
Meshing of flow and heat transfer problems

Luyao Zou ^a, Zhe Li ^b, Qiqi Fu ^c and Lujie Sun ^d

School of, Shandong University of science and technology, Shandong 266590, China.

^a zouluyaoxf@163.com, ^b 1214164853@qq.com, ^c fuqiqi1223@163.com,

^d 838457922@qq.com

Abstract

The grid is a series of discrete points in the calculation area. Computational fluid mechanics uses discrete control equations to obtain data (velocity, temperature, pressure, etc.) on a grid node using numerical methods, ie numerical solutions. The discrete methods of the control equations mainly include finite volume control method, finite difference method and finite element method. Before the most specific problem is calculated, the first step is to complete the grid generation technique for the calculation area in space. Because the flow problems encountered in the project mostly occur in complex areas, grid generation methods in irregular areas are very important. The most commonly used at present is the unstructured grid.

Keywords

Meshing, ANSYS, Solid model.

1. Introduction

Current fluid computing software uses computational grids. The main idea is to divide the spatially continuous computational region into sufficiently small computational domains. Then, fluid control equations are applied to each of the computational regions to calculate and calculate the fluid calculation equations in all regions. Finally, the physical quantity distribution over the entire computational region is obtained.

Since the generation of body-fitted grids was proposed by Thom-son et al. [2] in 1979, the importance of grid generation technology in computational fluid dynamics has become increasingly known. Today, grid generation technology has evolved into computational One of the important branches of fluid mechanics.

This paper takes the generation of hybrid tube grids as an example. It emphasizes that every step is very important in geometric modeling or meshing as well as establishing mapping relationships. Its accuracy will directly affect the generation quality of the grid and subsequent solutions. Accuracy provides a powerful reference for model post-processing.

2. Grid Definition and Classification

The discrete we mentioned in the finite element method is to divide a continuous elastic body (actually, a geometric region that describes the shape and size of the elastic body, called the solution domain) into units of a certain shape and number so that the continuum can be converted. Become a combination of a limited number of units [1]. Because cells appear as a grid of shapes in appearance, cells can be referred to as grids only geometrically.

In the 2D model, the common grid types include triangular grids and quadrilateral grids, as shown in Figure 1.

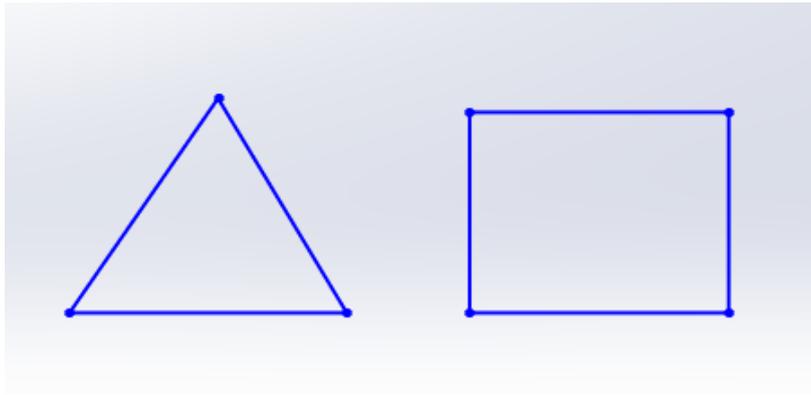


Figure. 1 triangular and quadrilateral meshes

In the 3D model, the common grid types include: tetrahedral and hexahedral grids, prism grids, pyramid grids, and polyhedral grids. The tetrahedral and hexahedral grids are shown in Figure 2.

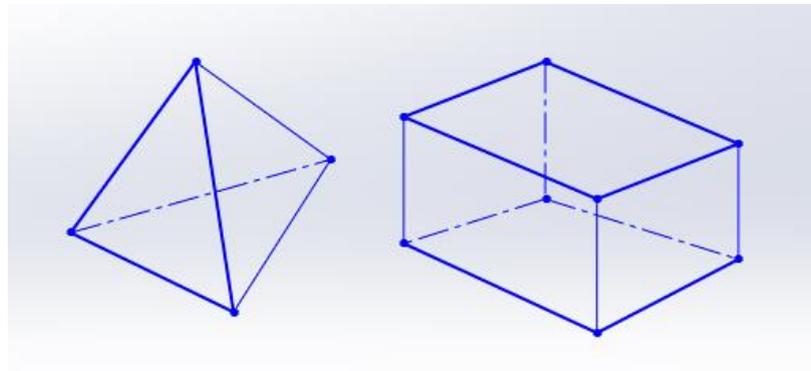


Figure. 2 tetrahedral and hexahedral meshes

The grid can be divided into structural grids and unstructured grids according to storage methods. The workload of generating structural grids is large, but the amount of calculation is small, and the grid quality can be well controlled. At the same time, the boundary layer grid is guaranteed, the calculation speed is fast, and it is easier to converge; the adaptability of the unstructured grid to the complex model Well, the workload is small, but the amount of calculation is large, the computer is more demanding, the grid quality is not easy to control, and the boundary layer mesh is not guaranteed. In this paper, a two-dimensional structural grid is used for numerical calculation.

