Rhine-Waal University of Applied Sciences

Advanced Modeling & Simulation

Final Report

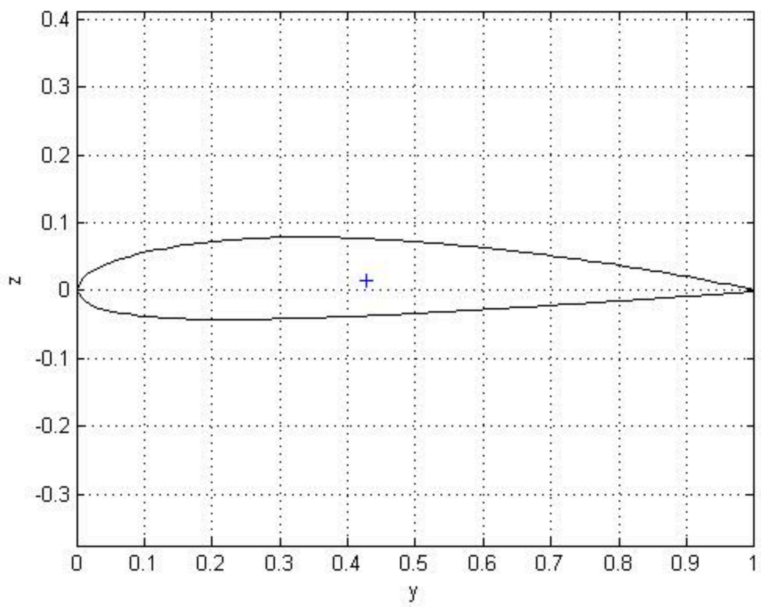
July 1, 2023

1.Abstract

This research paper aims to explore the optimisation of wing design for enhanced aerodynamic performance by utilising differential equations. Aerodynamic performance plays a crucial role in various fields, including aviation, aerospace engineering, and wind energy. By applying differential equations, we can mathematically model the flow of air around a wing, enabling us to analyse and optimise its design parameters. This paper will present an overview of the fundamental principles of aerodynamics, introduce the concept of differential equations in relation to aerodynamic modelling, discuss the key parameters affecting wing performance, and outline the optimisation techniques used to improve aerodynamic efficiency.

2.Introduction

This research aims to optimise the NACA2412 airfoil by exploring various parameters to improve its performance and reduce drag. The NACA2412 airfoil, known for its favourable lift and drag characteristics, offers potential for further enhancement through optimisation techniques. The study focuses on design parameters, including the camber line, maximum thickness location, and local geometry variations. Computational methods, such as potential flow theory and computational fluid dynamics, will be employed to simulate and analyse the airfoil's aerodynamics. The optimisation process involves formulating the problem, conducting numerical simulations, applying optimisation algorithms, validating the results, and analysing the impact of the optimised parameters on lift, drag, and stability characteristics. The research outcomes will contribute to advancements in airfoil design, leading to improved performance, reduced drag, and increased efficiency in various applications.

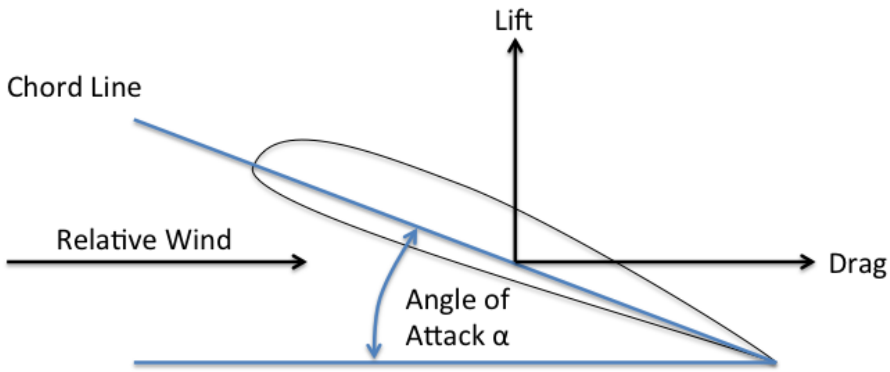


3.Aerodynamic principles

Understanding the fundamentals of aerodynamics entails being able to distinguish between basic aerodynamic variables like pressure, density, velocity, and temperature as well as elements like acting forces, flow types, airfoil design, boundary layers, and more. We'll talk about these essential ideas and how crucial they are to construct a safe airplane in this essay.

