iJRASET
International Journal For Research in Applied Science and Engineering Technology



INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 11 **Issue:** V **Month of publication:** May 2023

DOI: https://doi.org/10.22214/ijraset.2023.53198

www.ijraset.com

Call: 08813907089 | E-mail ID: ijraset@gmail.com

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 11 Issue V May 2023- Available at www.ijraset.com

Development of Application for Airfoil Generation and Testing

Yash Choudhari[1], Mandar Karande[2], Shreeram Bhoite[3], Mrugendra Gade[4]
[1, 2, 3, 4]*Walchand Institute Of Technology, Punyashlok Ahilyadevi Holkar Solapur University, Solapur, Maharashtra, India*

Abstract: ***CAD automation works by utilizing tools and technology to increase the speed at which a design is completed. The CAD workflows that you set up can be carried out over and over again much like any other repetitive task in your office. This allows for multiple designs to be completed with reduced effort, freeing up valuable time for more important tasks and creativity. The application devloped automates the processes required to create an 3D part file of any airfoil.The application also provides a script to automate the next step i.e CFD analysis of the airfoil. The application is divided in 4 major parts according to the task automated. In the first section , the user can enter the parameters of desired airfoil,these parameters are then utilized by the software to create coordinates of the points that lie on airfoil. After this a curve is fitted to the airfoil points and the coefficients of the fitted curve are displayed and stored. In the section , the coefficients generated in the first section are utilized to generate CAD files of the airfoil. In the third section the software provides a python script that automates the task of carrying out the CFD simulation of airfoil. Lastly the "results" section provides a workspace to analyze the output of CFD simulation. Thus the application acts as a complete workstation for design life cycle of the airfoil.***
Keywords: Airfoil, Automation, CFD, python script, Windows MFC, Autodesk Inventor.

I. INTRODUCTION

In engineering scenarios, one may encounter complex design situations. Previously, designs were drawn on sheets and manufactured manually using old school techniques. But now, things have changed with the introduction of CAD customization and design automation.

1) CAD customization is the development of support tools and technology which drives CAD automation of repetitive tasks in the design process.
2) Design automation is a knowledge-based engineering approach which logically combines various engineering concepts with real time application study during product development.

In this project, design automation of airfoils has been carried out. CAD Automation applications are developed as add-ons over CAD platforms (like AutoCAD, Revit, Inventor, SolidWorks, Creo, etc). Mundane tasks that require rule-based decision making are ideal for automation. These applications intelligently extract data, apply rules, make decisions, and perform operations automatically. In this project we have used Autodesk Inventor API to provide design automation. The next functionality i.e automation of CFD simulation utilizes the python scripting functionality provided by Autodesk CFD. The application has been developed using C++ language with the help of Microsoft's MFC library.

A. Project Identification

CAD integration enables engineers to create 3D designs instantly, removing the need for engineering consultation. In addition, CAD integration lets customers see a 3D model of their product without requiring the engineer's input. This advancement is the next step in the manufacturing industry's evolution. Automating the design process will allow companies to move the configuration process along faster and ensure the products are appropriately designed. And, with today's faster processors and improved graphics, CAD automation is becoming more popular. Currently there is no software that provides automation facility for airfoil generation in CAD ecosystem. The application developed would reduce the time required for product design and analysis and streamline the entire process.

B. Project Objectives

1) To develop a software that can automate CAD file generation of airfoil.
2) To automate the CFD simulation of airfoil.
3) To carry out data extraction from the automated CFD simulation.

©IJRASET: All Rights are Reserved

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 11 Issue V May 2023- Available at www.ijraset.com

C. Scope Of Work

1) Visualization of airfoil based on its parameters.
2) Rapid prototyping.
3) Carry out a complete design study of an airfoil
4) Select the optimum operating condition for a particular airfoil.

II. LITERATURE REVIEW

In the earlier chapter we looked at CAD automation, its benefits, challenges and its applications. In this chapter we will take a deep dive into the technologies used for developing the project application. Their origin, current state and usage is discussed.

