Research on Aerodynamic Optimization of Aircraft Engine Nacelle Based on Multi-Precision Deep Learning (MFDNN)

Che Chen *

Department of AAE, The Hong Kong Polytechnic University, Hong Kong, China

* Corresponding Author Email: 23104192d@connect.polyu.hk

Abstract. The design of an aircraft engine nacelle directly affects aircraft drag, fuel consumption, and noise levels, and is a key step in aerodynamic shape optimization. Traditional optimization design methods based on computational fluid dynamics (CFD) rely heavily on high-precision simulation data. However, high-precision CFD calculations are expensive and time-consuming, severely limiting the exploration of the design space and the efficiency of optimization. With the development of artificial intelligence (AI), optimization methods based on surrogate models have become a research hotspot. However, existing methods often rely on data of a single precision and are unable to effectively utilize abundant and easily accessible low-precision data resources. This study aims to introduce a multi-fidelity deep learning (MFDNN) method for the aerodynamic optimization of aircraft engine nacelles. By fusing CFD data of different precisions, a high-precision and efficient aerodynamic performance prediction model is constructed. This method significantly reduces the number of expensive CFD calculations while maintaining optimization accuracy, providing a new technical approach for the rapid and efficient design of aircraft engines.

Keywords: Multi-Fidelity Modeling; Aerodynamic Optimization; Aircraft Engine Nacelle; Deep Neural Network; CFD.

1. Introduction

The safety of air transport has always been a concern for the public and industry, and the takeoff phase of an aircraft is an accident-prone phase that cannot be ignored. The authoritative statistics of the General Aviation Accident and Incident Classification Team (CICTT) (as shown in the figure) clearly show that among the 1,775 fatal in-flight incidents covered in the statistics, loss of control (LOC-I) and controlled flight into terrain (CFIT) were the main types of fatal accidents, accounting for 694 and 229 cases respectively as shown in **FIG. 1** [1]. The takeoff phase is extremely risky because the aircraft is in an "energy critical state" during this phase. As the Federal Aviation Administration (FAA) clearly states in its authoritative flight guide: "The takeoff and initial climb is one of the most critical phases of flight. The airplane is operating at low altitude, low airspeed, and high power... with little margin for error." Specifically, the aircraft is just above the critical stall speed, at low altitude, undergoing configuration changes, and operating at high engine thrust, with limited speed margin [2]. At this point, if airspeed is lost for any reason (such as wind shear, bird strike, loss of engine thrust, or improper control), it is highly likely to cause loss of control of the aircraft, potentially leading to an irreversible accident due to insufficient altitude. In this case, how to effectively reduce the drag of an aircraft during flight is particularly important for aircraft flight and air transportation safety. Since drag is related to the flight performance of an aircraft, reducing drag can enable the aircraft to achieve better flight performance and provide the pilot with a greater altitude margin when the engine thrust remains unchanged. Therefore, reducing flight drag is very important for the safety of air transportation. The design of the aircraft engine nacelle directly affects the drag, fuel consumption and noise level of the aircraft, and is a key component for engineers to optimize the aerodynamic shape. Traditional optimization methods based on computational fluid dynamics (CFD) rely heavily on high-precision experimental data [3]. Although high-precision CFD can provide more accurate flow information and predictions, the computational cost is extremely high and the time is long. A single simulation takes several days or even weeks, which greatly limits the efficiency of aircraft shape design. With the development of artificial intelligence (AI) technology, optimization methods based on surrogate models have become a research hotspot. By establishing a mathematical mapping between design parameters and aerodynamic performance, surrogate models can achieve rapid performance prediction at a low computational cost. However, most existing methods rely on a single, high-precision dataset and fail to effectively utilize the abundant, readily available, low-precision data resources. Therefore, this study aims to introduce Multi-Fidelity Deep Learning (MFDNN) into the aerodynamic optimization of aircraft engine nacelles. By integrating CFD data of varying precision, a high-precision and efficient aerodynamic performance prediction model is constructed. This significantly reduces the number of expensive CFD calculations required, lowering design costs while also exploring the potential of MFDNN.

