{2h} = \frac{(\mathbf{v}_{2j-1}^h + 2\mathbf{v}_{2j}^h + \mathbf{v}_{2j+1}^h)}{4} \tag{1}$$

where h is the spatial step, I_h^{2h} – the restriction operation from level with h to one with $2h$, j – the grid index.

Testing and results

Cylinder

Figures 1-3 show the viscous grid, the viscous grid coupled with the unstructured triangle grid and the total hybrid grid. The Cartesian grid has 5 levels, and the maximum difference of levels between adjacent cells is 1. In order to ensure each level has enough cells for calculation accuracy, each grid level has a proper range.

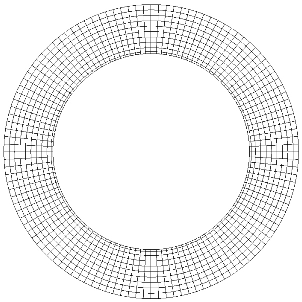

Figure 1. The viscous grid around cylinder

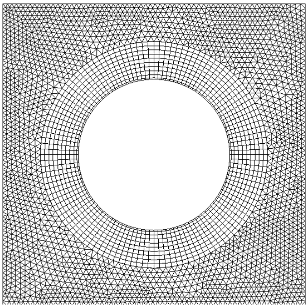

Figure 2. Couple of the viscous grid and the triangle grid

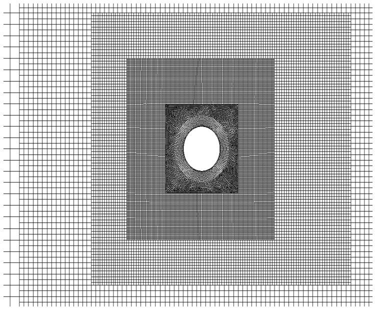

Figure 3. The couple of different grid for flow over a cylinder

For convenience the zones has been set to nested rectangular area, and in the later study, the improvement will be made. For the computation, the whole domain is a 30 × 18 rectangular area. The cylinder is place at the point (6,9) and the diameter is 1. Different parameters are set to calculate cases in Re = 100 and 200.

The result is shown in the tabs. 1 and 2 where Strouhal number St, dragging coefficient Cd, and lifting coefficient Cl are presented. According to comparing the numerical data to the experiment data, it can conclude that a reliable result could be achieved through the viscous Cartesian grid technique.

Table 1. The compare of characteristic parameters with Re = 100

	St	C_d	C_l
Braza *et al.* [6]	0.16	1.28	0.3
Present	0.16	1.3	0.28

Table 2. The compare of characteristic parameters with Re = 200

	St	C_d	C_l
Braza *et al.* [6]	0.19	–	0.7-0.75
Clift *et al.* [7]	–	1.16	–
Present	0.18	1.25	0.68

NACA 0012

Figures 4-6 show three cases. For the viscous boundary grid, the leading edge of the foil is refined to acquire an accurate distribution of pressure. The total level of Cartesian grid is 7. Again, as the same as the flow over a cylinder, each level of grid has a certain region, and the adjacent cells have a maximum difference of 1 level. The current study simulates a basic status: attack angle $\alpha = 0$, Re = 10000, and Ma = 0.01. Figure 7 show that the result from the current simulation is a little different with that from the experiment, but as a preliminary study, the difference is acceptable.

Figure 4. The viscous grid around NACA0012

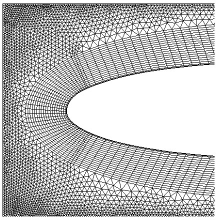

Figure 5. The viscous grid coupled with the unstructured triangle grid around NACA0012

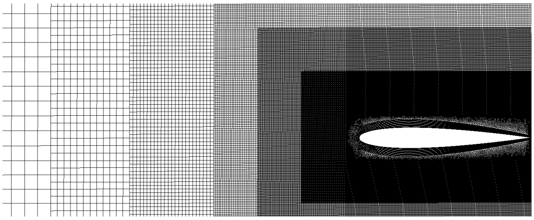

Figure 6. Total hybrid viscous Cartesian grid around NACA

Conclusions

The scheme utilized the body-fitted grid to compose the viscous boundary layer grid, and coupled with the Cartesian background grid with an unstructured triangle grid. The complex procedure to find so-called "cut-cell" is avoided, which tries to find those cells intersecting with the geometry surface. The scheme proposed here is easy to implement, and the resultant mesh has a relative good uniformity. It is easy to be extended to multi obstacle/body and 3-D situation.

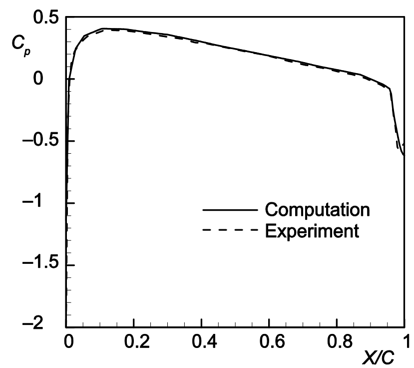

Figure 7. Pressure coefficient along the Chord for NACA0012 at Re = 10000, $\alpha = 0$ and Ma = 0.01

References

[1] Berger, M. J., Oliger, J., Adaptive Mesh Refinement for Hyperbolic Partial-Differential Equations, *Journal of Computational Physics, 53* (1984), 3, pp. 484-512
[2] Dadone, A., Grossman, B., Ghost-cell Method for Inviscid Two-Dimensional Flows on Cartesian Grids, *Aiaa Journal,42* (2004), 12, pp. 2499-2507
[3] Dadone, A., Grossman, B., Ghost-cell Method for Analysis of Inviscid Three-Dimensional Flows on Cartesian-grids, *Computers & Fluids, 36* (2007), 10, pp. 1513-1528
[4] Aftosmis, M. J., Berger, M. J., Melton, J. E., Robust and Efficient Cartesian Mesh Generation for Component-Based Geometry, *Aiaa Journal, 36* (1998), 6, pp. 952-960
[5] Charlton, E. F., Powell, K. G., An Octree Solution to Conservation-Laws Over Arbitrary Regions, *AIAA Paper*, 97-01983 1997
[6] Braza, M., Chassaing, P., Ha Minh, H., Pressure and Velocity Fields in the Near Wake of A Circular Cylinder, *Journal of Fluid Mechanics, 165* (1986), pp. 79-130.
[7] Clift, R., Grace, J. R., Weber, M. E., Bubbles, Drops, and Particles, Academic Press, New York, USA, 1978

Paper submitted: August 1, 2012
Paper revised: September 5, 2012
Paper accepted: September 12, 2012