
Scienxt Journal of Mechanical Engineering & Technology
2023; Volume-1; Issue-1, pp. 37- 46

# *Study of aerodynamics and fluid mechanics*

**Mahesh Bharatadri[1], Lokesh Kumar[2]**

*[1] *Professor, Department of Mechanical Engineering, Dr. Ambedkar Institute of Technology, Bangalore, India*

*[2] *Department of Mechanical Engineering, Dr. Ambedkar Institute of Technology, Bangalore, India*
*E-mail: lokesh.sira87@gmail.com*

***https://zenodo.org/deposit/8054577***

*Corresponding author: Lokesh Kumar

## Table of Contents



## List of Figures

## Abstract:

Computational aerodynamics is an increasingly important field in the field of turbo-machinery as well as the aerospace industry. In this part, we will highlight the range of complexity that is involved in fluid simulations by providing a few instances of simulations that involve fluids. The application of computational fluid dynamics, sometimes known as CFD, is an essential part of the design process in contemporary manufacturing (CFD). Given this context, we investigate the possible functions and markets for a given entity to a certain part in a period where portable computing is everywhere.

## Keywords:

Aerodynamics, Computational Aerodynamics, Computational Fluid Dynamics (CFD), Turbomachinery.

# 1. Introduction:

Fluid dynamics is concerned with the movement of fluids and their interaction with the flow's boundary conditions. Fluids can be gases or liquids, like water in seas or air in the atmosphere. Aeronautics, mechanics, thermodynamics, chemistry, and physics all deal with fluid flow in some way. It is necessary to have a thorough understanding of fluid dynamics when creating complex technical devices like pipeline systems, airplanes, and heat exchangers.

## 1.1. Background of the study

It is a branch of physics that deals with the movement of objects in the air, and it is called aerodynamics. As Icarus and Daedalus show, people have been engaged in aerodynamics and flight for the thousands of years, but only in the last hundred years have we been capable to fly in a heavier-than-air contraption. There are several single-phase (gas or liquid) fluid flows that can be modelled using the Navier Stokes equations. Once the terms explaining vicious actions have been removed, the equations that result are known as the Euler equations. The maximum ability equations can be obtained by omitting terms expressing fluid velocity from the equations. Linearization of these equations is also possible in the case of modest perturbations in the flow (i.e., in the supersonic ranges and subsonic, but not transonic or hypersonic).

## 1.2. Aim of the study

All aspects of fluid dynamics research related to aerodynamics, turbulence, hydrodynamics, and process engineering are included in this category (for example, multiphase flows).

Aerodynamics and Fluid dynamics are critical to a wide range of fields, including aerospace, complicated fluids, microfluidics, and developing technologies like wind power.

Research in the fields of aerodynamics and fluid dynamics is critical to improving aviation and transportation sustainability. Cooperating with other disciplines, such as those in Urban Systems and Infrastructure and Materials engineering, can have a greater impact on the transition of fundamental understanding into solutions.

## 1.3. Objectives

1. Predicting forces and moments on a body in motion through a liquid
2. Predicting heat transmission to moving entities in a fluid.
3. Flow and heat transmission in ducts can be studied using this method.

## 1.4. Research questions

Several questions can be found under the headings of aerodynamics and fluid mechanics.

1 Do other factors besides gravity have the potential to affect how air moves across a space?
2. What are some of the challenges that come with studying fluid mechanics?
3. What are the three most important facets of fluid mechanics?

# 2. Literature Review:

## 2.1. Introduction:

Aerodynamics is a discipline of physics that studies the flow of air or other gaseous liquids and the forces exerted on objects traveling through such a fluid. Aerodynamics focuses on the mechanics of airborne objects including planes, rockets, and missiles. As well as designing and building constructions such as tall buildings and bridges that can withstand strong winds, this field also looks at fast trains, autos, and ships (Tsiolkovsky, 2022).

Numerical analysis and data formats are used in Computational Fluid Dynamics (CFD) is to examine and solve problems through involving fluid flows. To simulate the fluid's free-stream flow and interactions with surfaces determined by boundary conditions, computers are utilized (Tsiolkovsky, 2022). Aerodynamics and aerospace analysis, hypersonic, weather simulation, industrial system design and analysis, fluid flows, biological engineering, environmental engineering, natural science and heat transfer, engine and visual effects for combustion analysis, films, and games all use CFD in some way. CFD is also used in many other fields of study and industry (Tsiolkovsky, 2022).

