Proceedings of the 2nd International Conference on Computing Innovation and Applied Physics
DOI: 10.54254/2753-8818/18/20230343

CFD based aerodynamic optimization design

Yunze Gao

College of Engineering & Applied Sciences, University of Colorado Boulder, Boulder, CO, 80303, USA

yuga7360@colorado.edu

Abstract. The CFD technology based on the N-S equation plays an extremely important role in the detailed aerodynamic shape design of the complex surface of civil aircraft. In this paper, a consistent parameterization method, response surface model and numerical optimization method are used to conclude the optimization design. The aerodynamic optimization based on CFD, the commonly used CFD methods and free-form surface modeling technology are studied, and the challenges faced in the process of aerodynamic shape optimization of civil aircraft are analyzed. The adoption of a genetic algorithm based on the response surface increases the effectiveness of the entire optimization process.The results show that the adopted design method is effective in solving the problem of complex shape optimization using computationally expensive CFD codes. The advantage of the proposed method is that it can flexibly shape the wing body design and can quickly respond to changes in design requirements during the design process; the proposed method can be used in the design of a wider range of complex aerodynamic shapes.

Keywords: erodynamic optimization, parametric modeling, CFD methods, design optimization.

1. Introduction

Computational Fluid Dynamics (CFD) has developed into an indispensable tool for the aerodynamic design of aircraft, and its role in the design process has been continuously enhanced with the enrichment of CFD tools and the improvement of precision, especially for the detailed modification design. , CFD calculation based on high-precision NS equation solution has important application value. The use of CFD can significantly reduce the time and economic cost of aerodynamic design and further improve flight performance. Due to the advantages of CFD in saving development costs, shortening development cycles, realizing digital automation of development, and improving development quality, more and more people believe that the determination of future aircraft performance will depend on data in the "virtual wind tunnel" (CFD) Based on the "virtual flight", this will be the main development direction of aircraft development. Aerodynamic shape optimization is an inevitable trend in civil aircraft design, but at present, in terms of aerodynamic shape optimization, especially in China, there are many optimizations for airfoils, wings and other shapes. There are few optimizations, and the possible reasons are: 1) The flow field calculation requires high precision and is time-consuming. 2) It is difficult to determine the parameterization method and parameter selection of such a complex three-dimensional shape, such as the fairing of the wing-body assembly. 3) These special components, such as the shape of the fairing at the wing-body junction, are difficult to determine the optimization goal, and at the same time, constraints such as the space requirements for the retraction and retraction of the landing gear need to be considered.

These factors hinder the aerodynamic optimization of complex components. However, the rectification at the wing-body junction is extremely important in the entire civil aircraft configuration. In order to increase the aerodynamic performance of the wing-body assembly, it is important to design an effective shape to modify the pressure distribution on the top surface of the wing and the turbulent airflow at the wing-body junction.

In this paper, the aerodynamic shape optimization design of the fairing at the junction of the civil aircraft wing body is carried out under the background of the development of the aerodynamic optimization idea. In order to reduce drag and improve aerodynamic properties, the wing-body rectification's optimal design can, from a narrow perspective, eliminate the airflow separation at the junction of the wing-body. Improve design efficiency. From a broad perspective, it is a practice under the guidelines of the integrated design of civil aircraft. It explores and excavates the difficulties and technical obstacles in the implementation of the integrated design of civil aircraft, and clarifies the future research direction and research content.

