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

Application of Artificial intelligence in Computational fluid dynamics

Bohua Liu, Mengjiao Gou, Xiaomao Sun, Hengyi Du

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

Abstract: With the continuous development of artificial intelligence (AI) and computer, the further improvement of computational fluid dynamics (CFD) algorithm and software, artificial intelligence technology has shown its advantages in many fields.AI is becoming increasingly common in engineering applications and is significant in reducing human labor. The purpose of this paper is to summarize the AI technology in the field of CFD, the application of artificial intelligence can through machine learning geometry model parameters, the grid generation technique, the turbulence model calculation, reduce manual intervention, improve the meshing degree, improve the predictive accuracy, rapid turbulence data visualization analysis, bring so much convenient for computational fluid dynamics.

Keywords: Artificial intelligence; Computational Fluid Dynamics; mesh generation; Turbulent flow prediction.

1. Introduction

Computational fluid dynamics (CFD) is a computational tool based on computer technology. It was first used in aerospace industry, and then gradually extended to many fields such as chemical industry, ocean engineering and bridge engineering. CFD has shown great development potential and engineering application value in the aerospace field. It has been widely used in aerodynamic layout design, aerodynamic performance evaluation and complex flow simulation of aircraft. The joint application of CFD simulation and AI technology has become the focus and difficulty of the current CFD development.

On the one hand, in the aero-engine flow field simulation, CFD focuses on the geometric parameter model, and the problem mainly depends on the computer aided design (CAD) system to solve. The models and assumptions used in CFD numerical methods will affect the numerical simulation results, so it is necessary to quantitatively analyze the model data through AI, quantitatively describe the uncertainty range of the prediction results, and evaluate the credibility of CFD results. On the other hand, the model transformation may be biased during mesh generation. As the geometric shape and flow field become more and more complex, the modeling and mesh generation, as an important part of the whole calculation and analysis process, are more difficult to achieve by manual operation, which requires a lot of time and manpower. In the analysis of calculated data, software is generally used to manually analyze and process the data. However, when the amount of calculation is huge, AI technology can greatly reduce the amount of manual operation.

The above problems restrict the application of CFD in various fields. With the development of technology, the application scope of AI has been expanded. In the field of CFD, machine learning can separate tasks, making machine assistance possible. AI can improve the efficiency of CFD meshes, reduce manual intervention, improve prediction accuracy, and realize quantitative data processing.

2. An overview of developments in computational fluid dynamics

Like other disciplines, computational fluid mechanics is built on the basis of theory and experiment. In fact, before the emergence of this discipline, there existed two major fluid mechanics, theory and practice. With the advent of computational fluid dynamics, these problems have been solved effectively. Throughout the development of computational fluid dynamics, indeed in the late 19th century, some scholars put forward the numerical method is used to solve the problem of fluid mechanics, at the time there is no computer technology, combined with various computational tools used by people at that time were very backward, so the thought theory, until before the advent of computer technology has not been effective, In particular, it is not suitable for solving complicated fluid mechanics problems at that time . Since the appearance of the computer, this idea has been fully certified. The appearance of the electronic computer has created a good opportunity for this idea and provided a new way of thinking and method to solve many complex problems.

3. Application of AI technology in Computational fluid dynamics

3.1. Geometric parameters

The problem of establishing geometric figures is the premise of generating high-quality computational grids, and the conclusions drawn are stored in the AI database. Aldefeld used an expert system based on symbolic reasoning and operation to establish a regular geometry system, store geometric constraint relations and other boundary conditions into the intelligent knowledge base, and gradually construct the whole graph by matching the information of the knowledge base. Oldfield's method is generally accepted as the representative of the solution based on regular geometric inference. Gao Shuming and Peng Qunsheng introduced new concepts such as known elements and known constraints, and extended general graphic data structures to express geometric elements and constraints in a unified way. And the geometric model suitable for geometric reasoning is adopted. Modeling is usually done manually by recursive relation between structure and feature. In the process of modeling using commercial software such as Unigraphics (UG), AutoCAD and Pro/Engineer, if a feature parameter or structure size changes, it is necessary to reconstruct the model from the previous step of this feature structure. The model with irregular boundary takes a lot of time and manpower. By USING artificial intelligence method, the constraint relationship and geometric structure between geometric bodies can be described through the cycle and stored in the knowledge database. The geometric model is extracted directly from the database, which greatly improves the modeling speed.

