Frontiers in Computing and Intelligent Systems
ISSN: 2832-6024 | Vol. 1, No. 3, 2022

Computational fluid dynamics grid technology development

Mengjiao Gou, Bohua Liu, Xiaomao Sun, Yuli Ma

Xi'an Shiyou University, Xi'an, Shaanxi 710000, China

Abstract: This paper reviews the development of computational fluid dynamics, especially computational aerodynamics. This paper summarizes the achievements of CFD in grid technology, analyzes the existing problems and perplexities, and prospects its development trend. The CFD grid technology includes structured grid, unstructured grid, hybrid grid and overlapping grid.

Keywords: Computational fluid dynamics; Mesh technology; Interdisciplinary.

1. Background of computational fluid dynamics

As an important branch of mechanics, fluid mechanics was established as a truly rigorous discipline in the second half of the 17th century and has been developing ever since. The formation of fluid mechanics is firstly attributed to Newton's calculus principle. In 1678, Newton put forward the shear stress formula of viscous fluid, which created conditions for the establishment of equations of motion of viscous fluid. In 1775, Euler, the founder of theoretical fluid mechanics, introduced the method of fluid motion description to study the fixed-point parameters in the flow field, and established the equations of motion for the study of investable and in rotational fluid. In 1781, Lagrange introduced a method for determining fluid particles that perfected the study of invistic and in rotational motion.

Today, CFD technology represented by numerical solution Euler equations and RANS equations has been widely used in aviation, aerospace, meteorology, ships, weapons and equipment, water conservancy, chemical industry, construction, machinery, automobiles, oceans, sports, environment, health and other fields remarkable achievement. In aerospace and other fields, CFD has revolutionized traditional methods of fluid mechanics research and design, driving technological advances in these fields.

2. Grid technology

The reasonable design and high-quality generation of grid are the preconditions of CFD calculation, one of the most important decisive factors affecting CFD calculation results, the most manual workload in CFD work, and also one of the bottleneck problems restricting the efficiency of CFD work. Even in countries where CFD is highly developed, grid generation still accounts for 70 to 80 percent of the total human time for a CFD computing task. Steger, one of the pioneers of CFD and grid generation, pointed out in 1991 that "grid generation is still a key step towards most applications of CFD" and that "complex shape grid generation requires dedicated teams". Grid generation is a tedious task, especially for complex aerospace vehicles. Therefore, it is very important for the application of CFD to find ways to simplify mesh generation, reduce the manual work of mesh generation, and improve the adaptability and flexibility of mesh to complex configurations. If this problem is solved, it will greatly promote the popularization of CFD in aerospace and other fields. The development of structural mesh, unstructured mesh and hybrid mesh are introduced in the following.

2.1. Overview of structured and unstructured grids

The computational grid can be divided into structured grid and unstructured grid according to the adjacency relationship between grid points. The adjacency between the grid points of the structural grid is ordered and regular, and the elements are two-dimensional quadrilateral and three-dimensional hexahedron. The adjacency between unstructured grid points is disordered and irregular, and each grid point can have different number of adjacency grids. The elements have a variety of shapes, such as two-dimensional triangle and quadrilateral, three-dimensional tetrahedron, hexahedron, triangular prism and pyramid.

2.2. Unstructured network technology

Since the 1980s, unstructured grids have been rapidly developed and applied in the field of CFD combined with finite volume methods. The basic idea of unstructured grids is that triangles and tetrahedra are the simplest shapes in 2D and 3D domains, respectively, and any region can be filled by them. There are three basic methods for unstructured mesh generation: array advance method, Delauny method and quad-octree method.