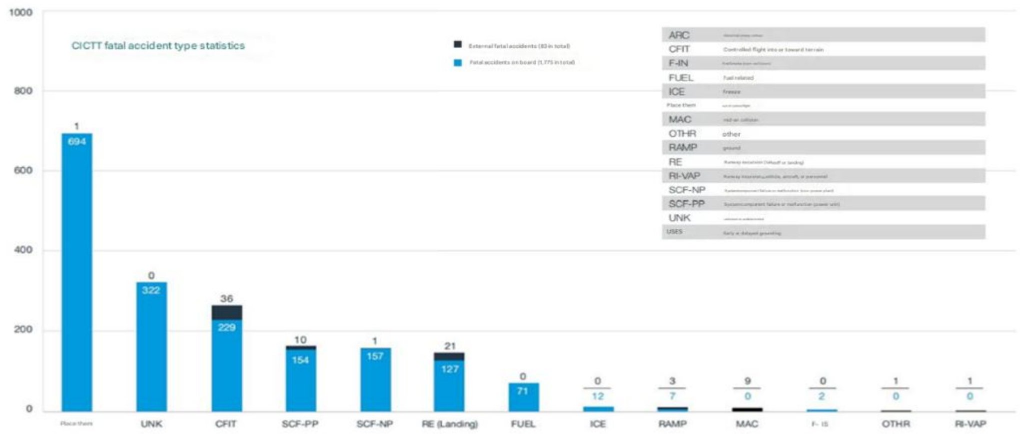

FIG.1 Statistics of air accident [1].

This technology not only has the potential to shorten the design cycle of aircraft nacelles but also provides a new approach to ensuring aircraft flight safety. By optimizing the aircraft nacelle in this way, it can also effectively improve the safety of the aircraft during the takeoff phase.

2. Aerodynamic Design and Drag Mechanism of Aircraft Engine Nacelle

This chapter mainly discusses the role of the nacelle in the aircraft's aerodynamic shape, the source of drag, its relationship with overall aircraft performance, and the limitations of current mainstream optimization methods. The aircraft engine nacelle is a critical aerodynamic component that encloses the engine. Its design is not simply an engine shell, but rather a highly integrated whole that directly affects the aircraft's drag, fuel efficiency, and flight safety. As Li et al. (2013) clearly pointed out: "The integration of the nacelle and the pylon with the airframe is a critical issue in aircraft design, because it significantly influences the aerodynamic performance of the aircraft, especially the drag rise and the stability." [4]. The quality of its design directly impacts the overall aircraft's drag, fuel efficiency, and noise performance. The nacelle's air intake design must ensure stable and uniform airflow to the engine under all flight conditions (especially during takeoff), preventing intake distortion and ensuring engine safety as shown in **FIG. 2**.

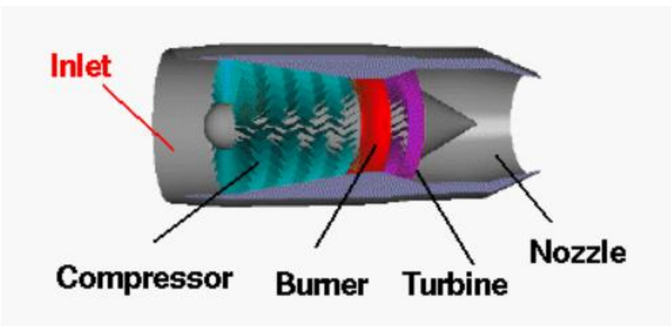

FIG.2 Structure of engine

The core function of the nacelle is to provide a stable and uniform intake airflow to the engine. The air intake is located upstream of the compressor. Although the air intake does not perform any work on the airflow, its performance significantly impacts the net thrust of the engine. As shown in the figure above, air intakes come in a variety of shapes and sizes, with the specific size generally depending on the aircraft's speed [5].