3.2. Grid Generation Technology

Slotnick believes that grid generation and grid adaptive technology are a major bottleneck in the current CFD process. In his CFD2030 Vision Research published in 2014: It is pointed out in "The Road of Computational Aeronautics Revolution" that before the grid error is eliminated, it is not possible to analyze other fluids quickly, and it is not possible to realize the comparison of experimental results of new models. Although there are some advanced grid generation software, such as ICEM, Gridgen, Gambit, etc., it takes a long time to generate a set of suitable grid for complex turbine cooling blade configuration. But for turbine cooling design workers, the maximum time they can accept for an analysis is two days. Consner et al., in one of their articles, pointed out that the key question facing CFD researchers is "how many days can you shorten the overall design cycle". Therefore, the automation of generating complex shape grids has become one of the most challenging tasks in applied aerodynamics and computational fluid dynamics. Many researchers have studied the grid generation technology of turbine cooling blades from different directions, such as partitioned structured grid, unstructured grid, Cartesian grid and overlapping grid. For grid generation software, first of all, its algorithm should have simple input, convenient and flexible use, universality, reliability and high efficiency of calculation, and can meet the computational requirements.At present, the 3D object recognition technology of artificial intelligence has been widely used in this aspect to construct the mesh topology of 3D computing watershed, so as to quickly generate high-quality mesh.

3.3. Calculation of turbulence Model

Reynolds mean method (RANS) enables the solution of Navier-Stokes (N-S) equation to be applied in engineering, but the Reynolds stress term is introduced in this process, and the corresponding expression must be supplemented to make the system of equations closed. Since large eddy simulation (LES), direct numerical simulation (DNS) and other methods cannot be widely used in the short term, RANS-based numerical simulation of turblineering flow will still be the main means of turblineering engineering applications in the next few decades. How to break through the shortcomings of inaccurate flow field prediction in existing turbulence models is still a thorny problem in the past one to two decades, which needs continuous attention. Driven by the maturing of artificial intelligence and its successful application in other fields (such as image recognition, autonomous driving, big data, etc.), the combination of machine deep learning and CFD has begun to emerge. In 2016, Julia Lin used deep neural networks to build the Reynolds stress term model and realized the deep combination of machine learning and CFD, which attracted widespread attention. There are two ways to realize the process: one is to constantly update or optimize the original empirical coefficients based on the existing turbulence model equations and driven by data; The second is to construct a specific input-output relationship equation through certain physical background knowledge, and then construct a new nonlinear model equation through machine autonomous learning Zang Zejia and Duraisamy used neural networks to modify the coefficients of the turbulence model and achieved good results. It is difficult to achieve a breakthrough by modifying the existing turbulence model. The second way of thinking, that is, input-output relationship of deep learning algorithm, can make a breakthrough in turbulent simulation of complex configurations. Tracey [10] used kernel regression to reconstruct the eigenvalues of the anisotropy tensor of Reynolds stress. Julia Lin realized the modeling based on the Reynolds stress anisotropy tensor of deep neural network, and realized the prediction of channel flow through the learning of DNS/LES data, which has very good effect.

4. Examples of application

With the gradual improvement of artificial intelligence and the deepening of its combination with CFD, artificial intelligence has great potential in solving turbulence problems. Three application examples with the passage of time and of the science and technology, AI in the field to start alone more, including in computational fluid dynamics, the combination of the two more than we are studying fluid flow problems more easily, of course, this is mainly thanks to their computer skills is the rapid development of computer industry. It makes the simulation of turbulent flow easier and more efficient and accurate. This makes many working conditions that were difficult to achieve can be automatically obtained with the support of technology. In the calculation software, this paper will use the multi-phase flow mixture model and the standard k-ε model in the AI database to simulate the change process and numerical changes and diffusion of the entire leakage silicone oil in the experimental device. The data of density under different conditions of leakage silicone oil flow rate were calculated respectively, and the corresponding sample database was obtained [11].