Unstructured grids can be divided into two types in terms of storage modes of flow field variables: lattice center type and lattice point type. The unknown quantity of lattice centered type is located in the center of the cell, and the control body is taken as the grid cell itself. The numerical flux needs to be calculated on the surface of the grid cell. The lattice type unknowns are located at the element nodes, and the control volume is usually constructed by connecting the center of the element around the node with the midpoint of the edge. Numerical fluxes need to be calculated on the surface constructed from the edges of the original grid, so lattice type discretization corresponds to edge-based data structures. For the same unstructured grid, lattice-centered discretization contains more degrees of freedom than lattice-centered discretization, which theoretically costs more and achieves higher computational accuracy. On THE OTHER HAND, THE lattice - centered grid template is SPARser than the lattice - centered grid template, so for the same number of unknowns, the numerical accuracy of lattice - centered discretization is higher than that of lattice - centered discretization. After more than 20 years of research, the superiority of these two numerical discretization methods remains to be determined. The numerical experiment conducted by Mavriplis in 2003 proved that, for the same computational grid, the calculation accuracy of lattice-centered discretization is higher than that of lattice-centered discretization, but the calculation efficiency is lower than that of lattice-centered discretization. The study of Levy and Thacke proved that, for transonic flow, if the number of surface mesh variables of the two grids is matched, the two discrete methods will achieve similar computational accuracy.

2.3. Hybrid Grid

Hybrid grid basic idea is: make full use of the structural grid CFD calculation of the characteristics of high precision, high efficiency, high stability, make full use of unstructured grid generation is simple, flexible, adaptable geometry characteristics of the object surface, such as large area USES a simple grid structure, flow gradient using unstructured grid geometry flexibility, the relatively simple structure of the grid area. The domain is connected by filling with unstructured mesh, and the flux conservation at the interface between structured mesh and unstructured mesh is strictly guaranteed to avoid interpolation errors. Hybrid grid technology began to be more and more attention, the current the method of using hybrid grid N - S equation can be divided into two categories:

(1) the structure/unstructured hybrid grid near the surface of the viscous effect area USES the structured grid, other area USES the regular tetrahedron grid hybrid grid technology: first, to the problems of multi-body each monomer or complex problems of each sub-block. Then, a hole is dug at the interface between the volume and the block, and the hole is filled by the unstructured mesh to realize the flux conservation between the two adjacent meshes. This technology is actually a development of overlapping grid technology, among which "zippergrid" and "Dragongrid" are representative methods, as shown in Figure 1.

(2) unstructured hybrid grid in the viscous effect using three prisms and the pyramid grid and the other flow field area using tetrahedral unstructured grid, this approach makes full use of the triangular prism and the pyramid grid high tensile property, can achieve similar structure grid viscous simulation ability, Therefore, it has been developed and applied quickly.

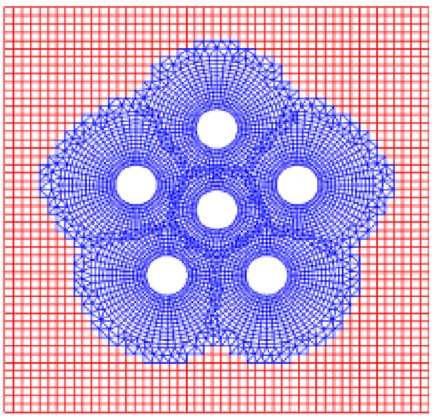

Fig.1 Diagram of Dragon grid

2.4. Overlapping grid

Benek and Steger et al. proposed the concept of "chimeragrids" (overlapping meshes) in 1982, which divided a complex flow area into multiple sub-regions with simple geometric boundaries. The computational meshes in each sub-region were generated independently, overlapping, nesting or covering each other, and the flow field information was interpolated Matching and coupling are performed at the boundary of the overlap region. The generation of overlapping meshes with complex shapes can be divided into four steps: geometric shape generation, surface mesh generation, volume mesh generation, and defining the interpolation relationship between meshes. The top three can draw lessons from the existing general three-dimensional software implementation, such as Gridgen EAGLEView, IGG, ICEM, etc. To define the interpolation relationship between meshes, three key technologies are required, which are overlapping of wall meshes, hole digging and point finding, which are also the core technologies of overlapping meshes. At present, overlapping grid programs and software developed abroad include PEGASUS5, SUGGAR, OVERGRID, etc. However, there is little research on overlapping grid in China, and there is no software with similar function.

