Eleyon SOUTH ASIAN JOURNAL OF ENGINEERING AND TECHNOLOGY

Full Length Article

Design and Analysis of Humpback Whale Shaped Wing

Ramya R[1], G Vel Murugan[2]

[1] Student, Department of Aeronautical Engineering, Excel Engineering College, Namakkal-647 303, Tamil Nadu, India
[2]Assistant Professor, Department of Aeronautical Engineering, Excel Engineering College, Namakkal-647 303, Tamil Nadu, India

*Corresponding Author
ramyars09 @gmail.com
(R.Ramya)

Tel.: +91 7010877433

ABSTRACT: This Project, 'Design and Analysis of Humpback Whale Wings' put forward the significant aerodynamic problem and deliberately strives to minimize by redefining the aerodynamic shape. This project mainly focuses on the aerodynamic shape of the wing to reduce the downwash generated by comparatively decreases the vortex generation at the trailing edge significantly provide that circulation remains zero. Redefining the aerodynamic shape of the wing in the sense by incorporating the circular cross-sections at the leading edge of the wing, termed as 'Ventral Fins' or 'Tubercles'. It is inspired from the Humpback whales. The tubercles on the leading edge act as passive-flow control devices that improve performance and maneuverability. .

Keywords: Circulation, Humpback whale, Leading edge, Tubercles, Ventral fins

1 Introduction

Humpback whales (Megaptera novaeangliae) are amazing animals Humpback whales developed some unique features as an individual or in a group, like breaching behaviour, generation of the most complex sound among the swimming animals and utilization of a smart bubble net fishing technique, to name a few. Despite their long body length, about 12-18 m, and their heavy body mass, about 30-40 tons, these animals are remarkable swimmers.

From hydro-dynamical point of view, high level of maneuverability of these species in turning, rolling and banking is majorly linked with the special topology of their pectoral fines, called 'flippers'. Humpback whales have the longest flipper among all whales (Fig. 1), with a length of about 0.3 of the body the length. Fig. 1 shows the planform of a humpback whale flipper. As shown, it posses a special pattern of tubercles on the leading edge and the trailing edge, which forms peaks and troughs with varying amplitude and wavelength.

On the other hand, there exist a net of ventral (throat) grooves (pleats) on the belly part of the animal. This net allows whales to expand their throat like an accordion in the lunge-feeding process. Presence of these grooves modifies external shape of the underside of the animal, even in the non-lunge stage, compared to the smooth body surface and results in some fluid dynamical consequences. Generation of the streamwise vortices is the key factor to understand the flow hydrodynamics over the humpback whale flippers.

In fact, two counter-rotating vortices with different vorticity signs are generated by the tubercle leading edge topology on different sides of the trough for any individual protuberance and also a secondary- spanwise flow forms in the leadingedge region.

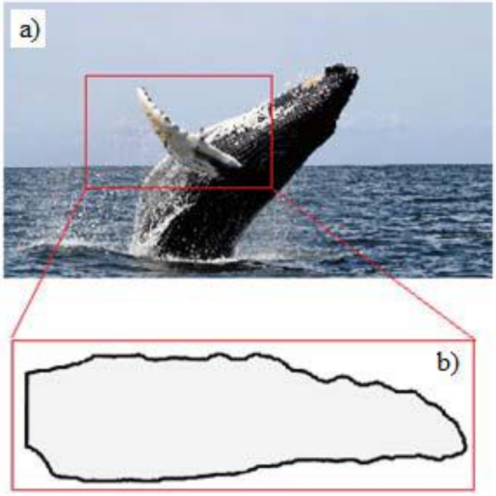


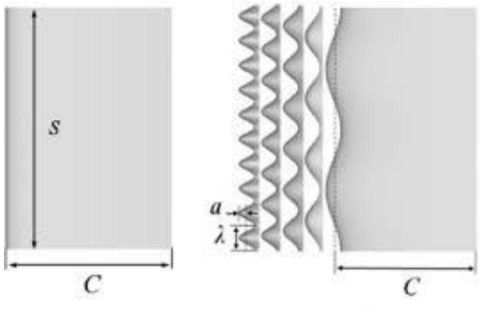
Fig. 1 Humpback whale; (a) breaching behavior [3], (b) flipper planform (reconstructed from a real geometry [4])

It is postulated that higher amount of momentum induced by streamwise vortices results in a softer/flatter behaviour in the post-stall region for wings with leading edge undulations. In this perspective, leading edge protuberances resembles vortex generators and can be seen as a passive flow control tool.