4.1. The physical model

As shown in FIG. 1, in the cuboid model, there is an inlet B at the bottom of the model, with constant inlet temperature and water temperature. B is the cooling water inlet, which will continuously provide uniform flow of cooling water at the inlet. When leakage occurs, the heating rod will leak silicone oil C from the breach. After the self-heating rod comes out, the silicon oil will be driven to flow by the cooling water and produce a certain degree of diffusion. The CFD model simulation method can be established by intelligent calculation or self-operation for reconstruction, and the diffusion range can be observed. The following is to simplify the flow process.

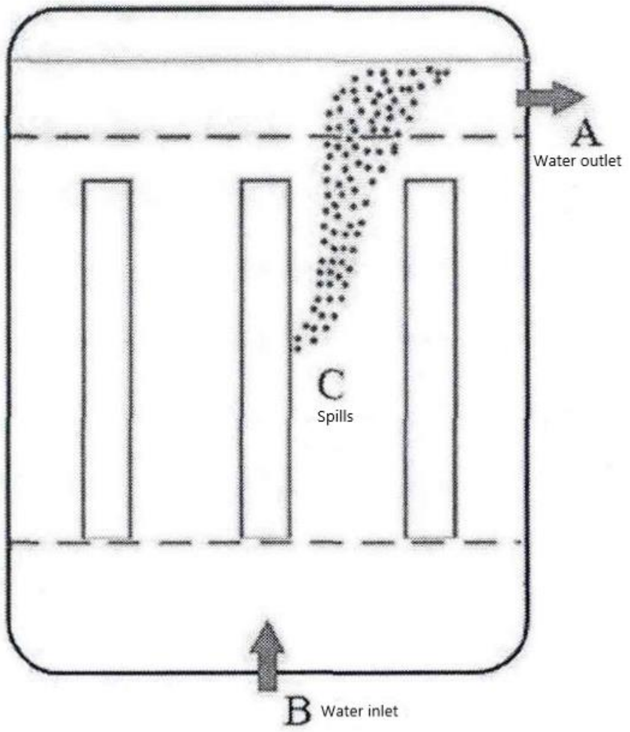

Fig. 1 Schematic diagram of simulated flow

4.2. Meshing

CFD numerical problems and adopted by the flow process of various methods are based on the discretization, it is concluded that the grid requirements of numerical solution is higher, the quality of the mesh model is closely related to whether to solve the convergence, of course also affects the convergence speed of the whole calculation process, high quality suitable and correct mesh is the most basic step in good simulation effect. The biggest advantage of using artificial intelligence for structured meshing is that it saves manpower and makes the simulation automatic. The type 0 mesh is used near the heating rod and encrypted as the manual mesh is used [12]. The number of grid nodes is selected according to the characteristics of the model, and the grid independence test is carried out, which can improve the computational efficiency and reduce the time loss.

4.3. Setting of simulation conditions

The first step to be carried out is to "initialize" the whole simulation process, that is, the initial value of the flow field will be established before the start. The initial value is the starting value for iterative calculation, and the termination value is the value that meets the convergence condition [13]. As the limiting condition of the grid, the most important boundary condition in this paper will be the fixed value of the flow velocity, and the change of the leakage velocity is also a condition that needs to be changed under different working conditions. In order to find the fixed solution of the flow field, the initial value and boundary conditions need to be determined. The closer the initial values and boundary conditions are to the truth, the more attention to detail will make the solution accurate. Initializing Hybrid Initialization, the inlet temperature of cooling water is 293K, the inlet velocity is 0.03m/s, different leakage velocity determines different leakage density, so the density of leakage is related to the velocity of leakage flow at the leakage port.

4.4. The error analysis of

Artificial intelligence in the application of computational fluid dynamics simulation is more convenient, there are still some limitations, particularly in the density of the multiphase flow study, considering its involves many links will have the possibility of errors, from the number of vector, the sampling points, leakage rate and matrix decomposition method to analyze its influence on reconstruction is analyzed, It is helpful to reduce errors in the later research process and make the experimental effect better[14].There are also some limitations in the application of computational fluid dynamics:

(1) It is difficult to establish an accurate mathematical model, determine the boundary conditions related to the flow process, and obtain the complete governing equation of the process;

(2) In the process of numerical simulation, finite element method should be applied to the flow, and the finite element method is developed from solving solid mechanics problems, which is more suitable for low-speed flow problems. How to better apply it to high-speed flow needs further exploration;

(3) There are many turbulence phenomena in engineering practice. The difficulty of turbulence is that in the same flow field, the scale difference of fluid particles is too large and the turbulence itself is nonlinear behavior, so the numerical simulation of turbulence in existing theories has certain limitations.