3. Meshing and Establishing Mappings

3.1 Create a Geometric Model

The injection mixing tube in this paper is a common fluid machine. Its figure size is shown in Figure 3. High-temperature fluid ($t=40^{\circ}\text{C}$) flows from the lower tube diameter at a speed of 1.2m/s, and the low temperature liquid ($t=20^{\circ}\text{C}$) The velocity of 0.4m/s flows from the side pipe diameter, and the high-low liquid blending flows out of the upper pipe diameter.

Use CFD software to create a geometric model:

(1)Create Point: Create the first point in the geometry menu. Create points by defining coordinate increments. Note that each point is defined to be labeled for further processing.

(2)Create Curve: Use the Curve function button in the Geometry menu to connect the created points once, and perform the break line operation on the part where the lower high-temperature fluid is injected.

(3)Micro-processing geometry: Piont[2] is complemented and Part is defined to facilitate subsequent mapping. The processed model is shown in Figure 3. The geometry model is saved and the model is Pipe.tin.

The resulting geometric model is shown in Figure 3 below.

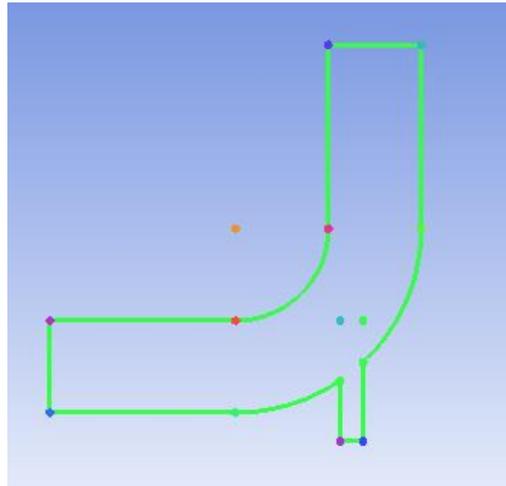


Figure. 3 mixing tube geometry model

3.2 Create and Divide Blocks

(1)Analyze the block generation strategy: Analyze the geometric characteristics of the mixing tube. If the thick tube is straightened into a rectangle, the thick tube and the thin tube are in a regular vertical shape, thereby creating the topological structure of the block, as shown in Figure 4 .

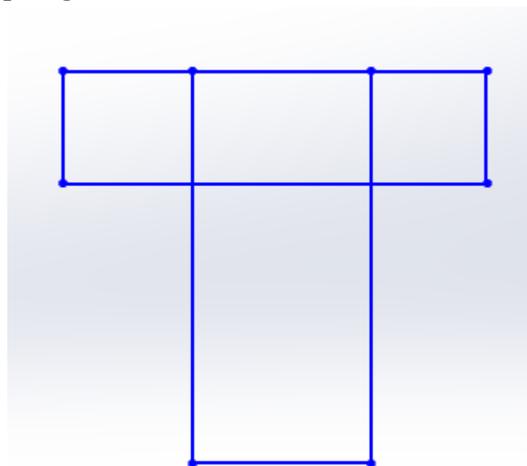


Figure. 4 block generation strategy

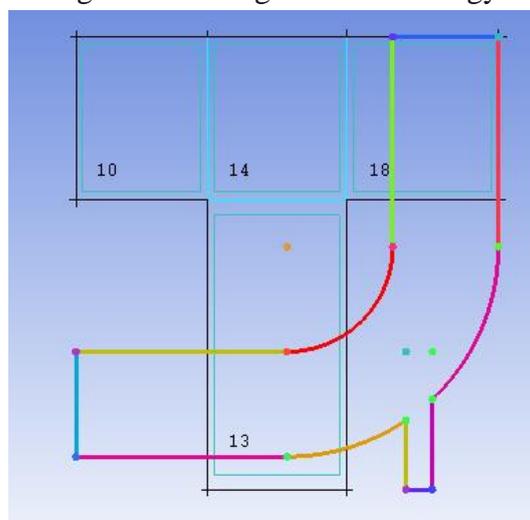


Figure. 5 blocking results

- (2)Initialization block: Using the block menu command, define the name of the block as FLUID and the type of the block as 2DPlane.
- (3)Divide the block: Use the block menu block command to divide the Block once in the Y direction and divide the Blocks twice in the X direction.
- (4>Delete useless blocks: According to the shape of the model, delete useless blocks. The final result is shown in Figure 5.