At the same time, the nacelle itself generates drag, and its shape directly affects the aircraft's friction drag, pressure differential drag, and interference drag. Due to the continuous development of high-bypass ratio engines, the size of the nacelle has continued to increase, and its aerodynamic impact on the aircraft has become increasingly significant. Therefore, the optimization of the nacelle shape has also become a problem that airlines need to solve. For example, modern aircraft such as the Boeing 787 and 777X all adopt a laminar flow nacelle design, which effectively reduces frictional resistance by extending the laminar flow area on the nacelle surface. The nacelle and its integrated area with the fuselage are significant sources of aircraft drag. Its composition is complex, primarily frictional drag, caused by the viscosity of air and acting on the entire wetted surface of the nacelle. Research has shown that, at cruise speed, frictional drag on the nacelle surface contributes approximately 80% of the total nacelle drag and around 3% of the total aircraft drag. As Seddon and Goldsmith point out," For a conventional subsonic intake the drag consists of two parts, the cowl drag and the boundary-layer bleed drag. The cowl drag is itself composed of two items: the friction drag and the pressure drag" [6]. Therefore, the nacelle design has a decisive influence on the overall performance of the aircraft. An excellent nacelle design can significantly reduce the resistance of the entire aircraft, thereby reduce fuel consumption and improve aerodynamic kinetic energy. The nacelle also acts as a barrier to engine noise. The shape of the nacelle's air intake and the tail nozzle's profile jointly determine noise transmission and attenuation. Optimizing nacelle design can help meet increasingly stringent airport noise limits. For example, the LEAP engine's nacelle features advanced air inlets and acoustic processing systems [7]. Therefore, reliable nacelle design is an important prerequisite for ensuring stable engine operation. However, the current mainstream method for nacelle aerodynamic optimization design relies heavily on computational fluid dynamics (CFD) numerical simulation. The typical process includes parametric modeling, automatic mesh generation, CFD flow field solution, and optimization algorithm selection. However, traditional CFD has many shortcomings, such as high computational cost. High-fidelity CFD calculations (especially when considering complex physical phenomena such as turbulence and intake-exhaust coupling) require extremely intensive computing resources, resulting in lengthy single calculations and severely limiting the design space. As Slotnick said," The computational cost of high-fidelity CFD simulations is currently too great for routine use in design" [8]. More importantly, nacelle optimization involves many variables and faces challenges. The study further clarified that this is one of the main obstacles to optimization because “as the number of design variables increases, the number of samples required to build accurate surrogate models grows exponentially” [8]. This fundamentally leads to the fact that optimization based on pure high-precision CFD is often limited in engineering practice due to unaffordable time and cost.

3. Advantages and Disadvantages of CFD Technology

Since its rise in the latter half of the last century, CFD technology has revolutionized the aerodynamic design process. By numerically solving the Navier-Stokes equations, it can simulate complex flow phenomena with high precision, becoming the third pillar of aviation aerodynamic design, following theoretical analysis and wind tunnel testing. Despite the cost challenges of CFD-based optimization methods, high-precision CFD simulations have demonstrated significant advantages in nacelle design.

The core idea of CFD is to discretize the real flow field, which has an infinite number of points in time and space, into a finite number of grid points. Partial differential equations are then solved at each grid point to obtain an approximate solution to the flow field. The basic process can be summarized in three steps:

Preprocessing:

Geometric modeling: Create a digital model of the object to be analyzed. Mesh generation: Divide the flow field into a large number of tiny grid cells (such as tetrahedrons and hexahedrons). The quality and quantity of the grid directly determine the accuracy and cost of the calculation. High-precision calculations require a very dense grid to analyze subtle flow structures.

Calculation:

Governing equations: Fluid motion is governed by three conservation laws, mathematically expressed as the Navier-Stokes (NS) equations. They are:

a. Mass conservation equation:

$$\partial\rho/\partial t + \nabla\cdot(\rho V) = 0 \tag{1}$$

Where ρ is the fluid density, t is time, and V is the velocity vector. This equation states that the increase in mass in a fluid element per unit time is equal to the net mass flowing into that element.

b. Conservation of Momentum:

$$\partial(\rho V)/\partial t + \nabla\cdot(\rho V \otimes V) = -\nabla p + \nabla\cdot\tau + \rho f \tag{2}$$

Where p is the static pressure, τ is the viscous stress tensor, and f is the force per unit mass.

c. Conservation of Energy:

$$\partial(\rho E)/\partial t + \nabla\cdot(\rho EV) = \nabla\cdot(k\nabla T) - \nabla\cdot(pV) + \nabla\cdot(\tau\cdot V) + Q \tag{3}$$

Where E is the specific total energy (internal energy + kinetic energy), K is the thermal conductivity, T is the temperature, and Q is the volumetric heat source.

Discretization: Converting a continuous partial differential equation (Navier-Stokes equation) into a system of algebraic equations at a grid point. The finite volume method is generally used (taking a steady-state problem as an example):

The core of the finite volume method is to apply conservation laws to each grid cell.