A. Microsoft Foundation Class Library (MFC)

Microsoft Foundation Class Library (MFC) is a C++ object-oriented library for developing desktop applications for Windows. MFC was introduced by Microsoft in 1992 and quickly gained widespread use. While Microsoft has introduced alternative application frameworks since then, MFC remains widely used. MFC was introduced in 1992 with Microsoft's C/C++ 7.0 compiler for use with 16-bit versions of Windows as an extremely thin object-oriented C++ wrapper for the Windows API. C++ was just beginning to replace C for development of commercial application software at the time. In an MFC program, direct Windows API calls are rarely needed. Instead, programs create objects from Microsoft Foundation Class classes and call member functions belonging to those objects. Many of those functions share their names with corresponding API functions.

B. Autodesk Inventor API

API, or Application Programming Interface, is a term used to describe the functionality exposed by an application that allows it to be used through a program.

For example, you can use Inventor's API to write a program that will perform the same types of operations you can perform when using Inventor interactively. Having an API is important because it allows you to add functionality to Inventor that is specific to your needs. Inventor, by necessity, is a general CAD system, meaning that it's not aimed at any specific industry or used to model only certain types of products.

By providing an API, Inventor allows you to add additional functionality and optimize repetitive operations to make it more productive for your individual needs. By providing an API, Inventor also provides you the ability to better integrate Inventor into your overall Enterprise process.

For example, you might write a program that interfaces with your company's inventory database to obtain the current price for components so that the price shown in a part list is always up-to-date. You might also write a program that extracts data from assemblies to provide MRP systems with their required information. All of this can be done manually, but by automating it using a program you're able to significantly increase productivity and minimize errors.

C. Autodesk CFD API

The Autodesk CFD Application Programming Interface (API) provides ways to work with Autodesk R CFD functionality that are not readily available in the user interface. The API is a platform for customization, and can significantly benefit your design process. It is very flexible, and can be used for a wide variety of tasks.

Here are a few examples of potential applications:

1) Automate repetitive tasks normally performed in the user interface.
2) Create custom tasks.
3) Create custom results quantities.
4) Output results in customized or specialized formats.

The API is based on the Python scripting language. Python has been widely accepted, and is both easy to use and flexible. Python has built-in connections with other technologies, including Microsoft Office tools such as Excel, that make it an ideal "glue language." Autodesk CFD is distributed with Python version 2.7.

©IJRASET: All Rights are Reserved | SJ Impact Factor 7.538 | ISRA Journal Impact Factor 7.894 |

III. METHODOLODY

The development of application has been divided into four parts according to the four major components of the application.
The four individual components are -

1) Sandbox - Allows 2D visualization of airfoil and generates parameters for CAD automation.
2) CAD Automation - Generates 15 CAD files of the airfoils with AoA ranging from 0 to 14 degrees.
3) CFD Automation - Creates 15 design studies for each individual airfoils and runs simulations for each one.
4) Result Interpreter - Provides functionality to interpret the data obtained from CFD simulation. Helps in generating insights from acquired data .

A. *SANDBOX*

The sandbox environment allows for rapid prototyping of airfoil. The curve fitting and subsequent parameter generation takes place in this component. The component also displays the accuracy with which the curve mimics the airfoil curvature.

1) *Designing a NACA 4-digit airfoil*

- *NACA 4 digit airfoil specification*

This NACA airfoil series is controlled by 4 digits e.g. NACA 2412, which designate the camber,position of the maximum camber and thickness. If an airfoil number is

NACA MPXX
e.g.
NACA 2412

then,
• M is the maximum camber divided by 100. In the example M=2 so the camber is 0.02 or 2
• P is the position of the maximum camber divided by 10. In the example P=4 so the maximum camber is at 0.4 or 40
• XX is the thickness divided by 100. In the example XX=12 so the thiickness is 0.12 or 12