2. Background information

Using numerical analysis and data structures, the field of computational fluid dynamics (CFD) studies and resolves issues involving fluid flows. The key reason why the application of CFD can lead to revolutionary changes in aircraft aerodynamic design is that by combining it with the inverse design method or the optimization design method, the geometric change of the aerodynamic shape after the optimized design can be directly obtained. The CFD evaluation of the aerodynamic properties of an existing model translates directly into obtaining the aerodynamic profile of the desired optimized design. Aircraft aerodynamic shape design involves the inverse problem of aerodynamics. For the traditional aerodynamic shape design, due to the lack of effective means to combine aerodynamic characteristics evaluation with aerodynamic shape design, it is usually mainly based on experience or a large number of wind tunnel tests, through manual trial-and-error methods[1-2]. The inverse design method was first proposed by Lighthill in 1945. With the development of CFD, it has gradually developed into an effective design method for aircraft wing/airfoil design [3-5]. For aircraft aerodynamic shape design, the problem of inverse design method is that, firstly, its application is greatly limited due to the difficulty in obtaining the optimal pressure distribution, especially for three-dimensional complex shape applications; Secondly, the application with this method, it is impossible to determine whether the obtained result is really the best design result; Thirdly, it is difficult to realize multi-point design, especially it is difficult to combine with other multi-disciplinary designs to realize the multi-disciplinary comprehensive optimization design required by modern aircraft design. In view of the above problems, another method of aerodynamic shape design, that is, the aerodynamic shape optimization design method formed by combining the aerodynamic positive problem solving and optimization algorithm, has received extensive attention. The optimization design method is to obtain the optimal design method by directly combining the aerodynamic characteristics of different shapes and states calculated by CFD with the target requirements of the design through the optimization search method. This method can eliminate the problems existing in the inverse design method, and is more conducive to the application in aircraft design. At the same time, the optimal design method is more convenient to deal with the design problems of complex shapes involving multi-disciplinary and multi-objectives. Because of this, with the continuous improvement of CFD method and the rapid improvement of computer technology and capability in recent years, the optimization design method of aerodynamic shape has been highly valued, and new design methods have been emerging [6].

The CFD-based aerodynamic optimization design process is generally divided into three steps:

a. Parametric modeling of the shape.
b. Automatic grid generation.
c. Flow field analysis and post-processing of results.

The system framework is shown in Figure 1. In this process, it is very important to choose the appropriate CFD method and the parametric modeling method of the shape. Although the design method of this paper uses commercial CFD and commercial CAD software, it is still necessary to do the analysis on the basic theory of CFD in order to select the appropriate CFD method in the optimization design to improve the efficiency of optimization. At the same time, the widely used CAD software CATIA integrates various curve and surface modeling methods. When doing aerodynamic optimization design, it cannot be directly adopted without analysis. Therefore, it is still necessary to analyze the principles, methods, advantages and disadvantages of the currently commonly used curve and surface modeling techniques in the alternative solutions, to select the suitable solution for this design problem. Appropriate parametric modeling methods can significantly reduce the workload for optimal design.

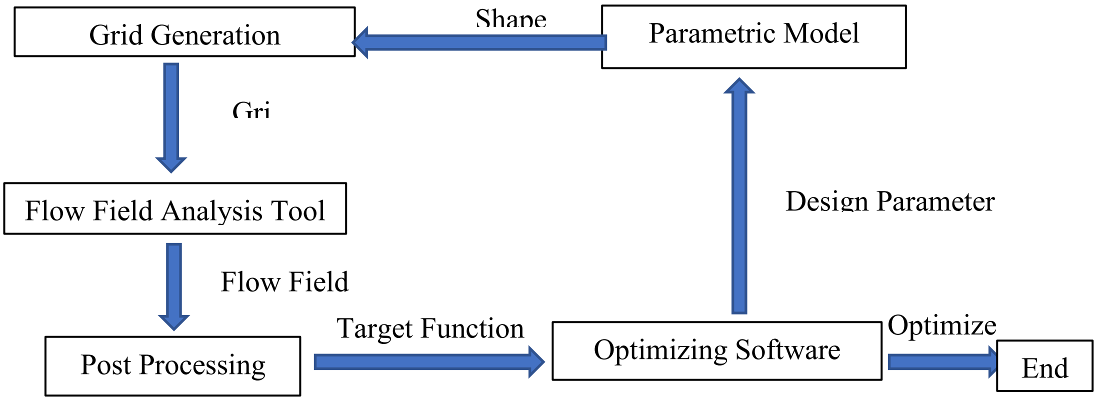

Figure 1. Design flow of CFD based aerodynamic optimization.