Basic formula (integral form):

$$\oint\_S F \cdot dS = Q\_v \tag{4}$$

S: Surface area of the grid cell

F: Flux vector (including convective and viscous terms)

dS: Area element vector

Q_v: Source term within the cell

Physical meaning: The net flux through all faces of the cell is equal to the source term within the cell.

Discretization: Discretize the above integral equation into an algebraic equation. For example, for cell i, the discretized equation is:

$$\Sigma\_\{faces\}\,(F\_c - F\_v)\_f \cdot S\_f = Q\_i * V\_i \tag{5}$$

Σ {faces} : Sum over all faces f of cell i.

F_c: Convective flux, related to ρV.

F_v: Viscous flux, related to ∇V.

S_f: Area vector of surface f.

Q_i: Source intensity of element i.

V_i: Volume of element i.

Solving the algebraic equations: Discretization results in a system of linear equations that must be solved on a computer.

Post-processing:

Visualize the results, then analyze and interpret the flow phenomena.

CFD offers high predictive accuracy, using refined meshes and high-order numerical methods to more accurately simulate complex flow phenomena such as turbulence, combustion, and multiphase flow. Its value also lies in its ability to provide full-scenario solutions and accurately analyze the complex vortex systems and separation flows of aircraft. A typical example is the nacelle design study conducted by Mani, Rider & Lunk for NASA's HWB concept aircraft [9]. In this project, researchers used high-precision RANS CFD methods to successfully identify and optimize the complex interfering flow field between the nacelle and the non-traditional airframe. The research showed that through CFD-driven shape optimization, a significant reduction in nacelle drag was achieved. This case fully demonstrates the unique capabilities of high-precision CFD in handling novel and complex aerodynamic configurations and resolving critical interference drag issues. These advantages have led to a large number of successful applications of high-precision CFD in nacelle aerodynamic optimization. Another classic example is the optimization of the aft body line shape of a modern high-bypass ratio engine nacelle. Zhang et al. (2015) elucidate the principles of the thrust-drag accounting system, stating that "In the thrust-drag accounting system, the thrust is defined as the integral of the pressure and viscous forces over the entire internal and external surfaces of the nacelle. The pressure recovery on the nacelle's aft body is no longer counted as part of the thrust but is instead treated as a reduction in the aircraft's drag. This approach allows for a more accurate physical representation of the airframe-propulsion interaction. High-fidelity CFD simulations are crucial in this process to precisely isolate and calculate the drag components attributable to the nacelle's aft body, thereby enabling a clearer understanding and minimization of the interference drag between the nacelle, pylon, and wing" [10]. For example, in the optimization design of a certain type of wing-mounted nacelle, CFD successfully guided the styling design of the fusion area between the pylon and the nacelle, reducing the interference drag in this area by about 15% and effectively improving the pressure distribution on the lower surface of the wing.

Although the advantages are outstanding, the application cost of high-precision CFD is extremely high, which constitutes its core bottleneck - huge computing resource consumption and long cycle costs. In order to accurately analyze complex flow structures, CFD simulation must be carried out with extremely fine grid division and high-order discrete format. As Ferziger and Perić clearly point out in their authoritative work "The number of grid points required to resolve a flow field is proportional to the Reynolds number raised to the power of 3/4. This means that for high Reynolds number flows, the number of grid points, and hence the computational cost, becomes very large" [11]. This means that for problems such as the high Reynolds number external flow field of an aircraft, "the number of grid points and the resulting computational cost become very large." This means that for problems such as the high Reynolds number external flow field of an aircraft, the number of grid points and the resulting computational cost become very large. In addition, although the use of high-order discretization formats improves accuracy, it increases the computational cost of each grid point and the complexity of the algorithm. Ultimately, in the solution process, "the large system of linear equations generated by solving the implicit method constitutes the majority of the total computational workload." All of this leads to an order of magnitude increase in the amount of computation, which in turn requires huge computing clusters and long computation times, severely limiting its widespread application in design iterations.