2.4.1. Hole digging

If an overlapping grid cell falls into a non-permeable surface (such as object plane, symmetry plane or manually specified hole digging surface) of another grid domain, it shall be marked as "point in hole" and shall not participate in the calculation of flow field. This process is vividly called "digging holes". The result of digging holes produces hole boundary surfaces closely surrounding the points inside the holes, which are used to interpolate and transmit the information of flow field solutions in different regions, and belong to the interpolated boundary surfaces. Therefore, the mathematical essence of the process of digging a hole is equivalent to solving a problem called "the relative position of a point and a closed surface".

The research on tunneling methods mainly focuses on how to improve the reliability, efficiency and automation of the tunneling process. Steger et al. mainly used the dot product result of the normal vector of grid surface and the relative position vector of grid points to judge the relationship between grid points and surface. This method is ERRor-prone IN the presence of concave surfaces, and the efficiency of digging holes is proportional to the number of grid points tested and the number of points determining the surface to be dug. Therefore, a variety of modification methods were developed. For example, the excavated surface was decomposed so that each surface was convex preserving. A combination of analytic shapes, such as a sphere, a cylinder, a cuboid and other simple geometric shapes, is used to replace the grid surface as the surface for digging holes. The ray intersection method determines the position of point P by finding the number of intersection points between any ray starting from point P and the closed hole surface. If the ray intersects the surface for odd times, the point is inside the hole surface; if the ray intersects the surface for even times, the point is outside the surface. This method involves the intersection operation, so the calculation amount is relatively large, but the efficiency of digging hole can be improved by using the ADT tree (Alternator digital tree). Chiu and Meakin proposed the hole-map method. In this method, the excavation surface was projected into the auxiliary Cartesian grid, so as to obtain the approximate excavation surface composed of Cartesian grid. According to the different relative positions, the Cartesian grid cells were divided into "in-hole elements", "out-hole elements" and "edge elements", so as to make points The relationship between and surface is transformed into a simple relationship between point and hole mapping units. The hole mapping method is highly efficient and automated, and requires very little memory, so it has been widely used, such as the Pegasus5 program. ObjectX-raymethod is an improvement of the hole-map method. It replaces the three-dimensional Cartesian grid of the hole-MAP method by establishing a two-dimensional Cartesian grid projected by the hole surface in the coordinate axis direction. The projection of points in the two-dimensional Cartesian grid is found, which is similar to the method of ray intersection, and the number of intersection points of the ray intersection of the projection point along the axis direction and the hole surface is obtained, so as to determine the point is the point inside the hole or the point outside the hole. Compared with the hole-MAP method, the proposed method has higher hole digging efficiency.

2.4.2. Point Finding

The problem to be solved by point finding can be simplified as follows: Given the coordinates (xp,yp,zp) of a point P in physical space, find the logical coordinates (i,j,k) of the grid cells surrounding the point in the grid. For structural grids, it is the problem of finding a hexahedral element to surround a given point in a discrete space. The efficiency and accuracy of point finding have great influence on the whole overlapping grid method, sometimes even the key to the success of overlapping grid method.

The easiest way to find a point is to traverse the grid, where each grid cell is examined to find all possible contributors to the interpolation point, which is the most time-consuming. Stencil walk also known as stencil jump, is the most widely used method at present, and its essence is the modification of Newton iteration method (also known as Newton-Raphson method). Therefore, it has many advantages and disadvantages of Newton iterative method. For example, the search method must have a good starting point, typically a point near a contributing cell. If the given initial point is very close to the contribution unit in the logical space, the method can quickly converge to the real contribution unit; otherwise, it may not converge. For the mesh with large distortion or thin thickness, stencilwalk may not search the correct contribution element.