## 2.2. Computational aerodynamics:

Even while high-resolution approaches for compressible flow were being developed and used by researchers, the aerodynamics community relied on simpler flow models, such as variants of the prospective flow equation for irrotational flow, and home tropic. The Panair code of Aircraft used panel methods to deal with the incompressible potential flow, while the small-disturbance equations still contained Mach-number implications for supersonic flows and subsonic flows. The equations for transonic flow, on the other hand, are nonlinear (Cummings, 2012).

It is difficult to overstate Jameson's impact on aeronautical CFD. Beginning in the late 1960s, he contributed a series of aerodynamic codes for ever more complex flow models, from tiny disturbances to full Navier-Stokes. Euler code FLO57 and also its inter version FLO87 were the most significant in convincing the aerospace world that the complete Euler flow model should be used. Joe Steger, Woodrow Whitlow, Mohammed Hafez, Earll Murman, and Wen-hui ou were just a few of the key scientists that worked for Flow Research Corp. in the establishment of CFD in the aerospace industry. For their work on the first transonic flow code using the complete potential equation, Whitlow and Hafez are known (Chen, 2017). After the Euler equations replaced Hafez's entire potential equation, he continued working on a non-isentropic correction for the transonic shock position (Ceze and Murman, 2016).

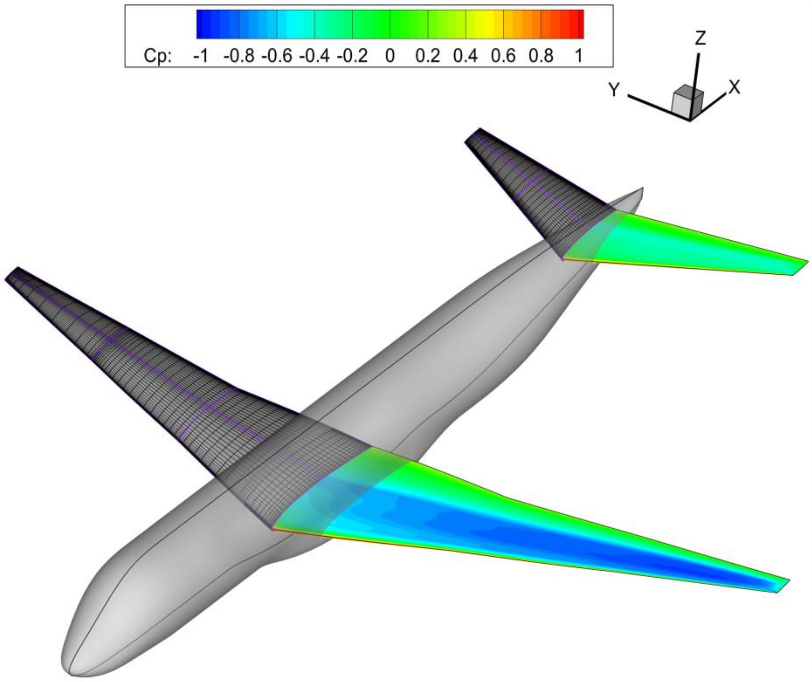


***Figure.1: Computational Aerodynamics [Source: (Zhao, Sheng, and Afjeh, 2014)]***

## 2.3 Computational fluid dynamics (CFD)

Computational Fluid Dynamics (CFD) is use of computer simulations to analyseunder systems including heat transport, fluid flow, and other phenomena like chemical reactions. CFD is sometimes abbreviated as CFD and is a branch of fluid mechanics algorithms and methods to solve and analyze fluid flow issues. It is necessary to use computers to execute the computations necessary to model the fluid surface interaction as determined by boundary conditions. Better and faster solutions are found using high-speed supercomputers. The

Navier–Stokes equations are the foundation of virtually all CFD issues. A single-phase fluid flow can be described in these terms for nearly any flow of fluids. Certain constants that define viscosity can be removed from the Navier–Stokes equations, leading to the Euler equations, which simplify the equations. Removing the parameters that describe vorticity simplifies the equations even further, resulting in the complete group of possible equations (Wu, 2012). After the equations have been simplified, they can be linearized to get the linear system potential equations after linearization.