2.1. *CFD fundamental theory*

The fundamental theory of computational fluid dynamics (CFD) can be summed up as follows: the original continuous physical quantity fields in the time domain and the space domain, such as the velocity field and the pressure field, are replaced by a set of variables at a finite number of discrete points, establish a system of algebraic equations about the relationship between the field variables at these discrete points by certain principles and methods, then the system of equations is solved for the field variables at the discrete points, and the results are One might think of CFD as a numerical simulation of flow governed by fundamental flow equations (mass conservation equation, momentum conservation equation, energy conservation equation). Through this type of numerical simulation, it is possible to determine the characteristics of the vortex distribution, cavitation, and de-flow area, among other things, as well as the distribution of basic physical quantities (such as velocity, pressure, temperature, concentration, etc.) at different locations in the flow field of extremely complex problems. Calculations can also be made for other relevant physical parameters including torque, hydraulic loss, and the effectiveness of rotary fluid machines. In addition, combined with CAD, the optimization design of aerodynamic shape and structure can also be carried out.

The CFD approach, conventional theoretical analysis approach, and experimental measurement approach make up a comprehensive system for researching fluid flow problems. Figure 2 presents a "three-dimensional" fluid mechanics diagram representing the relationship between the three [7].

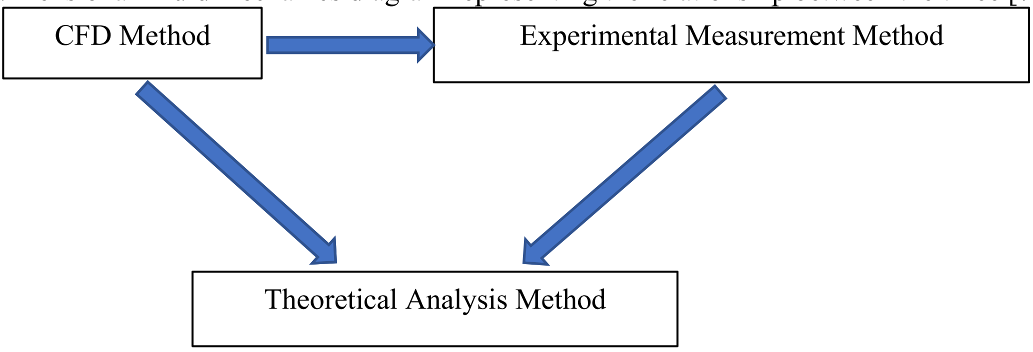

Figure 2. Schematic diagram of three-dimensional fluid dynamics.

The theoretical basis for experimental research and the validation of new numerical calculation techniques, the benefit of the theoretical analysis method is that the produced results are universal and diverse affecting elements are clear. Before theoretical answers can be found, though, computational objects frequently need to be simplified and abstracted. Only a few flows provide analytical conclusions for nonlinear circumstances.

The experimental measurement procedure produced accurate and reliable results. Its significance cannot be overstated because it serves as the foundation for theoretical analysis and numerical techniques. However, model size, flow field disturbance, human safety, and measurement precision frequently place restrictions on studies, and it is occasionally challenging to get results using experimental means. The project will also face a number of challenges, including a high cost, a significant use of labor and material resources, and a protracted cycle.

The CFD method merely addresses the drawbacks of the first two approaches and applies a particular computation on the computer, much like conducting a physical experiment on it. One may see, for instance, the motion and strength of the shock wave, the production and spread of the vortex, the separation of the flow, the distribution of surface pressure, the force magnitude and its variation over time, etc. by performing calculations and displaying the results on the screen. Visualization of the flow scenario is possible through numerical simulation.

2.2. CFD calculation process

We can create our own calculation programs or use commercial software to carry out the necessary operations in order to accomplish CFD calculations. The fundamental operation of both approaches is the same. The method of solving any problem, whether it be a flow problem, a heat transfer problem, a pollutant transport problem, a steady-state problem, or a transient problem, can be shown in Figure 3.

The establishment of governing equations is necessary to solve any problem. The rules of physics' conservation of energy regulate how fluids flow. The law of conservation of mass, the law of conservation of momentum, and the law of conservation of energy are the three fundamental conservation laws.