The inverse map method generates an auxiliary Cartesian grid covering the original grid, and obtains the parameter coordinate values of the grid points in curvilinear coordinate system through the inverse transformation calculation. The position of any given point in the Cartesian grid can be quickly determined by subscript calculation. Due to the fast localization in the process of point finding, inverse transformation method and stencil walk method are used together.

In a word, the efficient point-finding method generally divides the point-finding process into two steps: first, it searches for the nearest grid cell to the spatial point, and then it searches for suitable grid cells nearby to surround the spatial point. Firstly, good data structures such as binary tree, quadtree, octree and ADT tree are used to find the set of possible contribution units closest to the space point. Secondly, the geometric judgment method of relative positions of points and contributing elements is used to find the correct contributing elements of space points among possible contributing elements. For example, the dot-product operation of position vectors is used to judge the relationship between grid points and contributing elements. Finally, trilinear interpolation method is used to solve the interpolation coefficient of space points. If the coefficient is in the range of [0,1], a reasonable contribution unit is found for this space point. If the coefficient is outside the range of [0,1], the contribution unit is taken as the initial unit and the stencil walk method is used for further search to find the reasonable contribution unit.

3. Conclusion

Computational grid is one of the bottleneck problems restricting the development of CFD. At present, the grid needs to make breakthroughs in :(1) efficient, accurate, flexible and convenient complex grid technology, including overlapping grid technology, hybrid grid technology, adaptive grid technology, Cartesian grid technology, etc.;(2) To realize automatic and high-quality quantification of complex mesh generation;(3) Try to break through the dependence of CFD results on grids.

The focus here is on the overlapping grid approach. Although the meshes of each subregion of overlapping mesh method are generated independently, the matching problem between meshes must be considered when the meshes between subregions overlap. If the sub-grids are not matched, the quality of overlapping grids may be poor, there are grid lines in the walls, the number of solitary points is large, the possibility of non-object understanding is increased, the accuracy of flow field solution is low, and even the flow field calculation may diverge due to the sub-domain grid mismatch.

Overlapping mesh technology has been widely used in complex shape mesh generation, especially in engineering problems requiring numerical simulation of the relative motion of multiple bodies, such as submunition like shape object scattering, airborne separation, rocket booster separation and other problems with relative motion. According to our review, the SUGGAR program is the most developed and widely utilized overseas overlapping grid technology. The application of this program is aimed at the multi-body motion problem of complex objects, which includes arbitrary overlap between structured mesh, unstructured mesh and polyhedral mesh, and divides multiple components into sub-components of different levels, so as to realize hierarchical overlap of complex objects.

References

[1] Zhang Leicheng, Guo Xiaodong, Zhao Enduo, He Feng.Development of Computational Fluid Dynamics (CFD) [A]. Beijing Mechanics Society.

[2] Proceedings of the 23rd Annual Conference of Beijing Mechanics Society [C]. Beijing Mechanics Society: Beijing Mechanics Society,2017:3.

[3] Yu Jinwei, Feng Xiaofeng.Review of Computational Fluid Dynamics [J]. Modern Manufacturing Technology & Equipment,2013(06):2526+28.

[4] Sun Liqiang, Wang Di, Zhu Hongbo, Li Fan, Wei Yaodong.Research progress on design and CFD simulation of three-phase separator [J]. Petrochemical Equipment Technology,2017,38(04):6-12+5.

[5] Van Leer B. CFD education: past, present, future. AIAA paper, 1999-0910, 1999 Yan Chao.Computational fluid dynamics methods and applications.Beijing: Beihang University Press, 2006

[6] Mavriplis D J. Aerodynamic drag prediction using unstructured mesh solvers. In: CFD-Based Drag Prediction and Reduction. Deconoinck H, Sermus K, van Dam C.

[7] Eds. VKI Lecture Notes, Von K´arm´an Inst. for Fluid Dynamics, Rhode-Saint-Gen`ese, Belgium, Mar. 2003

[8] Levy D W, Thacker M D. Comparison of unstructured cell and node-based schemes for the Euler equations. AIAA paper, 1999-3185, 1999.