There exist more interesting lessons considering humpback whale swimming hydrodynamics, however in the present research; focus is on the behavior and design of the wings with wavy leading edge inspired by humpback whale flippers. In the following sections, details are presented. A study of characteristic viscous forces revealed an increase in form drag comparable to the savings in induced drag at 10° angle of attack. We will study changes in tubercle spacing and shape to see if further enhancement is possible at the same angle of attack. However, the greatest potential gains can be made at angles of attack greater than 15°, where boundary layer separation and wing stall can be expected. Wavy separation lines on bluff bodies support the hypothesis, yet to be confirmed experimentally, that tubercles may alter, delay, or reduce separation at higher angles of attack. Boundary layer control techniques such as vortex generators that delay wing stall often increase maximum lift by 30% or more. Optimal control of maneuvering vehicles usually consists of maximal deflection of control surfaces. Streamlined bodies such as fins and rudders may experience larger control forces and extended operating envelopes due to the addition of leading edge tubercles. The ability to reduce wing tip vorticity suggests that tubercles could also enhance the stealth of marine vehicles.

2. Wing Planform Parameter

Basic geometrical parameters adopted in the proposed model for the wing planform design are defined in Fig. 2. In general, for a swept wing, these parameters include: wing span b, amplitude a, wavelength λ .To represent geometry of the wing airfoil section, chord length c , maximum thickness max t and position of the maximum thickness max x are considered .



(a) (b)

Fig.2 wing planform without tubercle and with tubercle

3 Wing Geometry

For the tubercle wing, the leading edge of the planform is given by

xLE = 0:04 cos (4:878_z)

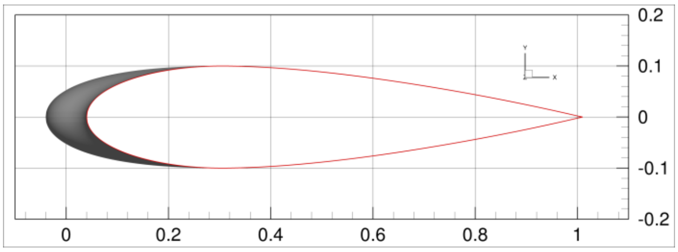


Fig 3 Tubercle wing cross section

During the recent decade, effects of the leading edge waviness on the wing performance have been studied both numerically and experimentally; Tables I and II summarize these studies, respectively. As it is clear, flow over different planforms, majorly with rectangular shape, has been simulated . Aspect ratio, AR = b! c range of the planforms is limited from 0.45 to 4.33 Reynolds number range based on the free stream velocity and mean chord length is limited from 800 to 106. The lower limit of Re , i.e. 800, is lower than transition to turbulence state, although large enough to grow first spanwise instabilities before transition to turbulence .

4 Process Methodology

4.1 Modelling

This is the first step in Ansys fluent workbench, at this step u can create a model or u can Import your existing design model to do analysis, In very first u have to import you are Catia V5 file Which is should be saved in IGES

Extension file format then double click on Geometry to open the design modular, where you have to generate your model and then add an Enclosure which is present in Tools.

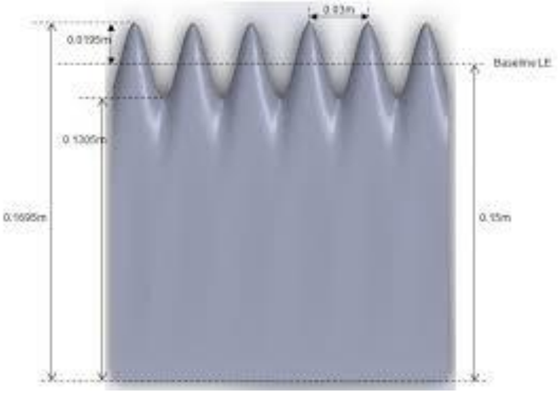


Fig.4 humpback whale shaped wing design

4.2 ANSYS Software

ANSYS offers engineering simulation solution sets in engineering problems that a design process requires. Companies in various industries use this software. ANSYS uses FEM and various other programming algorithms for simulating and optimizing various design problems. ANSYS has many sub parts out of which ANSYS Fluid Flow and Structural are chosen to run the simulations. CFD is applied for analysis of fluid mechanics and dynamics problems. The physical modelling capabilities and the fast, accurate CFD results show that ANSYS Fluent is one of the most comprehensive software for CFD modelling available in the world today.