2) *Curve Fitting*

In the previous step we found out the coordinates of points that lie on the airfoil. In this step the coefficients of a curve that passes through this points is calculated. The method used for calculation of coefficients is least square polynomial fitting
Mathematics of least square polynomial fitting
Consider a set of n data points

$$\{t_i, v_i\} \in \Re$$

In such a case, a polynomial fit of order k can be written as:

$$v^* = a_0 \cdot t + a_1 \cdot t^2 + a_2 \cdot t^3 + \ldots + a_k \cdot t^k$$

Residual in this case is given by

$$R^2 = \sum_{i=1}^{n} \left[v_i^* - a_0 \cdot t_i + a_1 \cdot t_i^2 + a_2 \cdot t_i^3 + \ldots + a_k \cdot t_i^k \right]^2$$

The objective of the least-square polynomial fitting is to minimize R2. The usual approach is to take the partial derivative of Equation 2 with respect to coefficients a and equate to zero. This leads to a system of k equations. Such a system of equations comes out as Vandermonde matrix equations. In matrix notation, we can write as

$$V = T \cdot a$$

Above equation can be solved by pre-multiplying by the transpose of T, thus the solution vector
comes out to be,

$$a = (T^T \cdot T)^{-1} \cdot T^T \cdot V$$

B. CAD Automation

This component of the application automates the steps that are required for creating an extruded profile of an airfoil. Autodesk Inventor API was selected for this task, because it has wide user base, and a complete library of functions that can automate any task needed.

The tasks that are automated in the order in which they execute are -

1) Importing parameters of airfoil from a text file and saving it in the current autodesk environment.
2) Creation of "sketch" object.
3) Creation of the upper and lower curve of airfoil from the parameters that were imported earlier.
4) Closing the sketch loop and adding fillets and other end effects required to obtain a sketch profile that can be extruded.
5) Providing an angle of attack to the airfoil profile created in previous step.
6) Extruding the profile.
7) Creation and extrusion of an enclosure profile that will act as fluid domain for the airfoil. This is utilized in the next component i.e CFD Automation.
8) Saving the file in project directory.

Above steps are repeated a total of 15 times to create 15 CAD files of airfoil ranging from AoA of 0o to AoA of 14o.
In order to initiate the execution of the above steps a button labelled "Generate Airfoil" is created and added to tools tab of Autodesk Inventor. By clicking on the "Generate Airfoil" button the above steps are initiated.

C. CFD Automation

This component of the application deals with automating the general steps that are undertaken in a CFD analysis. Autodesk CFD is used for automation, since it has a built-in facility to run python scripts.
The tasks that are automated in the order in which they execute are -

1) Creating a new design study.
2) Applying material to the airfoil and fluid domain.
3) Applying boundary conditions at inlet, outlet and the reamining walls of fluid domain.
4) Meshing and configuration of solution control.
5) Running the simulation.
6) Calculation of lift and drag force on airfoil.
7) Saving the reuslt for further analysis.

The above steps are repeated 15 times, one for each CAD file that was created by the previous component.

Property	Entity	Value
Material	Airfoil	Aluminium
Material	Fluid Domain	Air
Boundary Condition	Inlet Velocity	30 m/s
Boundary Condition	Outlet Pressure	0 Gage
Boundary Condition	Wall Condition	Slip Symmetry
Mesh Setting	Meshing Method	Autosize
Solution Control	Enable Adaptation	ON
Solution Control	Adaptation Cycles	1
Solution Control	Iterations	500
Solution Control	Turbulence Model	SST k-Omega DES
Solution Control	Analysis Type	Transient (1s Time step)

Table 3.1: Simulation Settings

D. Result Interpretation

The final component of application helps in summarising and analysing the data obtained in the previously done CFD simulation.

1) Lift Coefficient (CL)

In fluid dynamics, the lift coefficient (CL) is a dimensionless quantity that relates the lift generated by a lifting body to the fluid density around the body, the fluid velocity and an associated reference area. A lifting body is a foil or a complete foil-bearing body such as a fixed-wing aircraft.

