Retraction

Retraction: Research on 3D Fluid Simulation Technology Based on Computer Simulation Technology (J. Phys.: Conf. Ser. 1915 022015)

Published 9 September 2022

This article has been retracted by IOP Publishing following an allegation that raises concerns this article may have been created, manipulated, and/or sold by a commercial entity. In addition, IOP Publishing has seen no evidence that reliable peer review was conducted on this article, despite the clear standards expected of and communicated to conference organisers.

The authors of the article have been given opportunity to present evidence that they were the original and genuine creators of the work, however at the time of publication of this notice, IOP Publishing has not received any response. IOP Publishing has analysed the article and agrees there are enough indicators to cause serious doubts over the legitimacy of the work and agree this article should be retracted. The authors are encouraged to contact IOP Publishing Limited if they have any comments on this retraction.

Retraction published: 9 September 2022
Research on 3D Fluid Simulation Technology Based on Computer Simulation Technology

Hui Li¹,*
¹Wuhan Institute of Shipbuilding Technology, Wuhan, 430050, China
*Corresponding author e-mail: lihui1@wspe.edu.cn

Abstract. With the progress and development of science and technology, 3D fluid simulation technology is gradually moving towards a more systematic and comprehensive development. Of course, more and more people use the three D fluid simulation technology, which further promotes its development. In order to make the three fluid simulation technology more advanced and the whole society, it needs to be studied and improved [1].

Keywords: 3D Fluid Simulation Technology, Two-Way Coupling, Development

1. Introduction
In recent years, fluid simulation has been developed at a high speed. Fluid simulation makes it more possible to simulate natural phenomena based on physical principles of high performance computing requirements. Therefore, fluid simulation is one of the important research directions in the computer simulation of natural phenomena.

2. Research status at home and abroad
The fluid simulation under the physical model contains many fields. The development of these fields has contributed to the research of fluid simulation. Research provided a lot of help [2]. This section will introduce the current research status of the main fields involved in fluid simulation.

2.1. The development of computational fluid dynamics
From the seventeenth century to the nineteenth century, a large number of experiments were conducted on the theory of fluid mechanics, and great progress was made. The Navier-Stokes equation describing fluid motion was established during this period. After entering the 20th century, fluid mechanics has made great progress in theory and experiment. However, because the Navier-Stokes equation is nonlinear and the actual situation is also changeable, there are still very few problems that can obtain analytical solutions [3]. At the same time, fluid mechanics also has many limitations in experiments. Based on these factors, computational fluid dynamics has made great progress.

Since the late twentieth century, computer technology has developed greatly in decades, and at the same time promoted the progress of computational fluid dynamics. It can calculate some complex flows in theoretical fluid mechanics, at the same time it can replace some experiments, and discover some new phenomena that have not been discovered in the past theories and experiments. During this
time, the main progress of computational fluid dynamics can be included in two parts: complex flow fields and turbulence phenomena.

Computational fluid dynamics uses computer simulation to solve fluid-related phenomena, and its main function is to simulate numerical values. Due to its strong ability in simulation, computational fluid dynamics has been used in many fields in many industries [4]. Therefore, we should be full of confidence in the future of computational fluid mechanics, and believe that it will be widely used in the near future.

2.2. Research on fluid simulation technology
In the early stage of fluid simulation, parametric modeling was mainly used, which was limited by the computing power at that time. For example, in order to simulate waves, the wave function is mainly used, and then the particle system is used to simulate the waves. Because using this method can only make the water move around the initial position, this method cannot reproduce the flow scene in reality, nor can it control the effect of the boundary on the water flow. The FFT model is also used to simulate the sea surface [5]. This method can achieve better results on the sea surface with small fluctuations. However, for the models introduced above, it is more difficult to control and cannot be used for complex or detailed simulations. Therefore, researchers began to consider using physics-based methods for fluid simulation.

For fluid simulation based on physical description, it is based on computational fluid dynamics (CFD). Therefore, more and more researchers are looking for solutions for fluid simulation in CFD. Through the study of CFD, we can know that Navier-Stokes equation is the most complete way to describe fluid behavior [6]. For the solution of this equation, there are two main solutions: the first solution is to find a fixed point filled with fluid in space, and then study the changes of fluid parameters at the fixed point, mainly the changes of fluid parameters such as velocity. In addition, when changing from one fixed point to another, the changes of these parameters also need attention. This method is called Euler’s method, which is based on grids; the second method is to track individual micelles in the fluid, study their movement parameters, such as changes in information such as speed and density, and from a micelle Changes in these parameters when turning to another micelle. This method is called Lagrangian method and it is based on particles.