4.3. Description of the geometry model

A schematic of the geometry model of the aerofoil and aircraft wing is shown in Fig.4. There are several numbering schemes used to characterize the shape of aerofoil, as NACA four digits, five digits, etc. In this study, NACA 2412 aerofoil is used to design the wing, in which the first digit is the maximum camber in hundredths of the chord, the second digit is the location of the maximum chamber from the leading edge in tenths of the chord, and the last two digits represent the maximum thickness in hundredths of the chord . The parameters are chosen, such as aerofoil chord c = 0.3m , aerofoil span l = 1.6m . These dimensions are used to fabricate the experimental wing model, which are also consistent with the open data of a number of test UAVs samples in Vietnam.

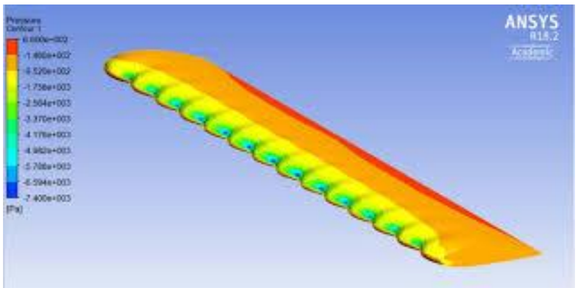


Fig 5. Aircraft wing model created in ANSYS

4.4 Meshing

Meshing process was carried out by using ANSYS Meshing. The mesh is created using several size function such that the result found to be accurate without any errors. Mesh generation is the patch Conforming Tetrahedral Meshing Method. It meshes the edges at first and then the volume. To get the better results at the surface of the wing, the boundary layer meshing is done in the wing section. The mesh is then done using parallel meshing process using all the possible systems core for fast processing of mesh generation

4.5 Material selection

Metallurgy has played a key role in the development of aviation. Until recently, some new materials have been applied in aircraft construction, such as titanium alloy, or composite. However, these superalloys are still quite expensive for the aircraft home-builder. With its advantages in weight and cost ratio, aluminum alloy is still used very widely.

4.6 Boundary condition

Boundary conditions is one of the most important and challenging parts of setting up a simulation. It is not necessarily difficult to find a combination that works, but it is essential to produce meaningful results instead of random numbers. Boundary conditions define the inputs of the simulation model. Some conditions, like velocity and volumetric flow rate, define how a fluid enters or leaves the model. Other conditions, like film coefficient and heat flux, define the interchange of energy between the model and its surroundings. .Boundary conditions connect the simulation model with its surroundings. Without them, the simulation is not defined, and in most cases cannot proceed. Most boundary conditions can be defined as either steady-state or transient. Steady-state boundary conditions persist throughout the simulation. Transient boundary conditions vary with time, and are often used to simulate an event or a cyclical phenomena. Initial conditions are a different type of condition that are active only at the beginning of the simulation. For more about initial conditions,

4.7 Simulation

In order to analyze fluid flow, The flow domain is split into smaller sub domains, which is called mesh generation. The intended use of the mesh is to separate and compute the properties of the fluid flow. Fluent uses the meshes to model the fluid space. The mesh used is shown in Fig. 6. It solves the Navier-Stokes equations numerically at each node of the mesh. Moreover, an iterative method is used by ANSYS Fluent to converge on a solution of this analysis.

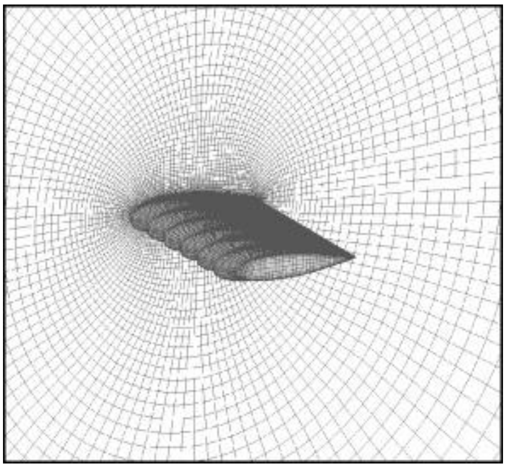


Fig 6. Meshed region

Before the simulation can be run, specific parameters and boundary conditions had to be set. First some general settings needed to be established. For example, gravity is to be neglected, time to be treated as a steady case, velocity to be taken to be in an absolute reference frame, and the solver used to be pressure based. Next the laminar model is selected. More specific methods also had to be specified. Pressure, momentum, dissipation, and energy are all modeled using second order functions. These higher order functions are generally more accurate than first order approximations , but are also more time consuming. Next steps, boundary conditions are set for the different areas of each mesh, such as wall face with zero velocity, symmetry faces, velocity inlet and pressure outlet for the fluid.

