Numerical Simulation of Buffeting Effect on Wings

Rahul Maurya
Mechanical & M. U
Viva Institute of Tech.
rahul.maurya9516@gmail.com

Pankaj Chauhan
Mechanical & M. U
Viva Institute of Tech.
pan.chauhan143@gmail.com

Krishna Chavan
Mechanical & M.U.
Viva Institute of Tech.
chavan8397@gmail.com

Swapnil Raut
Mechanical & M.U
Viva Institute of Tech
rautswapnil125@gmail.com

Abstract

The one of the major problem in the aircraft while accelerating in transonic regime the aircraft's wing gets shock waves due to change in velocity from sonic to supersonic regime. The critical Mach number is that free-stream Mach number at which sonic flow is first achieved on the aerofoil surface. In order to fly the aircraft in transonic regime or above transonic speed we will analyze the critical mach number. The analysis of buffeting effect on the particular type of wing of the aircraft, while predesigning is required to avoid any unpleasant conditions. In this project work numerical method will be used to analyze buffeting effect on wing of the aircraft. Using CFD technique numerical model of aerofoil will be generated and iterations will be carried numerical equations by applying appropriate Boundary conditions. Thus, validating the project work.

Keywords: Buffeting, wings ,aerofoil ,simulation, mach number
.

1. Introduction

The Buffeting is high-frequency instability, caused by airflow separation or shock wave oscillations from one object striking another. It is caused by a sudden impulse of load increasing. It is a random forced vibration. Generally, it affects the aircraft structure due to air flow downstream of the wing.

2. Objectives

The following objectives of the project are as

a. To predict the best method for turbulene modelling
b. To reduce the number of iterations
c. To minimize the residual error for numerical solution
d. To study behavior of aerofoil undergoing buffeting effect.

3. Literature Review

Azizul Hasan [2016] [3], have introduced, numerical study of some iterative methods for solving non-linear equations. Many iterative methods for solving algebraic and transcendental equations is presented by the different formulae. Using bisection method, second method is the Newton's iterative method and their results are compared. Newton's method always uses two iterations whereas the others take only one. They have concluded that the secant method is formally the most effective from the Newton method, as iterating only a single function evaluation per iteration. Analysis of efficiency from the numerical computation shows that bisection method converges too slow but it will converge.

Damien Szubert, et al. [2016] [4], have presented an analysis on the turbulent flow around a supercritical aerofoil at high Reynolds number and in the transonic regime, involving shock-wave/boundary layer interaction and buffet, by means of numerical simulation and turbulence modelling. They carried URANS simulation and OES approach. The URANS simulations based on the k−epsilon SST model have indicated a high turbulence diffusion level and a decrease in the appearance of instabilities pas the trailing edge, as well as a short shock amplitude. The OES approach provided an intermediate behaviour between the two mentioned with a reasonably extended shock amplitude and capturing of the von Karm´an and shear- layer vortices downstream and of the trailing edge

Prof. Sigrid REITER, 2008 [14] presented Simulation results which shows that fluent is good tool for evaluating critical effects of wind around buildings from the view point of pedestrian’s comfort. A specific feature of the hybrid wind tunnel is existence of the feedback signal to compensate the error in the pressure on the side wall of the cylinder and the feed forward signal to adjust the upstream velocity boundary condition.

4. Problem Definition

After carrying out several search we found the following major problem in aircraft while accelerating in transonic regime is that aircraft wing gets shock waves due to change in velocity from sonic to supersonic regime

The problem associated with existing wind tunnel testing method is as follows

a. The method can only be perform after post designing as far as time and cost is concerned
b. Wind tunnel testing machine requires large space and expensive equipment , which is why it is only conducted by several large international firms and universities
c. This Model consumes high power to test even small sized aerodynamic model.
d. High skilled and scientist are required to conduct the experiments
e. Physical model of test specimen is required hence re-alteration is not possible.
f. Since the results are in photograph and video recording format hence not easy to analyse

Therefore, we are proposing another method called numerical simulation method which can be done while pre-designing. So, we can reduce the cost & the time required.

5. Methodology

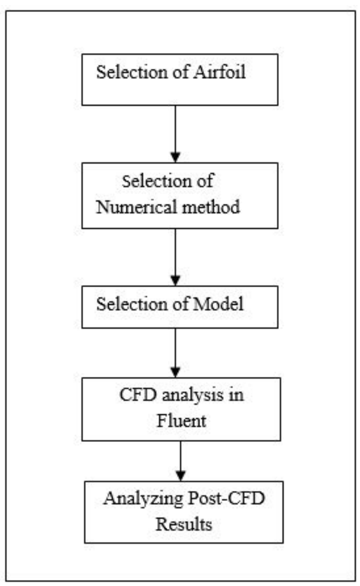

Figure 1.flowchart

5.1. Selection of aerofoil

For our experimentation we selected selig1223 and naca0013 for the following results.