Given that CFD and wind tunnel testing each have their own strengths and weaknesses, current mainstream industrial practice adopts a strategy of tightly coupling the two. Typically, CFD is used to pre-screen design options and reduce the number of models required for wind tunnel testing. At the same time, wind tunnel test data (such as pressure distribution and force/torque coefficients) is used to verify the credibility of CFD models and calibrate key parameters such as turbulence models. Wind tunnel test data is particularly crucial in areas where the accuracy of CFD predictions remains questionable. However, while this coupling approach is effective, it does not fundamentally address the high cost issue. This is because this coupling method combines two high-cost methods. The entire design process remains cumbersome and lengthy.

However, surrogate model approaches that rely solely on a single high-precision dataset have limitations. The accuracy of the surrogate model is heavily dependent on the number and quality of high-precision sample points. Furthermore, they incur significant computational costs, prolonging the design cycle for aircraft components such as nacelles.

4. Application of Artificial Intelligence and Agent Model in Aerodynamic Optimization

To alleviate the computational burden of CFD optimization, the Surrogate-Based Optimization Model (also known as approximate model-based optimization) has emerged. Its advantage is that it replaces the computationally expensive CFD with a lower-cost mathematical model. With the application of CFD in aerospace design, its high computational cost has become a major bottleneck in the optimization process. As an efficient approximate modeling method, the Surrogate Model effectively reduces the computational burden by approximating complex physical simulation processes through mathematical functions as shown in **FIG. 3**.

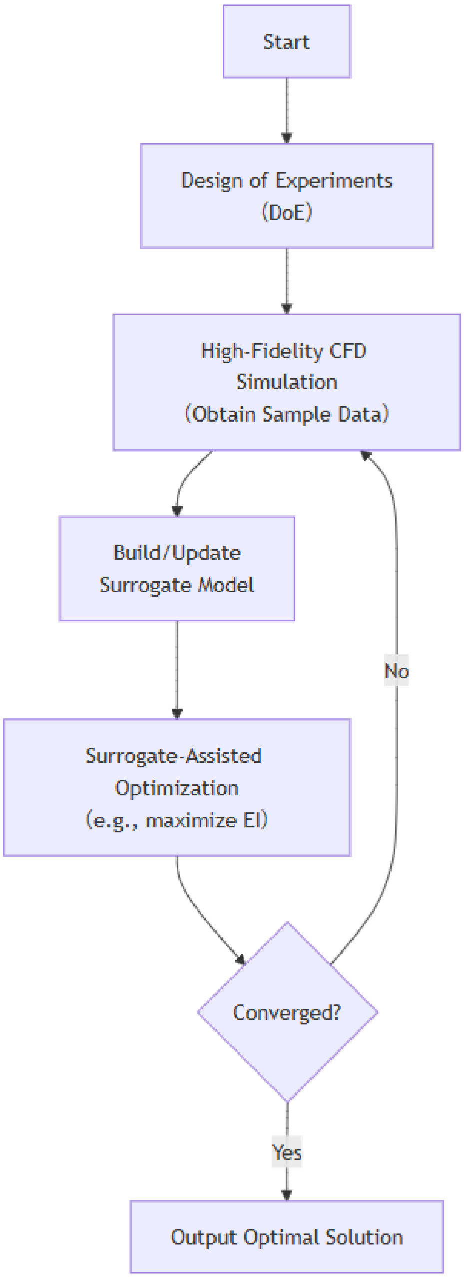

FIG. 3 Agent design process.

Its basic principles and processes include:

First, within the feasible region of the design variables, a representative set of initial sample points, $X = [x_1, x_2, ..., x\_n]^T$, is selected experimentally, where each x_i represents a vector of design variables. Then, the corresponding aerodynamic performance response (such as the drag coefficient) for each sample point is calculated through high-precision CFD simulation, resulting in data $Y = [y_1, y_2, ..., y\_n]^T$. This constitutes the initial training dataset {X, Y}.

Using the dataset {X, Y}, a surrogate model ŷ = f(x) is constructed to approximate the CFD simulation process y = CFD(x). The following are two of the most commonly used and representative surrogate models and their mathematical formulations:

a) Radial Basis Function Model

The RBF model is a powerful interpolation tool whose prediction form is a linear combination of basis functions. For a new point x, the predicted value is:

$$\hat{y}(x) = \Sigma\_\{i = 1\}^n \omega\_i \varphi(||x - x\_i||) \tag{6}$$

ŷ(x) : The predicted value at the unknown point x.

n: The number of known sample points.

x_i: The design variable vector for the i-th known sample point.

||x - x_i||: The Euclidean distance from the unknown point x to the known sample point x_i.