4.5. Summary of Application examples

The above applications mainly introduce the establishment of the model, the division of the grid, the setting of boundary conditions and error analysis. First established a CFD model, and introduces the meshing process and boundary conditions are set up, by changing the leakage rate of leak and transform the density distribution of the fluid, compared with the database data acquisition nodes form a sample database, sample database after reuse based on singular value decomposition and the nonnegative matrix decomposition algorithm.

It can be seen from the results that, in the process of multi-phase flow movement, non-negative matrix factorization and singular value factorization based on artificial intelligence can quickly extract the basis matrix, improve the efficiency, and can also quickly reconstruct the situation outside the sample database, playing a predictive role.

5. Conclusion and Prospect

The application of artificial intelligence in computational fluid dynamics is various, involving geometric parameter model, mesh division, turbulence model calculation and scientific visualization of flow field, etc. At present, a lot of achievements have been made. The introduction of artificial intelligence into computational fluid dynamics is one of the effective ways to improve the accuracy of forward processing fluid calculation, which has certain universality and popularization.

At present, the main problems of CFD research are the stability of calculation program, the convergence of calculation error and solution, and the cost of CFD technology and software. CFD technology has made great achievements in the field of aerodynamics such as aerospace, but the combination of AI and CFD is not very mature, and a lot of manpower is needed to deal with complex flows. Even if the general CFD software is not suitable for all fluid dynamics problems, we need to make accurate choices according to the research objects. Despite its shortcomings, as a new discipline, the combination of artificial intelligence and CFD is bound to become increasingly mature, and will be more widely used in various fields, and provide technical reference for some unknown fields.

References

[1] HU Ao. Multi-phase flow distribution detection based on CFD information and data reconstruction [D]. North China Electric Power University (Beijing), 2021.

[2] Bingbo Wang. Development and Application of Computational Fluid Dynamics [J]. Southern Agricultural Machinery, 2018,49(09):145.

[3] Wenzhe Zheng. Research on Hydrodynamic Characteristics of Amphibious Bionic Robot Based on CFD [D]. Xi 'an University of Technology, 2021.

[4] Gad-el-Hak M. Nine Decades of Fluid Mechanics[J]. Journal of Fluids Engineering, 2016, 138(10): 100802.

[5] GuangJung Zhou. Five Periods of development of fluid mechanics [J]. Mechanics in Engineering,2001,03:71-75+59.

[6] Li Jiachun. Review and prospect of the development of modern fluid mechanics [J]. Advances in Mechanics,1995,04:442-450.

[7] Miller A, Gidaspow D, Multiphase Flow and Fluidization: Continuum and Kinetic Theory Descriptions, AIChE Journal, 1992, 38(11): 1811.

[8] Miller A, Gidaspow D, Multiphase Flow and Fluidization: Continuum and Kinetic Theory Descriptions, AIChE Journal, 1992, 38(11): 1811.

[9] Shang Jinlong et al. Discrete unified gas kinetic scheme for incompressible Navier-Stokes equations[J]. Computers and Mathematics with Applications, 2021, 97: 45-60.

[10] Arpiruk Hokpunna et al. Finite surface discretization for incompressible Navier-Stokes equations and coupled conservation laws [J]. Journal of Computational Physics, 2020, 423.

[11] Lizhen Chen and Jia Zhao. A novel second-order linear scheme for the Cahn-Hilliard-Navier-Stokes equations[J]. Journal of Computational Physics, 2020, 423.

[12] Zhou Xueyi.Computational Hydraulics [M]. Beijing: Tsinghua University Press, 1995.

[13] Yubo, Lingxiao. Study on laying technology of product pipeline and hot crude pipeline in the same groove [J]. Acta Petrolei Sinica, 2004,28 (5): 149153.

[14] Haiyan Zhao. CFD simulation of sequential transport mixed oil [D]. Daqing: Daqing Petroleum Institute, 2010.

[15] Zhiling Huang,Yongjian Feng. Influence of Different Shape Impeller on Numerical Flow Field in submersible Centrifugal Pump [J]. Oil Field Equipment, 2011,40 (3): 36-39.