5.2. Selection of Numerical Method

FVM involves the division of the computational domain into volumes and the imposition of conservation laws on these volumes. Then the Gauss Divergence theorem is applied to convert volume integrals to boundary integrals. And finally, the fluxes are evaluated at boundaries. Fluid dynamics involves conservation laws such as conservation of mass, energy and momentum and hence, FVM was deemed useful to run CFD simulations Mathematically, it is a very intense method and engineers need to dig in deeper into the field of functional analysis to gain a deeper insight to build a physical, accurate and stable CFD model.

5.3. Selection of model

Turbulence modelling is the construction and use of a model to predict the effects of turbulence. A turbulent fluid flow has features on many different length scales, which all interact with each other. A common approach is to average the governing equations of the flow, in order to focus on large-scale and non-fluctuating features of the flow. However, the effects of the small scales and fluctuating parts must be modelled.

5.4 CFD Analysis in Fluent

We used ansys fluent to do CFD simulation of airfoil because of ANSYS Fluent software is the most-powerful computational fluid dynamics (CFD) tool available. . Fluent includes well-validated physical modeling capabilities to deliver fast, accurate results across the widest range of CFD and multiphysics applications.

5.5 Analyzing Post- CFD Results

We have analyzed the following in five different methods. We used following methods.

1. Allmars spalart
2. k – ω Method
3. K-ε Method
4. .SST (Shear Stress Transport) Method
5. RANS (Reynolds-averaged Navier–Stokes) Method

6. Results and Discussion

The following result were obtained in ANSYS by using Spalart Allamaras method for are SELIG1223 aerofoil.

6.1. Pressure vs Coeffecient graph

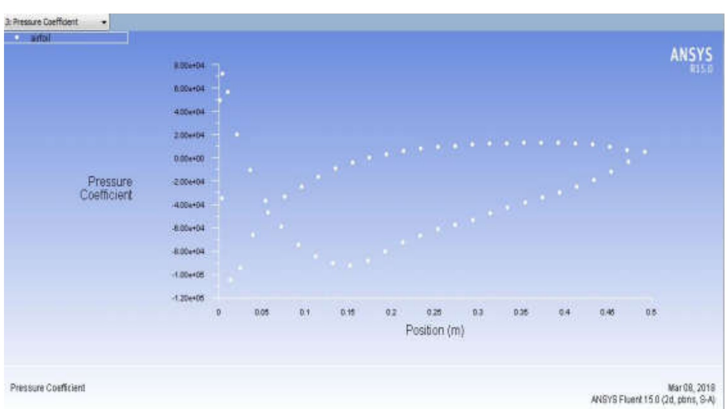

Figure 2. pressure vs position graph

In below graph of pressure coefficient vs position we can say position varies with position **.**

6.2. Residue graph

In below graph it is found that the solution is converged in 22 iteration and the range of residue is 10^{-4} to 10^{-8}.

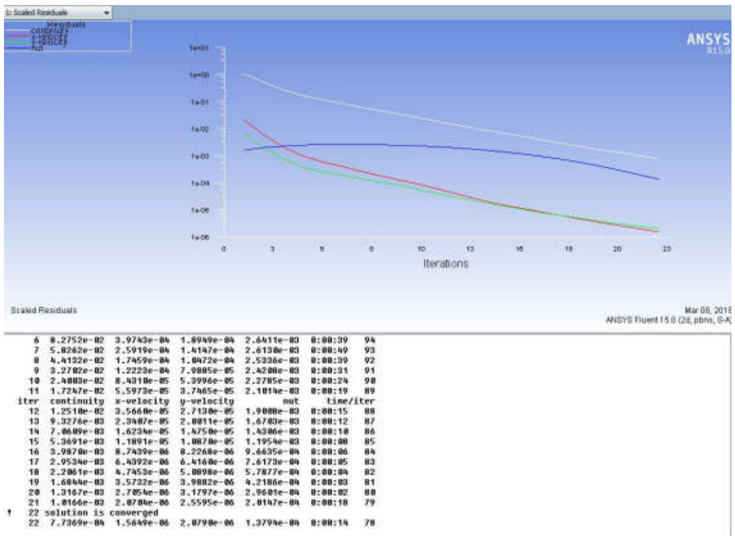

Figure 3. Residue graph

6.3. Coeffecient of Drag vs Iteration graph

We get constant value of coefficient of drag as the solution takes to converge and the net drag coefficient is 15086.868