CL is a function of the angle of the body to the flow, its Reynolds number and its Mach number. The section lift coefficient cl refers to the dynamic lift characteristics of a two-dimensional foil section, with the reference area replaced by the foil chord.
The lift coefficient CL is defined by -

$$C_L = \frac{L}{qS} = \frac{L}{\frac{1}{2} \cdot \rho \cdot u^2 \cdot S}$$

where,
• L, is the lift force,
• S, is the relevant surface area,
• q, is the fluid dynamic pressure,
• rho, is the fluid density,
• u, is flow speed.

2) Drag Coefficient(CD)
In fluid dynamics, the drag coefficient is a dimensionless quantity that is used to quantify the drag or resistance of an object in a fluid environment, such as air or water. It is used in the drag equation in which a lower drag coefficient indicates the object will have less aerodynamic or hydrodynamic drag. The drag coefficient is always associated with a particular surface area. The drag coefficient of any object comprises the effects of the two basic contributors to fluid dynamic drag: skin friction and form drag. The drag coefficient of a lifting airfoil or hydrofoil also includes the effects of lift-induced drag. The drag coefficient of a complete structure such as an aircraft also includes the effects of interference drag. The drag coefficient cd is defined as -

$$C_D = \frac{2 \cdot F_d}{\rho \cdot u^2 \cdot A}$$

where,
• **F** is the drag force, which is by definition the force component in the direction of the flow velocity,
• **rho** is the mass density of the fluid,
• **u** is the flow speed of the object relative to the fluid,
• **A** is the reference area.

3) Component Functionalities
Functionalities provided in this component are -
a) Calculates the lift and drag coefficient of airfoil from the CFD data.
b) Provides CFD data in a tabulated manner.
c) Calculates the maximum and minimum values of lift and drag coefficient and displays them with their respective angle of attack.
d) Estimation of stall angle.
e) Provides graphing capability, that helps in better understanding of the data. Three kinds of graph are available in this component, they are -

(a) C_D v/s $AoA(\alpha)$

(b) C_L v/s $AoA(\alpha)$

(c) $\frac{C_L}{C_D}$ v/s $AoA(\alpha)$

IV. RESULTS

In the previous chapter we discussed about the methodology followed during the development of each individual component of the application. In this chapter we will discuss about the tests that were carried out, and the technical obstacle faced during the development of application. To validate the working of our application, we ran three tests using it. For each test we chose a different airfoil. The airfoils that were selected for testing were symmetric airfoil NACA 0012, slightly cambered airfoil NACA 2412, large cambered airfoil NACA 4412.

©IJRASET: All Rights are Reserved | SJ Impact Factor 7.538 | ISRA Journal Impact Factor 7.894 |

These airfoil had been chosen because they have been widely used and have readily available experimental data. The drag and lift coefficient obtained from the application are compared with the experimental data available and the accuracy of the obtained results is presented.

A. Input Parameters

The first step was to generate the polynomial parameter file for each airoil, which is required for CAD automation. To do so, we first create a 2D structure of the test airfoils, by inputting their physical parameters.

Parameter	NACA0012	NACA2412	NACA4412
Max. Camber	0	0.02	0.04
Max. Camber Position	0	0.4	0.4
Max. Thickness	0.12	0.12	0.12