2.3. Research on fluid-solid coupling
The problem of fluid-solid coupling requires attention to the effect of fluid on solids, as well as the effect of solids on fluid. Many literatures only study the one-way influence between fluid and solid. For example, the movement of solids is known, and then the influence of solids on fluid movement is considered, but the influence of fluids on solids is not considered [7]. Or change the movement of the solid according to the movement of the fluid, while the movement of the fluid itself is not affected. Obviously, the real interaction between fluid and solid cannot be restored by the above-mentioned processing methods, so more and more researchers have begun to study the field of fluid-solid coupling.

3. Introduction to related technologies of 3D fluid simulation
3D fluid simulation is a relatively complex project, which involves many theories and technologies. This chapter introduces the main related technologies [8]. These technologies are the basis of 3D fluid simulation, and the technologies mentioned in this chapter will often be involved in subsequent chapters.

3.1. Level set method
The Level Set method is a numerical method mainly used to solve the problem of curve evolution. And it is computationally stable and adaptable to any dimensional space. Osher et al. extended the level set method, and Giga also extended related theories. This method has been widely used in the field of image processing, especially in the field of image segmentation.
(1) The mathematical definition of level set. The level set is defined using mathematical methods as follows: a flat closed curve can be implicitly represented as a level set, which is a level set of a two-dimensional function or called a horizontal line.

(2) The core idea of the level set method. The description of n dimensions can be regarded as a high one-dimensional level set, that is, n1 dimension. In other words, the description of n dimensions can be regarded as the level set of the level set function u, where u has n-dimensional variables. In this way, the evolution of the n-dimensional description can be realized by solving the change of the function u. Therefore, the evolution process of the level set can be obtained according to the change process of the function u. The main point of this change is the introduction of a relatively constant factor in the change: the level c of the level set function u remains unchanged, which is called universal symmetry. After introducing pan-symmetry, it is equivalent to introducing laws. Therefore, the level set evolution equation based on this law can be derived according to some specific conditions.

(3) Level set evolution. An example of the evolution of a level set is shown in Figure 1. From left to right in Figure 1 are the initial state, the mutual propagation and evolution results at the boundary.

![Figure 1. The evolution of level sets](image)

(4) The general algorithm of the level set. The general algorithm of the level set is shown in Figure 2.

![Figure 2. General algorithm for level sets](image)

3.2. Grid for fluid simulation

Because the Navier-Stokes equation is solved in the computer, the discretization step is performed for the equation. For the discretization of equations, it is necessary to define a simulation grid for the fluid. According to the introduction to the grid in computational fluid dynamics, the simulation grid can adopt the following two methods: central grid and staggered grid. The so-called center grid is to store the relevant quantities (such as density, velocity, etc.) in the center of the grid when the fluid is simulated. The biggest advantage that can be gained by using a central grid is that it is relatively simple to implement. The so-called staggered grid is more complicated than the central grid, and the relevant quantities during the fluid simulation process are stored separately. Separate storage mainly refers to storing the velocity of the fluid on the boundary of the grid, while other quantities are defined in the center of the grid. Three-dimensional MAC grid for 3D fluid simulation. It stores the speed at the grid boundary, and the grid unit is shown in Figure 3.
3.3. Grid for fluid simulation

Fluid will fill the space it is in. For fluids, it can be regarded as composed of mass points. When these mass points are in motion, they need to be described. At the same time, the movement of these mass points must be distinguished. For fluid motion, two methods can be used to describe: Lagrange method (Lagrange method, also called mass point method) and Euler method (Eulerian method, also called space point method or flow field method). Before introducing these two methods, we need to explain the concept of flow field: the essence of field is a space, and the special feature of this space is that there is a certain physical quantity filled with it. According to the definition of the field, when the fluid is flowing, the space it fills is the flow field.

1) Lagrangian method For this method, the movement of each fluid particle is the focus of the observer's attention. Observe the movement history of each mass point from beginning to end, that is, how their position changes over time [9]. When the movement of all the particles is known, the law of movement of the entire fluid field can be obtained.

2) Euler method. For this method, the focus is on fixed points in space. When different fluid particles flow through these fixed points, it is necessary to record the movement of the fluid particles passing through these fixed points. In this way, the movement of the entire flow field can be obtained.