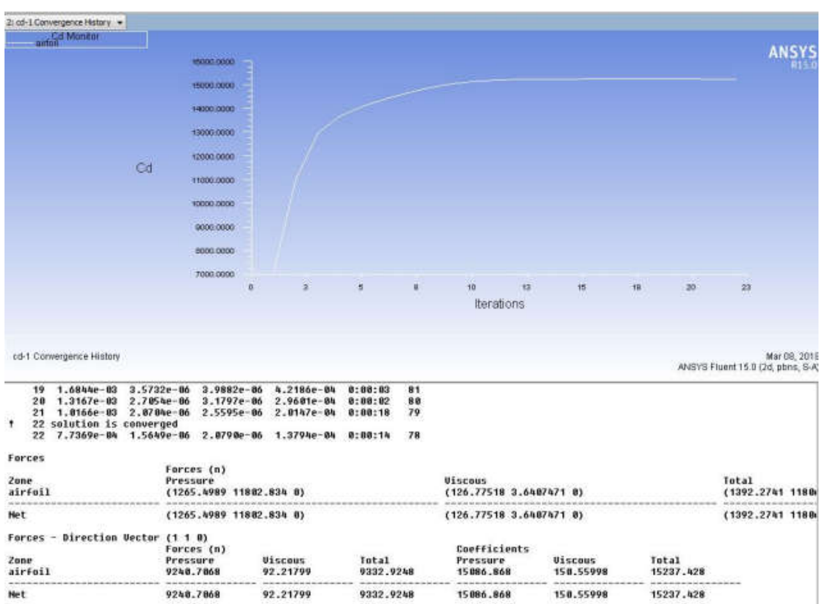

Figure 4. Coefficient of drag vs Iteration graph

6.4 Coeffecient of Lift vs Iteration graph

We get constant value of coefficient of lift as the solution takes to converge and the net lift coefficient is 19269.933.

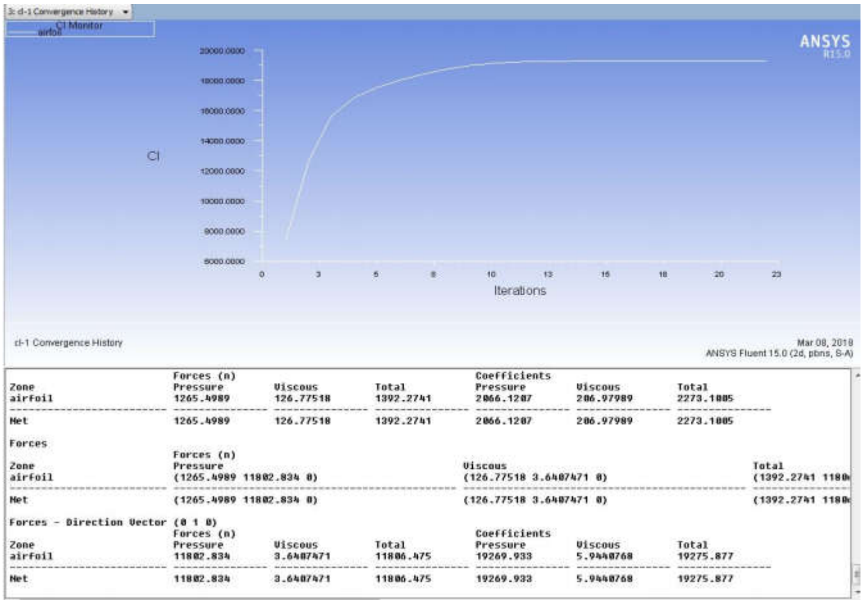

Figure 5. Coefficient of Lift vs Iteration graph

6.5 Velocity diagram

This is a velocity profile diagram. In this diagram we conclude that shockwave generates before maximum camber position.

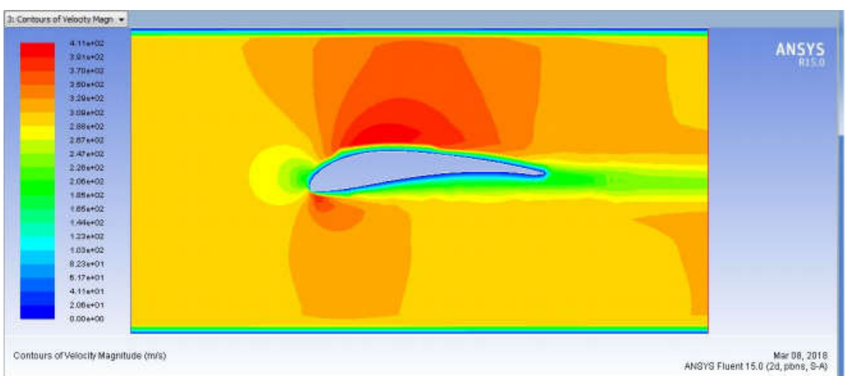

Figure 6. Velocity digram

7. Conclusion

The following conclusion would be derived from the above results after analyzing the several for various method

1. The Spalart - Allmaras method is most feasible method that generates the various graph for analyzing buffeting effect in minimum no of iterations.
2. The shock waves formed in various graph is best shown in Spalart - Allmaras method
3. The residue generated is very less in the Spalart - Allmaras method as it uses one equation model.
4. The coefficient of drag is approximately correct, because the solution gets converged easily as it takes minimum iteration .
5. The coefficient of lift is approximately correct ,because the solution gets converged easily as it takes minimum iteration .
6. The velocity profile diagram shows flows shock waves on the graph.