3.3 Establish a Mapping

- (1)Create a Vertex-to-Point mapping: Use the associated commands in the block menu to connect the points on the edge of the block to the points on the edge of the model. Note that the position of each vertex must be unified, and no chapters can be associated.
- (2)Set up the mapping from Edge to Curve: The theoretical method is the same as the mapping from vertex to point. In order to achieve a better mesh quality, it is necessary to move the vertex bits according to the situation after reassociation.

4. Generate grid and Check Grid Quality

4.1 Grid Preview

The two-dimensional structural grid used in this paper meshes the mixed flow tubes. The method of generating structure grid mainly includes algebraic generation method and partial differential equation generation method. The process of generating structure grid is the coordinate mapping process from computation plane to geometric plane. Among them, the algebraic generation method is to obtain the internal node coordinates through the algebraic difference, and the partial differential equations generate the internal node coordinates by solving a certain form of partial differential equations. The distribution of nodes in this paper is shown in Table 1, and the corrected grid is shown in Figure 6.

Table 1 Node distribution

Edge	Nodes	Mesh Law	Spacing1	Ratio1	Spacing2	Ratio2
I_1	31	BiGeomrtric	0	—	0	—
I_2	15	BiGeomrtric	0.1	1.2	0.1	1.2
I_3	31	BiGeomrtric	0	—	0	—
j_2	21	BiGeomrtric	0	—	0	—

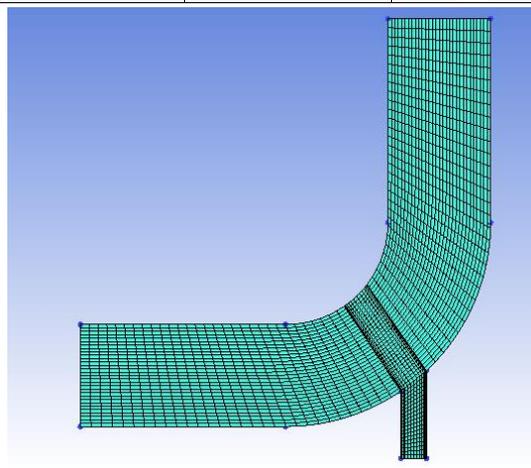


Figure. 6 grid preview

4.2 Check the Mesh Quality

There are certain requirements [3] for ICEM to generate two-dimensional structural grids: 1) The mapping at the boundary must be complete; 2) The two-dimensional grid is located in the X-Y plane. The quality of the mesh is related to the geometric characteristics, flow characteristics, and flow field

solving algorithms of the specific problem. Therefore, the network quality is most importantly judged by the calculation results. However, error analysis and experience show that CFD has general requirements for computational grids, such as smoothness, orthogonality, and sufficient grid nodes distributed in regions with drastic flow changes.

In this paper, the grid quality check adopts Determinant $3 \times 3 \times 3$ determinant. The output grid quality distribution is shown in Figure 7, and the grid quality is greater than 0.9.

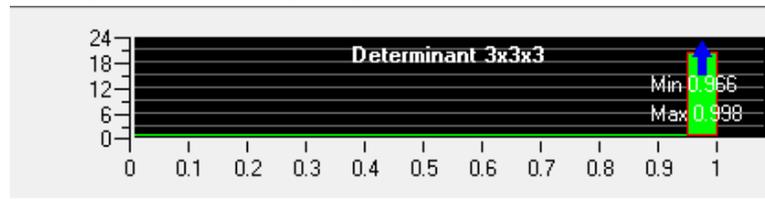


Figure. 7 grid quality

5. Conclusion

Today, most of the numerical calculations are done by computers. The time required to generate grids accounts for 80%-95% of the entire project cycle. High-quality grids will significantly improve the calculation accuracy and convergence speed, shorten the project cycle, and save Project cost.

From the generation of the hybrid pipe mesh in this paper, we can see that every step is very important in geometric modeling or meshing as well as establishing mapping relationships. Its accuracy will directly affect the quality of grid generation and subsequent Solve the accuracy, so the reader should be proficient in the operation of each link, as much as possible to improve the quality of the grid, so as to find the optimal numerical simulation solution.

References

- [1] Song Shaoyun, Yin Fang. St Vinen's principle in finite element mesh division and its application [J]. Mechanical design, 2012 (8): 02-04.
- [2] Ji Bingbing, Zhang Xiaoxia, Gu Yan. Detailed explanation of ANSYS ICEM CFD basic tutorial and examples [M]. Beijing: Mechanical Engineering Press, 2015: 161.
- [3] Hu Kun, Li Zhenbei. Detailed explanation of ANSYS ICEM CFD project[M]. Beijing: People Post Press, 2014:08.