φ(·): A radial basis function, such as the Gaussian function $\varphi(d) = \exp(-\varepsilon d^2)$.

ω_i: The weight coefficient corresponding to each sample point.

The weight vector $\omega = [\omega_1, ..., \omega\_n]^T$ is determined by requiring the model to interpolate exactly at all sample points, that is, to satisfy ŷ(x_j) = y_j for all j=1,...,n. This leads to a system of linear equations:

Aω = Y

Where the elements of matrix A are A_jk = φ (||x_j - x_k||). Solving this system of equations yields the weights ω.

b) Kriging Model

Kriging, also known as Gaussian process regression, not only provides predicted values but also provides prediction uncertainty. It expresses the unknown function as:

$$y(x) = \mu(x) + z(x) \tag{7}$$

μ(x): Global trend function, typically taken as a constant β.

z(x): A Gaussian random process with mean 0 and variance σ^2. Its covariance is defined as:

$$Cov[z(x\_i), z(x\_j)] = \sigma^2 R(x\_i, x\_j;\ \theta) \tag{8}$$

R(x): The correlation function with parameter θ. A commonly used Gaussian correlation function is

$$R(x\_i, x\_j; \theta) = \exp[-\Sigma\ \theta\_k\ (x\_i\^k - x\_j\^k)^2] \tag{9}$$

The model parameters (β, σ^2, θ) are typically determined via maximum likelihood estimation. For a new point x, kriging provides an optimal linear unbiased prediction. Its predicted value ŷ(x) and prediction mean squared error $s^2(x)$ have closed-form solutions:

$$\hat{y}(x) = \beta + r(x)\^T\ R^{-1} (Y - 1\beta) \tag{10}$$

$$s^2(x) = \sigma^2 [1 - r(x)\^T\ R^{-1} r(x)] \tag{11}$$

After the surrogate model f(x) is constructed, optimization is performed on the model itself, rather than directly invoking expensive CFD. To efficiently find the global optimal solution, strategies such as Efficient Global Optimization (EGO) are often employed. EGO searches for the next most valuable sample point by maximizing the Expected Improvement (EI) function:

$$EI(x) = [y\_min - \hat{y}(x)]\ \Phi(\ [y\_min - \hat{y}(x)]\ /\ s(x)\ ) + s(x)\ \varphi(\ [y\_min - \hat{y}(x)]\ /\ s(x)) \tag{12}$$

y_min: The optimal response value at the current sample point.

ŷ(x) and s(x): The predicted value and standard deviation of the surrogate model at point x, respectively.

Φ(x) and φ(x): The cumulative distribution function and probability density function of the standard normal distribution.

Scalability in three-dimensional design spaces is a common challenge faced by all surrogate models. As the number of design variables increases, the number of required training samples increases exponentially, a phenomenon known as the "curse of dimensionality." Traditional surrogate models such as Kriging and RBF gradually fail in high-dimensional spaces. Deep learning models, while theoretically capable of handling high-dimensional data, still require sufficient data to avoid overfitting in practice. These limitations have led researchers to develop methods that more efficiently utilize data resources. Multi-precision neural network (MFDNN) is an effective approach to address this problem.

5. A Review of Multi-Precision Deep Learning (MFDNN) Methods

The fundamental principle of multi-precision modeling is to integrate data resources of varying precision and cost to build a high-precision model while controlling computational cost. This strategy typically relies on a large amount of low-precision data to capture the system's macroscopic characteristics, then uses a small number of high-precision samples to correct local errors, thereby improving overall prediction accuracy.

Currently, commonly used multi-precision machine learning frameworks include co-multi-task learning architectures, deep transfer learning models, and multi-precision deep neural networks (MFDNNs). MFDNNs fuse information by describing the correlation characteristics between data of varying precision. A key step is the introduction of a Gaussian process δ(x) to characterize the differences between high- and low-precision data, thereby achieving precision transfer and model fusion. The core of MFDNNs lies in the concept of residual learning. The network is clearly divided into two parts that work together:

Low-precision network: Responsible for learning the low-precision output f_L(x) from the input design parameters x, aiming to capture macroscopic physical trends.