Acknowledgement

We would like to express our special thanks to our Principal, Dr. Arun Kumar and H.O.D., Prof. Niyati Raut who encouraged us and gave us the necessary permission to work on this project. We are extremely grateful to our Technical Project Guide – Prof. Swapnil Raut, who gave us the golden opportunity to do this wonderful project on the topic 'Wind Tunnel Testing Rig', which also helped us in doing a lot of Research. We learned a lot about so many new things and are really thankful to them.

I extend my sincere thanks to Mr. Divyesh Khanvilkar and Mr. Amish Bamania for their valuable time and support during my research work. My heartfelt gratitude also goes to all my graduate colleagues and faculty members of Mechanical Engineering Department for supporting me in direct or indirect way throughout my time.

Reference

Journal Article

[1] Boshun Gao, Aerodynamics and shock buffet of a transonic aerofoil in ground effect, Washington university in St. Louis, 2017.

[2] Jinli Liu and Zhichun Yang, numerical study on transonic shock oscillation suppression and buffet load alleviation for a supercritical aerofoil using a microtab, engineering application of computational fluid mechanics, Vol. 10, 2016, pp. 529-544.

[3] Azizul Hasan, Numerical Study of Some Iterative Methods for Solving Nonlinear Equations, International Journal of Engineering Science Invention, Vol. 5, 2016, pp.01-10.

[4] Damien Szubert, Ioannis Asproulias, Fernando Grossi, Régis Duvigneau, Yannick Hoarau, Marianna Braza, Numerical study of the turbulent transonic interaction and transition location effect involving optimization around a supercritical aerofoil, Institute de Mecanique des Fluides de Toulouse, 2015.

[5] Anthony S. Pototzky and Robert W. Moses, A Method to Analyze Tail Buffet Loads of Aircraft, RTO-MP-AVT-123, 2015.

[6] S. Derakhshan and A. Tavaziani, Study of Wind Turbine Aerodynamic Performance Using Numerical Methods, Journal of Clean Energy Technologies, Vol. 3, 2015.

[7] Hassam Nasarullah Chaudhry, John Kaiser Calautit, At El., "CFD and Experimental study on the effect of progressive heating on fluid flow inside a thermal wind tunnel", Computation ISSN 2079-3197, 2015, pp.15-24.

[8] V. Nikolic, D. Djekic, A. Radakovic, Dz. Pu cic, Numerical Methods for Solving the Dynamic Behavior of Real Systems, Scientific Publications of The State University of Novi Pazar, Vol. 6, 2014, pp.25-34

[9] Neethu Merlin Rajan, Aman Kalra, Pawan G Rebinal, Ramesh Sharma, a finite approach to modal parameter estimation of vertical tail fin, IOSR journal of Mechanical and Civil Engineering, Vol. 11, 2014, pp.30-34.

[10] A.K. Mittal, D. Ghosh, Et Al., "Wind flow simulation in the vicinity of tall buildings through CFD", The eighth Asia-Pacific on wind engineering ,2013, pp.682-690.

[11] Zhengzheng Yan, Rongliang Chen, Yubo Zhao and Xiao-Chuan Cai, A Scalable Numerical Method for Simulating Flows Around High-speed Train Under Crosswind Condition, Z. Yan al. / Comput. Phys., Vol. 10, 2013, pp.01-15

[12] A.K. Roy and P. K. Bhargava, "CFD modelling of wind flow around buildings for wind energy conversion", National conference on emerging trains of energy conservation in buildings, 2012, pp.1-10.

[13] Robert Howell, Ning Qin, At El., "Wind tunnel and numerical study of a small vertical axis wind turbine", Renewable energy an International journal, volume 35, 2010, pp.412- 422.

[14] Prof. Sigrid REITER, "Validation process for CFD simulation of wind around buildings", European built environment CAE conference, 2008, pp.1-18.

Website

[15] https://en.wikipedia.org/wiki/Aeroelasticity

[16] https://confluence.cornell.edu/display/SIMULATION/FLUENT+-+Flow+over+an+Airfoil

[17] https://en.wikipedia.org/wiki/Computational_fluid_dynamics

[18] http://airfoiltools.com/search/index

[19] https://www.cfd-online.com/