The beginning and boundary conditions set the stage for the definitive solution of the governing equation. The governing equation, the proper initial conditions, and the boundary conditions make up a full mathematical description of a physical process. The initial condition for the process is the geographic distribution of each solution variable for the object under investigation. Initial conditions must be made to account for temporary challenges. There is no need for initial conditions for steady-state problems. Boundary conditions are changes in a variable or its derivative over time and space at the edge of the problem area. Boundary conditions must always be provided for any problem, and how they are processed has a direct impact on how accurately the calculations come out.

Before attempting to solve the resulting discrete equations while attempting to solve the governing equations, all numerical methods make an effort to discretize the governing equations in the space region. To discretize the governing equations in the spatial domain, a grid must be used. Mesh generation techniques refer to a collection of approaches that have been created to discretize different regions in order to build meshes. The required grid forms alter when different numerical solutions are employed for various issues, but the process for creating grids is essentially the same. Triangles and quadrilaterals are frequently used grid elements for two-dimensional problems, whereas tetrahedrons, hexahedrons, and triangular prisms are frequently used grid elements for three-dimensional problems.

There exist theoretically correct solutions for the partial differential equations generated in the solution domain (or exact solutions, or analytical solutions). However, it is typically challenging to find the accurate answer to the equation due to the complexity of the issue at hand. In order to create a set of algebraic equations concerning the dependent variable values at a select few locations (grid nodes or grid center points) in the computational domain, it is necessary to numerically treat these values as basic unknowns. The values at the node locations, which are obtained by solving a set of algebraic equations, are used to determine the values at other places in the computational domain. Different assumptions about the distribution of the injected strains between nodes and the process of deriving the discretization

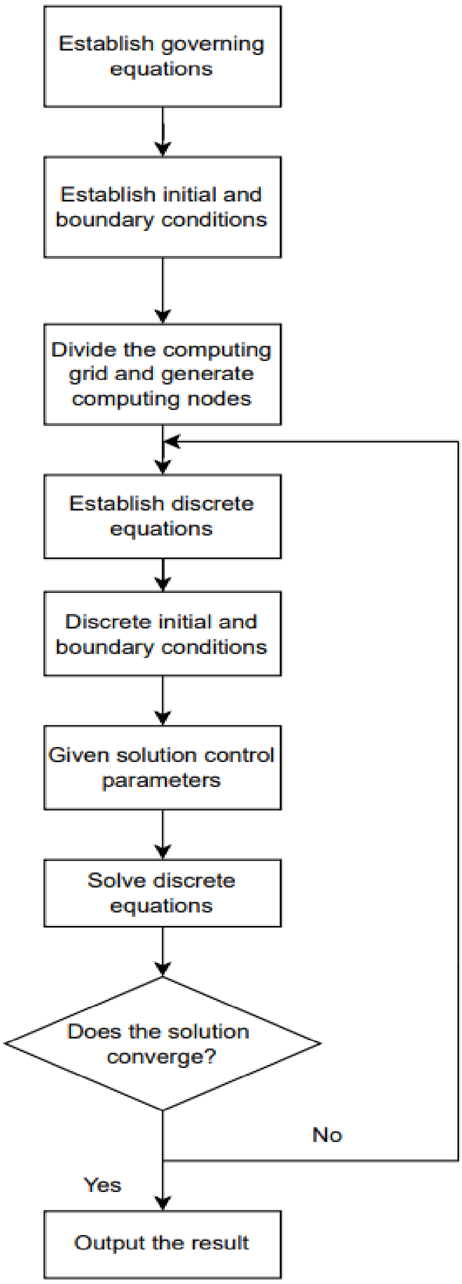

Figure 3. Flow Diagram of CFD.

equation lead to different types of discretization methods, such as the finite difference method, finite element method, and finite element volume method. The convection term's discretization format differs in the same discretization method, such as the finite volume method, which also results in a different final discretization equation.