3.4. Navier-Stokes equation for incompressible fluids

The velocity of the fluid changes with time and space. It can be expressed as a vector field. Assuming that the fluid is incompressible and uniform, using the two-dimensional Cartesian coordinate space as the parameterized space, for each position, record \( \mathbf{x} = (x, y) \) and the time variable \( t \). The fluid can be represented by its vector velocity field \( \mathbf{u}(\mathbf{x}, t) = (u(x, t), v(x, y), w(x, t)) \) and scalar pressure field \( p(\mathbf{x}, t) \), the changes of these fields over time can be described by the Navier-Stokes equation as shown in publicity (1) and (2):

\[
\frac{\partial \mathbf{u}}{\partial t} = - (u \cdot \nabla) u - \frac{1}{\rho} \nabla p + \nu \nabla^2 u + F
\]  
\[
\nabla \cdot \mathbf{u} = 0
\]  

Where: \( \rho \) is fluid density (constant); \( \nu \) is dynamic viscosity; \( F(\mathbf{f}_s, \mathbf{f}_e) \) represents any external force acting on the fluid. In practical applications, the calculation amount of directly solving Navier-Stokes equations is very large, which is difficult to complete in real time [10]. Therefore, physics-based real-time fluid simulation needs further simplification and more practical algorithms.
4. Two-way coupling method of fluid and dynamic obstacles

4.1. Simulation method of dynamic obstacles
In this article, dynamic obstacles are realized by multi-joint hinge bodies. The dynamic obstacle is simulated as a human body with multiple joints, as shown in Figure 4.

![Figure 4. Dynamic obstacle model](image)

When a dynamic obstacle moves in the fluid, the simulation of the fluid is affected by the movement of different joints. It should be noted here that the movement of dynamic obstacles is described in a generalized coordinate system. The so-called generalized coordinates are not specific coordinates, but independent parameters, or minimum parameters, used to describe the configuration of the system.

4.2. Two-way coupling method
Many researchers have devoted themselves to studying the coupling process of fluid and solid, and have proposed many methods to solve the problems in fluid-solid coupling. The two-way coupling method used in this article, the coupling method adopted in this article will be introduced in detail below. Among them, the boundary part where the fluid and the dynamic obstacle intersect is called the coupling surface.

The coupling between fluid and dynamic obstacles needs to meet the following three conditions:

1. For fluid elements and dynamic obstacle elements on both sides of the coupling surface, the speed in the normal direction should be consistent. This is to satisfy the free sliding boundary condition, that is, the fluid can only slide along the surface of the dynamic obstacle, but cannot flow into and out of the dynamic obstacle.
2. The movement of dynamic obstacles affected by fluid pressure must satisfy the Lagrangian equation of motion.
3. The fluid is incompressible.

The main idea of coupling is:

1. The influence of fluid on dynamic obstacles is realized by the fluid exerting pressure on the dynamic obstacle.
2. The influence of dynamic obstacles on the fluid is realized by transferring the acceleration to the fluid, that is, changing the pressure term of the fluid.

5. Conclusion
3D fluid simulation has always been the focus and difficulty in the field of computer graphics. With the progress and development of computer technology, the integration of computer simulation technology has made greater progress in the research of 3D fluid simulation technology. It is believed that when computer technology is more advanced, the development space of 3D fluid simulation technology will be greater.

References
[1] Chen Guoqiang, Li Qianqian. Research on apparel design application based on 3D virtual simulation technology [J]. Textile Science and Technology Progress, 2017
[2] CHEN Shan. Simulation Research on Low Current Ground Line Selection Technology Based on MATLAB [J]. Electrical Switch, 2015: 79-82+110.
[3] Fu Tao, Du Jianwei, Zhao Zhifeng, Zhang Zhijian. Research and Implementation of 3D Simulation Technology in Communication Equipment Simulation [J]. Gansu Science and Technology, 2014:14-16.

[4] Liu Chundong, Liu Ai, Zhang Donghui, Zou Yilong, Chen Long. Fluid Motion Simulation Technology Research Based on SolidWorks [J]. Journal of Hebei Institute of Civil Engineering and Architecture, 2012:54-56.

[5] Pan Zhengyu, Gu Zhengqi, Zhang Yong, Song Yahao. Research on Efficient Optimization Technology of Automotive CFD Flow Field Simulation Integration [J]. Journal of Hunan University of Technology, 2018:31-37.

[6] Wang Bingbing. Application of Fluid Simulation Technology in Analysis of Nozzle Flow Field [J]. Standardization & Quality of Machinery Industry, 2019:37-41.

[7] WU Jie. Discussion on the Simulation Function of 3D Simulation Software for SMT Reflow Soldering Furnace [J]. Modern Business Industry, 2016:468-469.

[8] Yang Junjie. Research on the application of 3D digital simulation technology to urban services [J]. Science and Technology Innovation and Application, 2017:289-289.

[9] Yun Zhonghua, Li Yongfeng. Research on Simulation Experiment of Converter Flow Technology Based on Simulink [J]. Laboratory Research and Exploration, 2018

[10] Zhu Hui, Yang Zhigang, Tan Peng, Feng Chengjie. Automotive Engineering, 2016:35-39+71. A Comparative Study on the Simulation Capability of Automotive Outflow Field Based on Fluid Simulation Platform [J]. Automotive Engineering, 2016:35-39+71.