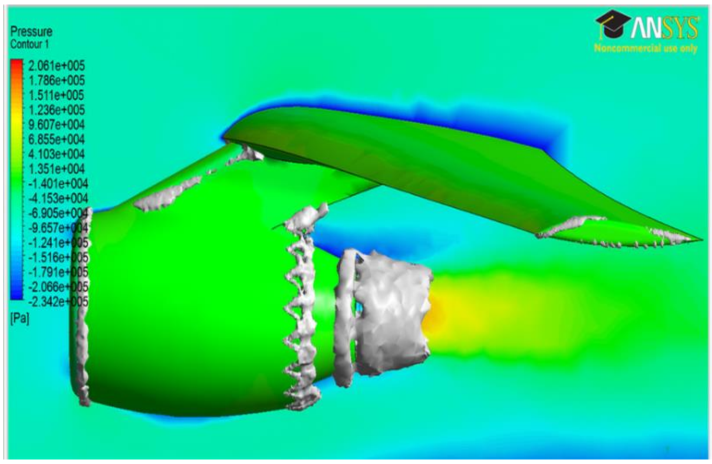


*Figure.2: Computational Fluid Dynamics [Source: (Thabet and Thabit, 2018)]*

## 3. Methodology:

### 3.1 Data analysis - Aerodynamics Analysis Involves CFD:

The use of fluid mechanic equations to research aerodynamics is a useful approach. In addition, the quality of the results is reliant on the CFD tools you choose. Here, a visual representation of the airflow-induced vorticity surrounding the aircraft will be shown in fig. 3.

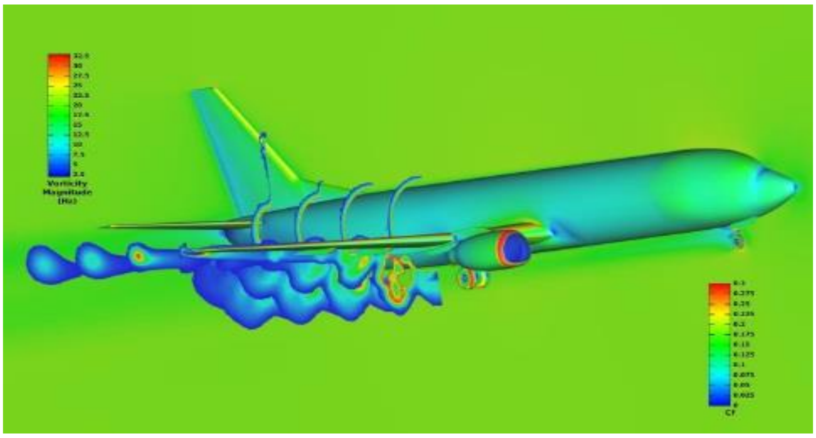

*Figure.3: Aerodynamics and Fluid Mechanics model [Source: (Cai, Kolomenskiy, Nakata and Liu, 2021)]*

If you want to know if vortex shedding is occurring in your system, you can use the graphic above to get an idea of what kind of analysis capabilities you'll need. When it comes to studying aerodynamics for your system design, Omnis from Cadence is a researcher who is confident in the CFD analysis tool (Ghoreyshi, Jirásek, and Cummings, 2014).

## 3.2 Data design:

The trade-off between possible benefits and expenses is what leads aerospace businesses to utilize CFD in the first place. Although the advantages are widely acknowledged, the design process cannot be swamped by computing costs. Setup and turnaround times must also be taken into consideration. There are three stages of the design process: conceptual, preliminary, and final detailed. Typically, a team of 15-30 engineers works on the conceptual design phase, which defines the objective given anticipated market needs and establishes a rough preliminary configuration, along with early estimates of size, weight, and performance. The total cost of this phase is between $6 and $12 million. Initial design progresses to the point when detailed performance predictions can be generated and assured to potential customers, allowing them to officially execute contractual agreements for the purchase of a fixed number of planes. For 60-120 million dollars, engineering staffs of 100 to300 peoples is usually working for up to two years. Computational simulations and wind tunnel tests are used to investigate the initial aerodynamic performance. Decisions made during the design largely impact the final performance as well as the development costs, even though they are still modest.