Residual network: Responsible for learning the residual δ(x) between the high-precision output f_H(x) and the low-precision prediction f_L(x), i.e., δ(x) = f_H(x) - f_L(x), aiming to perform local fine-grained corrections.

Core Mathematical Formula and Architecture

The prediction process of a typical MFDNN can be expressed as follows:

$$\hat{y}\_H(x) \;=\; f\_L(x;\, \theta\_L) \;+\; f\_\delta(x, f\_L(x;\, \theta\_L);\, \theta\_\delta) \tag{13}$$

ŷ_H(x): The model's final prediction of the high-precision output.

f_L(x; θ_L): The output of the low-precision network. It takes the design variables x as input, and θ_L is the network's weight parameter.

f_δ(...; θ_δ): The output of the residual network. It typically takes as input the design variable x and the intermediate layer features (or outputs) of the low-precision network. θ_δ is the residual network's weight parameter. This connection ensures that the residual network can make targeted corrections based on global trends.

The goal of network training is to minimize a loss function, such as the mean squared error loss:

$$L(\theta\_L, \theta\_\delta) \;=\; \Sigma\, [\, y\_H(x\_i) \;-\; \hat{y}\_H(x\_i)\,]^2 \tag{14}$$

The backpropagation algorithm simultaneously optimizes all parameters (θ_L, θ_δ) of the low-precision network and the residual network, ensuring that the final prediction ŷ_H(x) is as close as possible to the true high-precision data y_H(x).

Kennedy & O'Hagan proposed a foundational model in this field, the core formula of which is as follows [12]:

$$High - precision\ output(x) = \rho * low - precision\ output(x) + \delta(x) \quad (15)$$

In this formula, ρ is a constant scaling factor used to calibrate global trends in low-precision data, while δ(x) is a Gaussian process used to nonlinearly model and predict local differences between the low-precision model and the high-precision true value. This structure cleverly utilizes a large amount of low-precision data to capture macroscopic patterns, while using a small amount of high-precision data to fine-tune these patterns.

Multi-task learning improves data utilization efficiency by simultaneously learning related tasks through shared representations; deep transfer learning transfers knowledge learned from one domain (low-precision data) to another domain (high-precision data); and MFDNN achieves end-to-end fusion of low-precision and high-precision data through a specific network architecture.

Compared to single-precision models, MFDNN offers significant advantages: "The proposed MF-DNN is demonstrated on the aerodynamic coefficient prediction of an airfoil. The results show that with only 50 high-fidelity samples and 1000 low-fidelity samples, the MF-DNN achieves comparable accuracy to the single-fidelity model trained with 500 high-fidelity samples, reducing the computational cost by nearly 90%". "Our approach seamlessly integrates low-fidelity and high-fidelity data in a probabilistic fashion and can handle general nonlinear and space-dependent correlations between different fidelities... thus significantly reducing the need for costly high-fidelity simulations." However, the MFDNN approach also has some shortcomings. "The performance of traditional multi-fidelity deep learning models heavily relies on the assumed relationship between data of different fidelities. Their fundamental assumption is that the low-fidelity data can clearly reflect the variation trends of the high-fidelity output" (Chen et al., 2024). When the correlation between low- and high-fidelity data is weak, the predictive performance of the model degrades significantly. Moreover, "these models often require careful design of complex network architectures tailored to specific problems and consume substantial computational resources for hyperparameter tuning, which greatly limits their broad application in practical engineering problems" [13].