Table 4.1: Airfoil Parameters

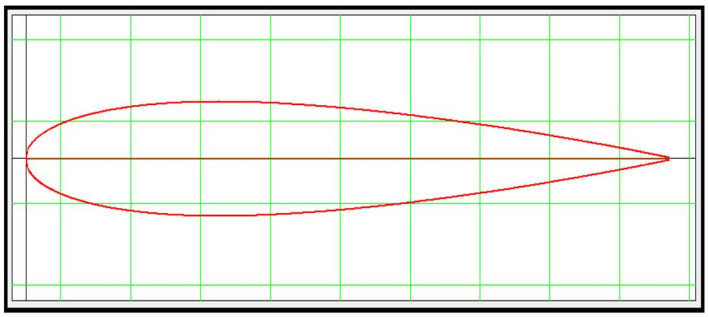


Figure 4.1: NACA 0012 Airfoil

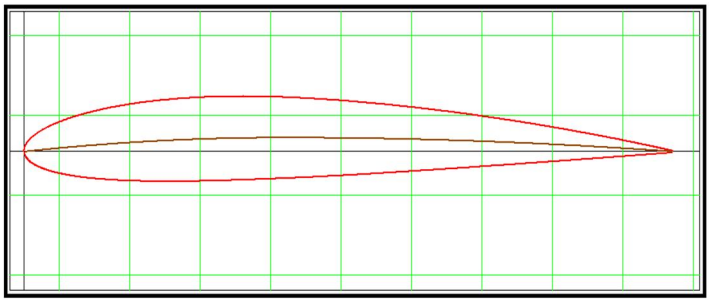


Figure 4.2: NACA 2412 Airfoil

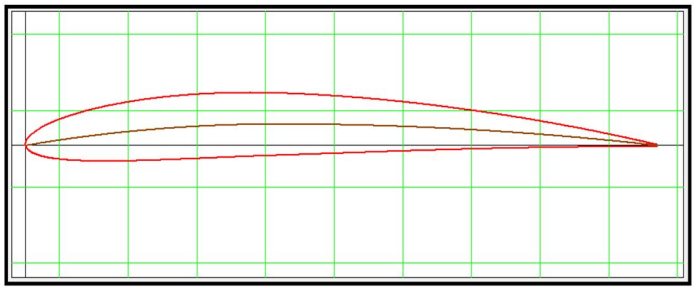


Figure 4.3: NACA 4412 Airfoil

B. Result Interpretation

Using the drag and lift force obtained in the previous step, we calculate the drag and lift coefficient for each airfoil. These calculated coefficients are then compared with the experimental data to ascertain that result obtained is within permissible limits and has the desired accuracy.

1) NACA 0012

AoA	C_L (Expt.)	C_L (Calc.)	C_D (Expt.)	C_D (Calc.)
0°	0	0	0.0081	0.0075
1°	0.107	0.168	0.0083	0.0079
2°	0.212	0.284	0.0085	0.0081
3°	0.298	0.331	0.0086	0.0084
4°	0.389	0.415	0.0086	0.0086
5°	0.457	0.485	0.0089	0.0087
6°	0.563	0.591	0.0090	0.0093
7°	0.703	0.712	0.0092	0.0094
8°	0.783	0.873	0.0093	0.0094
9°	0.916	0.928	0.0098	0.0096
10°	0.985	1.004	0.0105	0.0099
11°	1.024	1.045	0.0107	0.0101
12°	1.093	1.157	0.0112	0.0106
13°	1.136	1.169	0.0117	0.0110
14°	1.213	1.243	0.0121	0.0115

Table 4.2: Result for NACA 0012 airfoil

2) NACA 2412

AoA	C_L (Expt.)	C_L (Calc.)	C_D (Expt.)	C_D (Calc.)
0°	0.218	0.198	0.0181	0.0175
1°	0.301	0.287	0.0201	0.0195
2°	0.410	0.399	0.0226	0.0217
3°	0.472	0.441	0.0285	0.0245
4°	0.589	0.523	0.0354	0.0313
5°	0.628	0.596	0.0402	0.0398
6°	0.810	0.765	0.0482	0.0454
7°	0.881	0.843	0.0614	0.0584
8°	0.987	0.994	0.0705	0.0624
9°	1.012	1.004	0.0851	0.0821
10°	1.134	1.027	0.0991	0.0994
11°	1.194	1.089	0.1065	0.1132
12°	1.275	1.231	0.1165	0.1195
13°	1.328	1.288	0.1238	0.1213
14°	1.457	1.397	0.1328	0.1299

Table 4.3: Result for NACA 2412 airfoil

3) NACA 4412

AoA	C_L (Expt.)	C_L (Calc.)	C_D (Expt.)	C_D (Calc.)
0°	0.401	0.531	0.0080	0.0070
1°	0.513	0.617	0.0081	0.0072
2°	0.614	0.724	0.0079	0.0077
3°	0.725	0.852	0.0078	0.0079
4°	0.817	0.968	0.0082	0.0084
5°	0.928	1.059	0.0085	0.0082
6°	1.012	1.157	0.0088	0.0090
7°	1.085	1.128	0.0089	0.0091
8°	1.185	1.356	0.0092	0.0094
9°	1.254	1.398	0.0094	0.0096
10°	1.323	1.425	0.0101	0.0108
11°	1.396	1.441	0.0123	0.0119
12°	1.418	1.458	0.0162	0.0151
13°	1.478	1.512	0.0172	0.0166
14°	1.515	1.586	0.0185	0.0171