*Figure.4: Phases of Design [Source: Self-created]*

To enable a smooth transition from idea to production, the final design must include the structure, flight deck, control systems, electrical systems, avionics, and weapon systems for military planes, and the interior layout for commercial aircraft. At this time, major costs are incurred and a manufacturing strategy is developed. A single airplane has tens of thousands of engineers. The total cost is $3-10 billion. The final design is only adopted if enough orders are received to recuperate some of the initial cost. Early in commercial aerodynamics, aircraft design determines the external shape. Boeing 777 aerodynamic lines were stopped after initial orders, blocking additional design development. First, CAD is defined from conceptual design. From the outer multi-disciplinary loop comprises aerodynamic exploration and wind tunnel testing. Three six-month phases conclude wing design. CFD advancements could cut design time and costs. CFD's systematic application can increase the final design's performance. Lifts to drag (L/D) must be enhanced by 5 percent to reduce fuel usage. A 5% fuel savings over 25 years for a fleet of 500 long-range aircraft would save $5 billion or $10 million. A smaller aircraft may perform the same work as a larger one, reducing initial and operating costs. Even a small performance advantage can have a big impact on a $1 trillion market over the next few decades.

## 4. Findings and Discussion:

Research in the fields of aerodynamics and fluid dynamics is currently being carried out by several smaller groups, some of which may or may not be collaborating to find solutions to the most fundamental problems facing the sector. Outside of EPSRC, the greater research landscape is something that needs to be taken into consideration.

## 5. Conclusion:

Aerodynamics examines air and gaseous fluid dynamics and forces on moving objects. It explains how rockets, aircraft, and missiles fly. It also evaluates the wind resistance of cars, trains, ships, bridges, and tall buildings. Aerodynamics studies the forces that affect gas and particle movement. Gas particle speed and pressure change in the availability of a body, causing lateral force and lift resistance. Fluid mechanics problems are common in engineering and applied physics. Studying fluid behavior involves statics, kinematics, and dynamics. Fluids deform forever and respond fast to disturbances. Static fluid seems to be the type of fluid studied in fluid statics. Moving fluid is moving fluid. The ES and Cray X1 are expensive massively parallel vector supercomputers. Other techniques, like Blue planets virtual vector architecture and "system-on-a-chip" computers like Blue Gene/L, are being investigated to achieve cheaper performance.

## 6. References:


1. Cai, X., Kolomenskiy, D., Nakata, T. and Liu, H., 2021. A CFD data-driven aerodynamic model for fast and precise prediction of flapping aerodynamics in various flight velocities. Journal of Fluid Mechanics, 915.
2. Ceze, M. and Murman, S., 2016. Global convergence strategies for a spectral-element space-time discontinuous-Galerkin discretization of the Navier Stokes– equations. International Journal of Computational Fluid Dynamics, 30(6), pp.444-449.
3. Chen, Q., 2017. Error analysis of staggered finite difference finite volume schemes on unstructured meshes. Numerical Methods for Partial Differential Equations, 33(4), pp.1159-1182.
4. Cummings, R., 2012. Computational aerodynamics education at the US Air Force Academy. International Journal of Aerodynamics, 2(1), p.93.
5. Ghoreyshi, M., Jirásek, A. and Cummings, R., 2014. Reduced order unsteady aerodynamic modelling for stability and control analysis using computational fluid dynamics. Progress in Aerospace Sciences, 71, pp.167-217.
6. Thabet, S. and Thabit, T., 2018. Computational Fluid Dynamics: Science of the Future. International Journal of Research and Engineering, 5(6), pp.430-433.
7. Tsiolkovsky, K., 2022. Aerodynamics | fluid mechanics. [online] Encyclopaedia Britannica. Available at: <https://www.britannica.com/science/aerodynamics> [Accessed 10 June 2022].
8. Wu, B., 2012. Computational Fluid Dynamics Study of Large-Scale Mixing Systems with Side-Entering Impellers. Engineering Applications of Computational Fluid Mechanics, 6(1), pp.123-133.
9. Zhao, Q., Sheng, C. and Afjeh, A., 2014. Computational Aerodynamic Analysis of Offshore Upwind and Downwind Turbines. Journal of Aerodynamics, 2014, pp.1-13.



***Cite as***

*Mahesh Bharatadri1, Lokesh Kumar2. (2023). Study of Aerodynamics and Fluid Mechanics.*

*https://doi.org/10.5281/zenodo.8054577*