The first step to understanding aerodynamic properties is to have knowledge about the forces acting on a body. The main aerodynamic forces of flight include:

- **Lift**: Lift is generated when the fluid and airfoil interact with each other, with either of them being in motion. A force is generated that acts against the weight of the aircraft, counterbalancing to hold the plane in the air.
- **Drag**: Drag is the resisting force that works against the aerodynamic motion or thrust. For the airplane in motion, the air resists the forward motion due to the opposing drag force, reducing the velocity.
- **Thrust**: Thrust is the forward propelling force in the aircraft, which acts against the resistance of drag. Constant thrust maintains constant airplane speed while its increase or decrease may be required during lift-off or landing. Jet engines and propellers in aircraft can create thrust.
- **Weight**: Weight can be expressed as the force of gravity pulling the craft towards the Earth. Thus, it is always directed downwards from the center of mass of the airplane. Lift, the opposing force to weight, is required to enable flight. During flight, weight can constantly change, affecting the balance of the aircraft. Thus, constant control is always required.



4.Fluid Mechanics

In **C**omputational **F**luid **D**ynamics, the governing equations of fluid dynamics, such as the Navier-Stokes equations, are discretised using techniques like the finite volume method. The computational domain is divided into a grid or mesh of smaller control volumes. The fluid properties and flow variables, such as velocity, pressure, and temperature, are approximated at each control volume. The discretised equations are then solved iteratively on a computer to obtain the numerical solution.
To apply (CFD) to study the aerodynamic behaviour of a NACA 2412 airfoil, several steps are involved:

- **Geometry Definition:** The shape, dimensions, and associated components of the airfoil are defined using computer-aided design (CAD) software or by specifying the airfoil parameters.
- **Mesh Generation:** A grid or mesh is created around the airfoil, dividing the computational domain into smaller control volumes. The mesh should have appropriate resolution to accurately capture flow features without being overly dense in order to manage computational costs.
- **Boundary Conditions:** The inflow, outflow, and wall conditions for the simulation are specified. This includes determining the velocity and pressure conditions at the inflow and outflow boundaries, as well as the no-slip condition at the airfoil surface.
- **Numerical Solution:** The discretised equations governing fluid flow are solved using numerical algorithms. These algorithms calculate the flow variables (velocity, pressure, etc.) at each control volume based on the defined equations and boundary conditions.
- **Post-Processing:** The obtained numerical solution is analysed to gain insights. This involves visualising flow patterns, calculating aerodynamic forces (such as lift and drag), and examining pressure distributions over the airfoil surface.

For instance, in simulating the flow over a NACA 2412 airfoil using CFD, the airfoil's geometry and relevant parameters (e.g., chord length, camber) are defined. A mesh is then generated around the airfoil, with higher resolution near the surface, leading, and trailing edges. Boundary conditions are set, including specifying the inflow velocity and the no-slip condition at the airfoil surface. The numerical solution is obtained by iteratively solving the discretised equations using suitable algorithms. Finally, post-processing involves analysing the flow field, examining pressure distribution on the airfoil surface, and calculating aerodynamic forces like lift and drag.

5.Differential Equations in Aerodynamic Modeling

The Navier-Stokes equations are a set of fundamental equations in fluid dynamics that describe the motion of fluid substances, taking into account the effects of viscosity and compressibility. They are derived from the application of Newton's second law of motion to a fluid element.

The Navier-Stokes equations consist of three main equations: the continuity equation and the momentum equations in each of the three spatial dimensions (x, y, and z). Let's break them down:

- Continuity Equation:
- The continuity equation expresses the conservation of mass for an incompressible fluid. It states that the rate of change of mass within a control volume is equal to the net rate of mass flow across its boundaries. In mathematical form, the continuity equation can be written as:

$$\partial\rho/\partial t + \nabla\cdot(\rho V) = 0$$

where ρ is the fluid density, t is time, V is the velocity vector, and $\nabla\cdot$ represents the divergence operator.

- Momentum Equations:
- The momentum equations represent the conservation of momentum for each of the three spatial dimensions. They take into account the effects of pressure, viscous forces, and body forces (such as gravity). The momentum equations can be expressed as:

$$\partial(\rho V)/\partial t + \nabla\cdot(\rho VV) = -\nabla P + \nabla\cdot\tau + \rho g$$

where P is the pressure, τ is the stress tensor representing viscous forces, and g is the acceleration due to gravity.

The stress tensor τ accounts for the effects of viscosity and is dependent on the fluid's rate of deformation. The precise form of the stress tensor depends on the chosen constitutive model, such as the Newtonian or non-Newtonian behaviour of the fluid.

Solving the Navier-Stokes equations analytically for most practical fluid flow problems is often challenging or even impossible due to their complexity. Therefore, numerical methods, such as finite difference, finite element, or finite volume methods, are typically employed to obtain approximate solutions through computational fluid dynamics (CFD) simulations.

CFD techniques discrete the fluid domain into smaller control volumes and solve the Navier-Stokes equations iteratively in a numerical manner to obtain the flow field variables (velocity, pressure, etc.) at discrete locations within the domain. These simulations allow for the prediction and analysis of fluid behaviour in various aerodynamic scenarios, aiding in the design and optimisation of aircraft, vehicles, and other aerodynamic systems.