5. Results and Discussion

Simulations were performed for a straight wing, which served as a baseline for comparison, and for wavy wings with several combinations of wavelength and amplitude. Three wavelengths were considered, and for each one of them three amplitudes, totaling nine different geometries of wings with waviness. Also, for each geometry simulations were performed for angles of attack α between 0 and 21 deg, with increments of 3 deg, allowing for the observation of the aerodynamic performance over a wide range of conditions

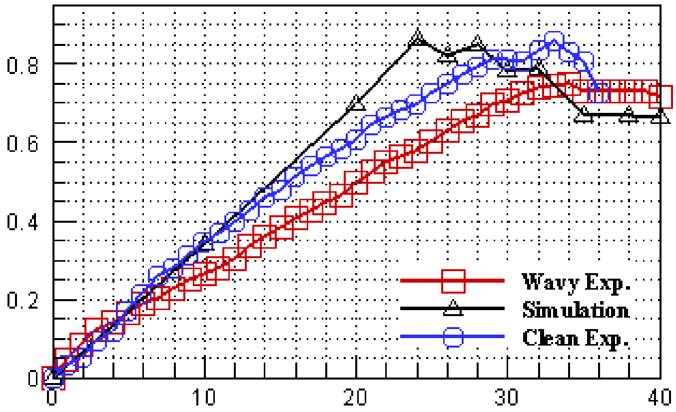


Fig.7 Tubercled rectangular wing performance curves

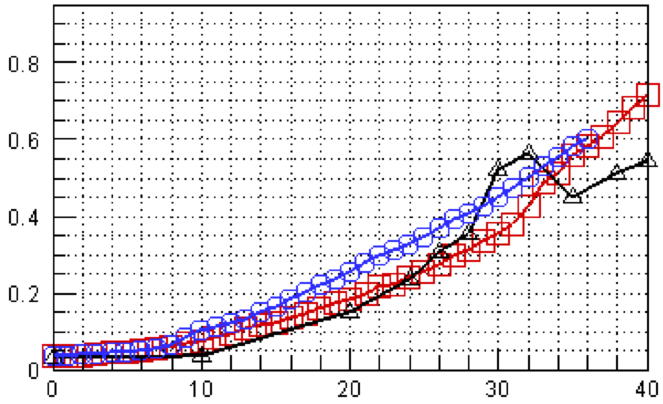


Fig. 8 Tubercled rectangular wing performance curves

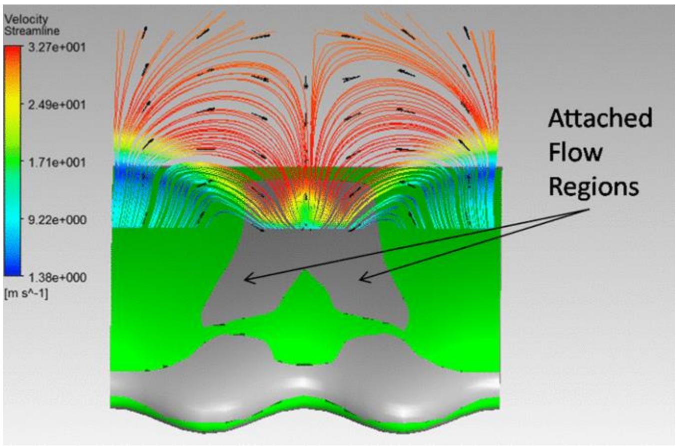


Fig. 9 Streamwise vortices on turbulence leading edge

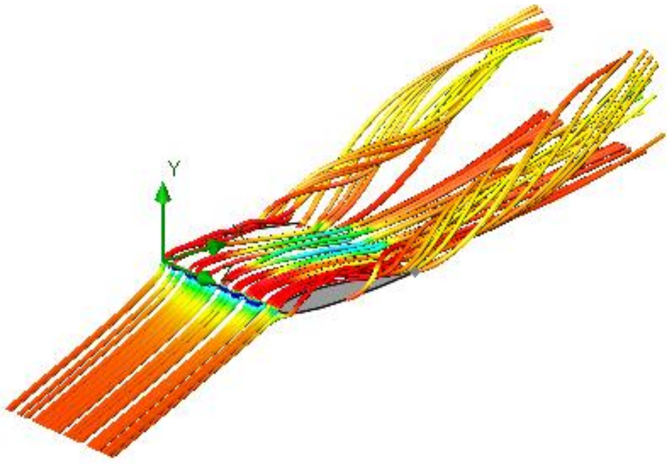


Fig. 10 Path lines over the tubercle wing

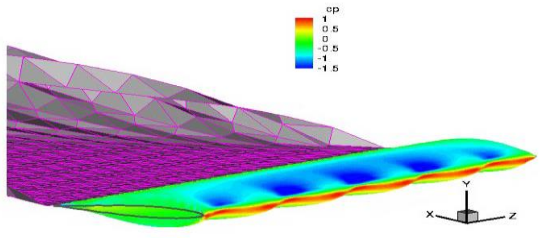
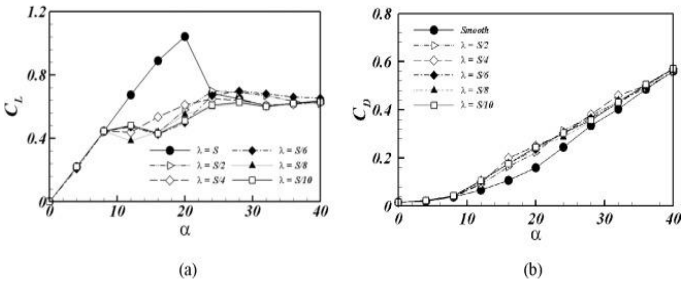


Fig.11 Simulation of flow over a finite span wing with leading edge tubercles

6. Conclusion

The flow around wings with spanwise waviness was investigated numerically for a Reynolds number Re = 1000. Several combinations of wavelength and amplitude were considered. For the shortest wavelength $\lambda/c = 0.25$, the modifications had no significant effect on the results. For $\lambda/c = 0.5$ and $\lambda/c = 1.0$, there is a reduction of the lift-to drag ratio, caused by the combination of reductions in both the lift and the drag. This reduction of L/D is accompanied by a suppression in the lift coefficient fluctuations. Also, flow visualizations showed that this behaviour is caused by a flow regime where there is a tendency for the flow to remain attached behind the waviness peaks, leading to the formation of distinct separation regions behind the troughs.



(a) (b)

Fig. 12 angle of attack vs cl and cd