Table 4.4: Result for NACA 4412 airfoil

Comparing the results obtained with experimental data, it is observed that the obtained results are within 5% of the experimental values. Thus it can be concluded that the application generates accurate results for all types of airfoils, ranging from symmetric to highly cambered.

V. CONCLUSION

In the earlier chapter results and challenges were discussed. By the results it was found out that the application provides accurate results and helps in determining the optimal working condition of the airfoil. In this chapter, the project is concluded.

1) Rapid prototyping of airfoil has been achieved.
2) Redundant CAD and CFD tasks have been automated.
3) Tools to generate insights from results has been developed.

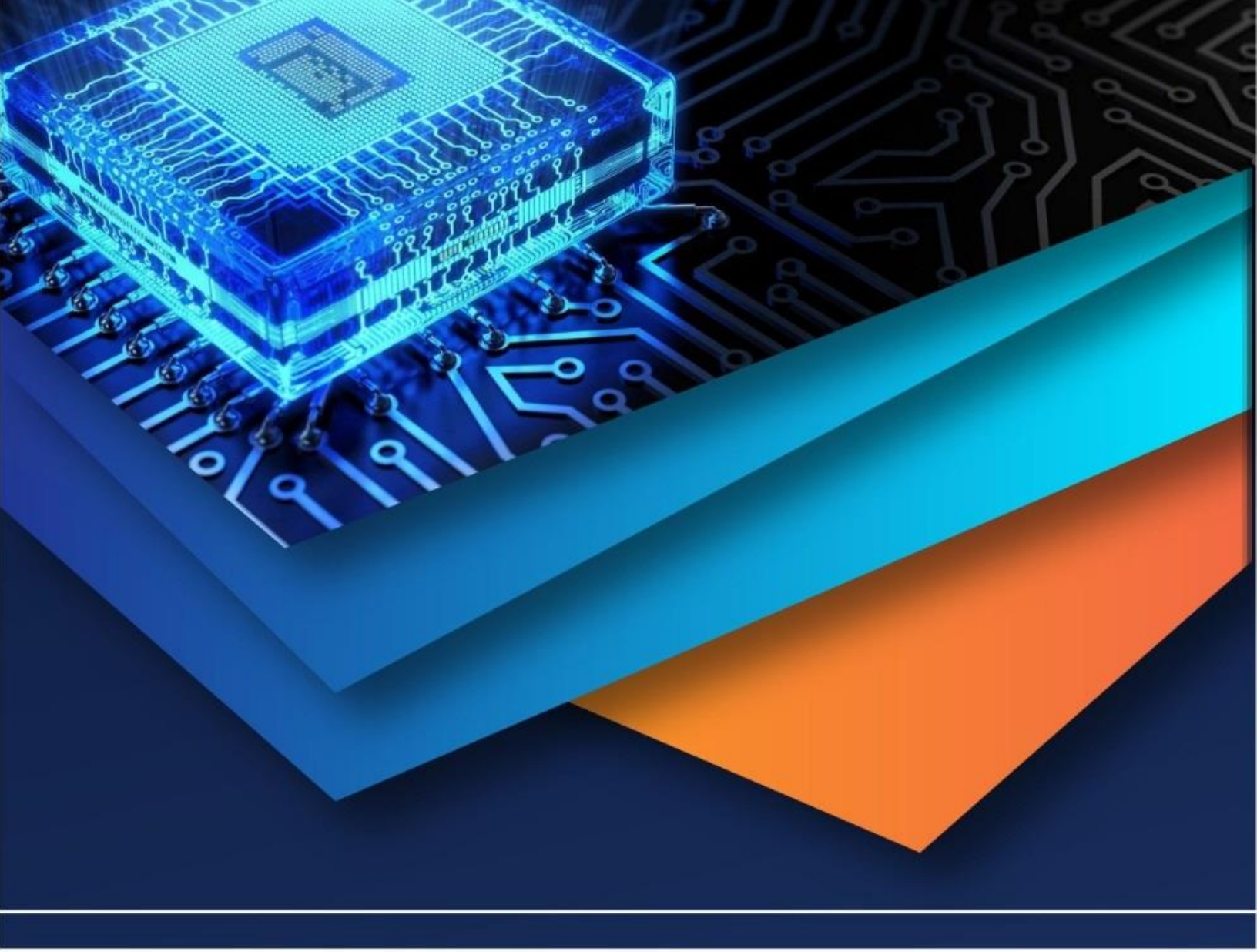
VI. ACKNOWLEDGEMENT

We would like to express our heartfelt gratitude to all those who have contributed towards the successful completion of this mechanical project report. First and foremost, we would like to extend our sincere thanks to our project guide, Mr S.P.Mankani, for providing us with invaluable guidance, support, and motivation throughout the project. Their expertise and insightful feedback have been instrumental in shaping the report and achieving the project's objectives. We would also like to thank the Mechanical Department of WIT Solapur for providing us with the necessary resources and facilities to carry out this project. Furthermore, we would like to express gratitude to our colleagues, friends, and family for their constant encouragement, moral support, and valuable insights, which has helped us to stay focused and motivated throughout the project. Lastly, we would like to thank all the authors, researchers, and professionals whose works I have referred to while preparing this report. Their ideas, findings, and suggestions have been a great source of inspiration and have enriched our understanding of the subject.