The step 2 boundary conditions and beginning conditions are continuous. For instance, the wall is immobile and has a zero velocity. The continuous beginning conditions and boundary conditions must now be transformed into particular nodes for the created mesh. The static wall's nodes should have zero velocity if there are 90 of them. In this way, together with the discrete governing equations established at each node, the equation system can be solved. In commercial CFD software, starting conditions and boundary conditions are explicitly specified on the boundary after meshing is finished in the preprocessing step, and the preprocessing software then automatically assigns these initial conditions and boundary conditions to the relevant node up.

After constructing the discretized algebraic equations on the discrete space and using the discretized beginning conditions and boundary conditions, the fluid's physical properties and the empirical turbulence model coefficients are also required. Also included are the time step and output frequency of the transient problem, the control accuracy of the iterative computation, etc. These parameters are not worth discussing or researching in CFD theory, but they significantly affect the calculation's accuracy and effectiveness when it is really being done. An algebraic equation system with specific solution conditions is constructed when the aforementioned settings have been made. In mathematics, there exist analogous solutions to these equations. For example, the Newton-Raphson approach can be used to solve nonlinear equations, whereas the Gauss elimination method or Gauss-Seidel iteration method can be used to solve linear equations. Commercial CFD software frequently offers a wide range of solutions to accommodate various problem types.

Numerous iterations are frequently used to reach the steady-state solution or the transient solution at a particular time step. On rare occasions, the grid shape or size, the convection term's discrete interpolation format, etc., may cause the solution to diverge. If an explicit format is used for time-domain integration of transient problems, the solution may fluctuate or diverge if the time step is too large. As a result, during the iterative process, the convergence of the solution should be checked at all times. The iterative process will end whenever the system gets the desired accuracy.

The findings on the full computing domain need to be represented appropriately after the solution on each computing node has been reached through the solving process. Line value diagrams, vector diagrams, contour diagrams, streamline diagrams, and cloud diagrams can all be used to visualize the results of calculations.

In the above-mentioned CFD solution process, the establishment of the first control equation is particularly important. Different problems and different accuracy requirements can be handled differently by multiple control equations. This is also the main line of CFD development, with the increasing complexity of solving equations and increasing simulation accuracy, as described below.

2.3. Commonly used CFD methods

The tools adopted for aircraft aerodynamic design have undergone significant changes, from the early development of relying heavily on empirical formulas and wind tunnel tests to the current use of CFD tools accounting for more than 70%. The analysis tools of CFD have also experienced the panel method, the full-speed potential equation, and the Euler equation solution to the Reynolds number time-averaged NS equation (RANS). These different CFD codes vary in computational precision, required computational resources, and input information, and play different roles at different stages of design. For example, the panel method is often used in the analysis of airfoils and high-lift devices. The CFD techniques commonly used in the aerodynamic design are briefly described below in order of computational accuracy, confidence in results, and ability to handle complex shapes from low to high.

a.Empirical formulas: The empirical formulas summed up based on a significant number of wind tunnel tests and flight tests still play an important role in the conceptual design stage and can quickly evaluate and optimize the aerodynamic characteristics of different schemes.

b.Panel method: The panel method based on the Laplace equation, combined with the boundary layer theory, can predict the initial separation phenomenon. It can be used in the inverse design of airfoils. The 3D panel method can predict the flow field characteristics of complex shapes including airfoil assemblies and can still play a role in the preliminary design stage.

c.Euler method: Combined with boundary layer theory, the Euler equation-based solver has been applied more instead of the surface element method and is applied to the aerodynamic design of high subsonic airfoils, wings and wing-body assemblies, Euler's method can give a more accurate prediction of wave resistance than this previous method.

d. N-S equation: The Reynolds time-averaged N-S equation can be used to calculate most flow fields and can handle flow field analysis of complex geometric shapes, but it requires more computing resources and takes a long time to calculate. In the preliminary design, stage applications are restricted. However, it is an important tool for three-dimensional fine aerodynamic design such as wing-body fairing, high-lift device slide fairing and calculation of engine nacelle interference.

The comprehensive use of a variety of CFD models with different precisions can effectively improve the efficiency of aerodynamic optimization design and the reliability of the results. In engineering applications, the choice of CFD code is determined by the physical phenomena to be described and the analytical capabilities of the CFD model.