6. Conclusion

This study systematically explores the key challenges and feasible solutions faced by aircraft engine nacelle aerodynamic optimization. Through an in-depth analysis of the development process from traditional methods to cutting-edge artificial intelligence technology, we believe that multi-precision deep learning methods have shown significant advantages and application prospects in breaking through the cost limitations of high-precision CFD calculations. The design quality of aircraft engine nacelles directly affects the aerodynamic drag, fuel efficiency and flight safety of aircraft. Industry experts generally believe that the integrated design of nacelles and fuselage is one of the most complex technical problems in current aircraft design. As modern aircraft engines develop towards large bypass ratios, the size of nacelles continues to increase, and the impact of their aerodynamic characteristics on the performance of the entire aircraft is becoming increasingly significant. Although computational fluid dynamics (CFD) methods can provide accurate numerical simulation results, their huge computing resource requirements and the "dimensionality curse" problem encountered in high-dimensional space seriously limit the efficiency of design optimization. This technical bottleneck has become a major obstacle to the progress of aircraft engine nacelle design. In order to overcome the problem of low computational efficiency, researchers have proposed a variety of alternative solutions, such as Kriging interpolation method, neural network proxy model, etc. However, these methods still rely mainly on data input at a single level of precision, failing to effectively resolve the inherent contradiction between computational cost and prediction accuracy. Although deep learning technology has powerful nonlinear fitting capabilities, its training process requires a large amount of high-quality data support. This feature has largely limited its promotion and application in engineering practice. Multi-precision deep learning provides a new technical path to solve the above problems by integrating data sources of different precision levels. The core idea of this method is to make full use of a large amount of low-precision data and a small amount of high-precision data to construct a hybrid model with high prediction accuracy. Experimental data shows that this method only requires 50 high-precision samples combined with 1,000 low-precision samples to achieve the prediction accuracy of 500 high-precision samples of the traditional method, and the computational cost is reduced by nearly 90%. Compared with traditional single-precision models, multi-precision deep learning has three outstanding advantages: significantly reducing dependence on expensive high-precision data; showing better adaptability in high-dimensional parameter space; and improving the prediction stability of the model through multi-source information fusion. Of course, this method also has limitations such as complex model structure, sensitivity to data quality, and difficulty in parameter debugging [5], which need to be focused on in subsequent research. The emergence of multi-precision deep learning technology provides an effective solution for overcoming the computational limitations of high-fidelity numerical simulations, laying a solid foundation for efficient and accurate aerodynamic optimization design. This approach opens up new technical avenues for the intelligent, integrated optimization design of aircraft components such as engine nacelles.

References

[1] CICTT (General Aviation Accident and Incident Classification Team). Statistics of air accident.

[2] Oceanic operations an authoritative guide to oceanic operations. (1994). U.S. Dept. of Transportation, Federal Aviation Administration, FAA Flight Standards Service.

[3] Slotnick, J., Langley Research Center, issuing body, & United States. National Aeronautics and Space Administration, sponsoring body. (2014). CFD vision 2030 study: a path to revolutionary computational aerosciences. National Aeronautics and Space Administration, Langley Research Center.

[4] Li, J., Gao, Z., Huang, J., & Zhao, K. (2013). Aerodynamic design optimization of nacelle/pylon position on an aircraft. Chinese Journal of Aeronautics, 26(4), 850–857.

[5] NASA. Inlet Design.

[6] Haines, A. B. (2000). [Rev. of Intake Aerodynamics — Second edition. J. Seddon and E.L. Goldsmith. Blackwell Science, Osney Mead, Oxford OX2 0EL, UK. 1999. 1407pp. Illustrated. £59.50. ISBN 0-632-04963-4]. Aeronautical Journal, 104(1032), 96–96.

[7] NASA. Inlet Design.

[8] Slotnick, J., Khodadoust, A., Alonso, J., Darmofal, D., Gropp, W., Lurie, E., & Mavriplis, D. (2014). CFD Vision 2030 Study: A Path to Revolutionary Computational Aerosciences. In NASA Center for AeroSpace Information (CASI). Reports. NASA/Langley Research Center.

[9] Mani, M., Rider, B., & Lunk, M. (2013). Nacelle design and analysis for an N3-X hybrid wing body configuration. In 31st AIAA Applied Aerodynamics Conference. American Institute of Aeronautics and Astronautics.

[10] Zhang, Y., Chen, H., Fu, S., Zhang, M., & Zhang, M. (2015). Drag prediction method of powered-on civil aircraft based on thrust drag bookkeeping. Chinese Journal of Aeronautics, 28(4), 1023–1033.

[11] Ferziger, J. H. (2019). Computational Methods for Fluid Dynamics (R. L. Street & M. Perić, Eds.; Fourth edition.). Springer Nature.

[12] Tuo, R., & Wu, C. F. J. (2018). PREDICTION BASED ON THE KENNEDY-O’HAGAN CALIBRATION MODEL: ASYMPTOTIC CONSISTENCY AND OTHER PROPERTIES. Statistica Sinica, 28(2), 743–759.

[13] Baining CHEN, Fangfang XIE, & Xuhui MENG. (2024). Adaptive Multi-Fidelity Composite Deep Neural Networks. PHYSICS OF GASES, 9(4), 1–8.