REFERENCES

[1] Abbott, Ira Herbert; Von Doenhoff, Albert Edward (1959). Theory ofWing Sections, Including a Summary of Airfoil Data. Dover. ISBN 978-0-486-60586-9.
[2] Babinsky, Holger (November 2003). "How do wings work?" (PDF). Physics Education. 38 (6): 497–503. Bibcode:2003PhyEd..38..497B. doi:10.1088/0031-9120/38/6/001. S2CID 1657792
[3] Bertin, John J.; Cummings, Russel M. (2009). Aerodynamics for Engineers (5th ed.). Pearson Prentice Hall. ISBN 978-0-13-227268 1.
[4] Clancy, L.J. (1975). Aerodynamics. London: Pitman. ISBN 0-273-01120-0.
[5] Halliday, David; Resnick, Robert (1988). Fundamentals of Physics (3rd ed.). JohnWiley and Sons.
[6] Houghton, E. L.; Carpenter, P. W.; Collicott, Steven H.; Valentine, Daniel (2012). Aerodynamics for Engineering Students (6th ed.). Elsevier. ISBN 978-0-08-096633-5.
[7] Morris, Wallace J., II (2009). A universal prediction of stall onset for airfoils at a wide range of Reynolds number flows (PhD). Harvard University.
[8] Morris,Wallace J.; Rusak, Zvi (October 2013). "Stall onset on aerofoils at low to moderately high Reynolds number flows". Journal of Fluid Mechanics. 733: 439–472.
[9] Phillips, Warren F. (2004). Mechanics of Flight. John Wiley and Sons. ISBN 978-0-471- 33458-3.
[10] Ramesh, Kiran; Gopalarathnam, Ashok; Granlund, Kenneth; Ol, Michael V.; Edwards, Jack R. (July 2014). "Discrete-vortex method with novel shedding criterion for unsteady aerofoil flows with intermittent leading-edge vortex shedding". Journal of Fluid Mechanics. 751: 500–538.
[11] Traub, Lance W. (24 March 2016). "Semi-Empirical Prediction of Airfoil Hysteresis". Aerospace.

©IJRASET: All Rights are Reserved | SJ Impact Factor 7.538 | ISRA Journal Impact Factor 7.894 |



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Call : 08813907089 (24*7 Support on Whatsapp)