Tubercles appear to be functional adaptations of humpback whale flippers. Leading edge shape modifications such as tubercles can increase useful force production while simultaneously reducing parasitic forces. Few other passive means of altering fluid flow around a wing can both increase lift and reduce drag at the same time. The idea of appropriating tubercles from humpback whales for engineering purposes is a direct consequence of biomimicry. The performance enhancement that we document for 10° angle of attack vanishes as the angle of attack decreases to zero, implying no penalty for the presence of tubercles during a null state.

References

[1] Fish, F.E., Battle, J.M.. Hydrodynamic design of the humpback whale flipper. Journal of Morphology 1995;225:51–60.

[2] Miklosovic, D.S., Murray, M.M., Howle, L.E., Fish, F.E.. Leading-edge tubercles delay stall on humpback whale (Megaptera novaeangliae) flippers. Physics of Fluids 2004;16:L39

[3] Abbot, I. H. and von Doenhoff, A. E. (1959). Theory of wing sections. Dover.

[4] E.A. Van Nierop, S. Alben and M.P. Brenner, “How bumps on whale flippers delay stall: an aerodynamic model”, Physical Review Letters, PRL 100, 054502, February 2009.

[5] A. Taheri, “On the hydrodynamic effects of humpback whale’s ventral pleats”, American Journal of Fluid Dynamics, vol.8, no.2, pp.47-62,2018.

[6] J. Favier, A. Pinelli and U. Piomelli, “Control of the separated flow around an airfoil using a wavy leading edge inspired by humpback whale flippers”, Journal of Comptes Rendus Mecanique, vol. 340,pp.107–114, 2012.

[7] M. Bolzon, R.M. Kelso, and M. Arjomandi. "Force measurements and wake surveys of a swept tubercled wing", Journal of AerospaceEngineering, Vol.30, No.3, 04016085, 2017.

[8] M. W. Lohry, D. Clifton and L. Martinelli, “Characterization and design of tubercle leading-edge wings”, 7th International Conference on Computational Fluid Dynamics (ICCFD7), Big Island, Hawaii, July 9-13, 2012.

[9] J. Joy, T.H. New and I. H. Ibrahim, “A computational study on flow separation control of humpback whale inspired sinusoidal hydrofoils International Journal of Mechanical, Aerospace, Industrial,Mechatronic and Manufacturing Engineering, vol.10, no.2, 2016.

[10] C. Cai, Z. Zuo, S. Liu and Y. Wu, “Numerical investigations of hydrodynamic performance of hydrofoils with leadingedge protuberances”, Journal of Advances in Mechanical Engineering, vol. 7,no.7, pp.1–11, 2015.