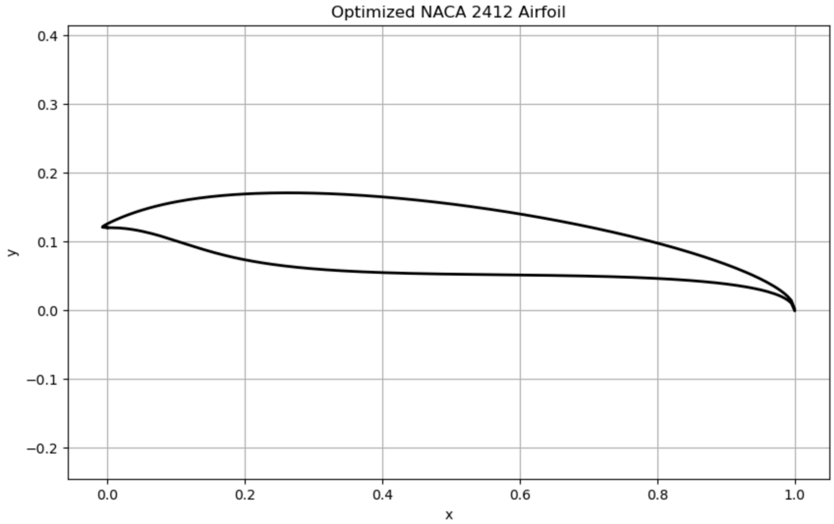
6.Optimisation Techniques:

In the context of optimising the NACA 2412 airfoil using genetic algorithms, here's an explanation of how the different components of a genetic algorithm can be applied:

- **Initialisation**: Begin by creating an initial population of potential airfoil shapes for the NACA 2412. Each individual in the population represents a unique airfoil shape and is encoded as a set of genes or chromosomes, which could be the coordinates defining the airfoil's shape.
- **Fitness Evaluation**: Evaluate the fitness of each individual in the population by calculating an objective function that represents the desired optimisation criteria. For the NACA 2412 airfoil, this objective function could be based on the lift-to-drag ratio, which measures the trade-off between lift (desired) and drag (undesired) forces. You can use CFD simulations or other analysis methods to compute the lift and drag coefficients for each airfoil shape.
- **Selection**: Select individuals from the current population based on their fitness. The selection process favours individuals with higher fitness values, as they are more likely to contribute to the next generation. Common selection methods include roulette wheel selection, tournament selection, or rank-based selection.
- **Reproduction**: Create offspring for the next generation through genetic operators, such as crossover and mutation. Crossover involves combining genetic information from two parent airfoil shapes to create new offspring with a mix of their characteristics. Mutation introduces small random changes to the genetic information, allowing for exploration of the search space. In the case of airfoil shapes, crossover can be applied to the coordinates defining the shape, and mutation can introduce slight modifications to these coordinates.
- **Replacement**: Combine the offspring with some individuals from the previous generation to form the population for the next generation. The size of the population typically remains constant throughout the optimisation process.
- **Termination**: The process of selection, reproduction, and replacement continues iteratively for a specified number of generations or until a termination condition is met. Termination conditions can include reaching a satisfactory fitness level, convergence of the population, or a maximum number of generations.

By repeating the selection, reproduction, and replacement steps, genetic algorithms explore the design space of the NACA 2412 airfoil, gradually improving the fitness of the individuals and converging towards an optimal or near-optimal airfoil shape with an improved lift-to-drag ratio.

It's important to note that the implementation of genetic algorithms for airfoil optimisation involves considerations specific to the NACA 2412 airfoil, such as the encoding of the airfoil coordinates, the definition of the objective function based on the lift-to-drag ratio, and the selection of appropriate genetic operators.



The effectiveness of the genetic algorithm depends on the proper design of these components and careful parameter tuning, such as the population size, crossover and mutation rates, and termination criteria. It took several iterations and evaluations to achieve this desirable results.

7.Conclusion

The primary objective of this research paper is to leverage the intrinsic capabilities of differential equations as a powerful tool in investigating and optimising wing design parameters for the purpose of achieving heightened levels of aerodynamic performance. By employing sophisticated mathematical modelling techniques, in conjunction with state-of-the-art optimisation methodologies, this study aims to undertake a comprehensive exploration of an extensive array of design configurations. Through this integrative approach, an in-depth understanding of the intricate interdependencies among various aspects, including wing geometry, flow physics, and performance metrics, can be attained. As a result, this research endeavour not only propels the field of aerospace engineering forward but also assumes a critical role in fostering the development of aircraft designs that transcend conventional standards by embodying enhanced efficiency while staying steadfastly committed to the overarching principles of sustainability and environmental stewardship.