2.4. Free-form surface modeling methods

The mathematical model of parametric geometric shape is the basis for the optimization design of aerodynamic shape. First of all, the model needs to be able to correctly reflect the geometric shape of the aerodynamic shape. Variation-controlled geometry can incorporate the aerodynamic profile required for optimization. Secondly, the model should be able to express the most complex shape with the fewest design parameters, and it should be easy to realize the requirements on the surface quality. Therefore, the choice of parametric modeling method is very important for the optimization of aerodynamic shape. Selecting a geometric parameterization method needs to consider many factors. A good parameterization method should have the following characteristics: 1). The geometric shape represented by it has continuity; 2) The modeling efficiency is high, and its numerical algorithm is stable; 3) Relatively few geometric variables are required; 4) The design parameters have clear physical meaning; 5) It is easy to implement in a CAD system; 6) There are good boundaries and it is easy to use in optimization; 7) It is easy to calculate the sensitivity information. It is not easy to have all the above characteristics at the same time. In engineering practice, it is often necessary to balance appropriately according to the characteristics of the design task to achieve the best effect. For example, in the preliminary design stage, the requirements for efficiency are often higher than the requirements for accuracy, and a parameterization method with few geometric variables is required; while in the detailed design stage, the selected parameterization method must meet the high-precision requirements for shape design. In addition, from the perspective of optimization, the smaller the degree of coincidence between design variables, the higher the efficiency of optimization, especially when the number of variables is large.

The modeling process of geometric parameterization is generally realized in the CAD system. By changing the relevant parameters, the geometric definitions of different topologies or shapes can be obtained. The transfer of geometric data from the CAD system to the grid system can be realized in two different forms. The "loosely coupled" way is through a third-party data exchange standard (IGES or STEP, etc.), and the "tightly coupled" way is through the CAD data interface provided by a specific CFD program, such as FLUENT for CATIA V5 provided by FLUENT. The former sometimes produces changes in shape topology with parameter changes, which is not conducive to the realization of mesh automation; the latter is generally better than the former in maintaining the consistency and integrity of geometric data, but only for specific solvers and relatively simple shape change. The former has a wide range of applications and can relatively easily replace solvers. It has generality and scalability. By combining multidisciplinary solvers, it can be expanded into a multidisciplinary optimization system.

3. Conclusion

This paper studies and analyzes several key issues in the aerodynamic optimization design based on CFD: the solution process of CFD is briefly analyzed, and the application of commonly used CFD techniques is compared, and the CFD method suitable for this study is selected; at the same time, the free curve surface is compared. There are several methods commonly used in modeling techniques. The parametric modeling method can realize rapid prototyping, and can coordinate engineering constraints for modification; under the premise that other parameters are determined, it can give guidance on the adjustment of two parameters; From the big drum bag to the small drum bag, and then returning to the big drum bag, the design idea of realizing a better match with the small drum bag has certain application value. The future study and improvements can be focused on the different methods of Free-form surface modeling methods including Bezier methods, B-spline curve and free-form surface method.

References

[1] A.Jameson. Efficient Aerodynamic Shape Optimization. AIAA 2004-4369, 2004.
[2] Takanashi S. An Iterative Procedure for Three-Dimensional Transonic Wing Design by the Integral Equation Method. AIAA-84-2155, 1985.
[3] Campbell, R.A., and Smith, L. A. A Hybrid Algorithm for Transonic Airfoil and Wing Design. AIAA-87-2552, 1987.
[4] Yu, N. J., and Campbell, R. L. Transonic Airfoil and Wing Design Using Navier-Stokes. Codes. AIAA-92-2651, 1992.
[5] Taylor, A.C., Hou, G.W., and Koriv, V.M. Methodology for Calculating Aerodynamic. Sensitivity Derivatives [J]. AIAA Journal, 1992, 30(10):2411-2419.
[6] N M Alexandrov, R M Lewis, C R Gumbert, L L Green, P A Newman. Optimization with Variable-fidelity Models Applied to Wing Design. AIAA 2000-0841, 2000.
[7] J.D. Anderson, Computational Fluid Dynamics: The Basics with Applications. McGrawHill.1995