Design optimisation and analysis of dual radius circulation control airfoils

[1]P Karthi, [2]T Banu, [3]P.Saranya
[1,2,3]Assistant Professor, Department of Aeronautical Engineering
Nehru Institute of Technology, Coimbatore.
nitcvishnu@nehrucolleges.com[1]

Abstract: Many airfoils have been developed for aircraft, most notably the old NACA airfoils. New airfoils can also provide less obvious advantages. Airfoils can be designed to exhibit more docile stall characteristics, for example. Such characteristics improve the flying qualities of aircraft and reduce the loads on wind-turbine and fan blades. Airfoils can also be designed to produce maximum lifts that are essentially unaffected by roughness. This characteristic leads to increased flight safety for aircraft, consistent peak power for wind turbines, and reliable operation for fans. The main theme or goal of this experimentation is to obtain the expected lift characteristics of the newly designed circulation control airfoils by varying the thickness, basically reduced in accordance with the GACC conventional dual radius circulation control airfoil about 0.25% to each airfoil's thickness and finding out the airfoil attaining maximum altitude with the help of a solidworks and solidworks simulation. New airfoils will increase your profits and your customers' profits. Let's examine the economics for three different applications: aircraft, wind turbines, and fans. The cost of the airfoil design is trivial compared to the economic benefits of the new technology. For example, the cost of tailoring an airfoil to a single-engine airplane is less than 0.1 percent of the cost of bringing that airplane to production, yet the new airfoil determines to a large extent the airplane's performance and handling. For larger aircraft, the cost-benefit ratio is even better because the relative cost is lower; for smaller aircraft, the ratio is also better because the relative benefit is larger. For wind turbines, the cost of the airfoil design is less than five percent of the annualenergy increase. In other words, the increased energy production will pay for the airfoil design within the turbine's first month of operation. For fans, the cost-benefit ratio is similar. Computer simulation has become an essential part of science and engineering. Digital analysis of components, in particular, is important when developing new products or optimizing designs. Today a broad spectrum of options for simulations available; researchers use everything from basic programming languages to various high-level packages implementing advanced methods. Computer simulation has become an essential part of science and engineering. Digital analysis of components, in particular, is important when developing new products or optimizing designs. Today a broad spectrum of options for simulations available; researchers use everything from basic programming languages to various high-level packages implementing advanced methods. Though each of these techniques has its own unique attributes, they all share a common concern: When considering what makes software reliable, it‟s helpful to remember the goal a computer simulation environment is simply a translation of real world physical laws into their virtual form.

Keywords: New airfoil,EpplerAirfoils, Solidworks,Solidworks simulation.

Introduction

Computer simulation has become an essential part of science and engineering. Digital analysis of components, in particular, is important when developing new products or optimizing designs. Today a broad spectrum of options for simulations available; researchers use everything from basic programming languages to various high-level packages implementing advanced methods.

Though each of these techniques has its own unique attributes, they all share a common concern: When considering what makes software reliable, it‟s helpful to remember the goal a computer simulation environment is simply a translation of real world physical laws into their virtual form. How much simplification takes place in the translation process helps to determine the accuracy of the resulting model .It would be ideal, then, to have a simulation environment that included the possibility to add any physical effect to your model. That is what COMSOL is all about. It‟s a flexible platform that allows users to model all relevant physical aspects of their designs. Expert users can go deeper and use their knowledge to develop customized solutions, applicable to their unique circumstances. With this Kind of allinclusivemodeling environment, COMSOL gives you the confidence to build the model you want with real-world precision. Certain characteristics of COMSOL become apparent with use Compatibility stands out among these. COMSOL requires that every type of simulation included in the package has the ability to be combined with any other. This strict requirement mirrors what happens in the real world. For instance in nature electricity is always accompanied by some thermal effect; the two are fully compatible. Enforcing compatibility guarantees consistent multi physics models and the knowledge that you never have to worry about creating a disconnected model again. Another noticeable trait of the COMSOL platform is adaptability. As your modeling needs change, so does the software. If you find yourself in need of including another physical effect, you can just add it. If one of the inputs to your model requires a formula, you can just enter it. Using tools like parameterized geometry, interactive meshing and custom solver sequences, you can quickly adapt to the ebbs and flows of your requirements. The flexible nature of the COMSOL environment facilitates further analysis by making “what-if” cases easy to set up and run. You can take your simulation to the production level by optimizing any aspect of your model. Parameter sweeps and target functions can be executed directly in the user interface. From start to finish, COMSOL is a complete problem-solving tool.

EpplerAirfoils

The application of potential-flow theory together with boundary-layer theory to airfoil design and analysis was accomplished many years ago. Since then, potential-flow and boundary layer theories have been steadily improved. With the advent of computers, these theories have been used increasingly to complement wind-tunnel tests. Today, computing costs are so low that a complete potential-flow and boundary-layer analysis of an airfoil costs considerably less than one percent of the equivalent wind-tunnel test. Accordingly, the tendency today is toward more and more commonly applicable computer codes. These codes reduce the amount of required wind-tunnel testing and allow airfoils to be tailored to each specific application. The code described in this paper has been developed over the past 45 years. It combines a conformalmapping method for the design of airfoils with prescribed velocity-distribution characteristics, a panel method for the analysis of the potential flow about given airfoils, and an integral boundary-layer method. It is very efficient and has been successfully applied at different Reynolds numbers. A compressibility correction to the velocity distributions, which is valid as long as the local flow is not supersonic, has been incorporated into the code.

Dual Radius Air foil Structure

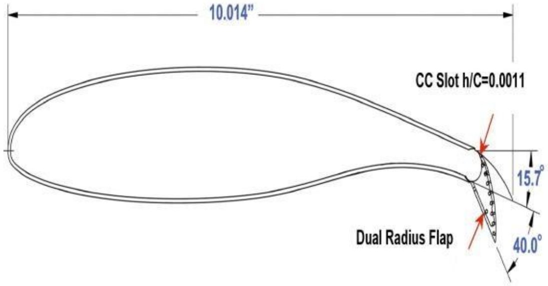


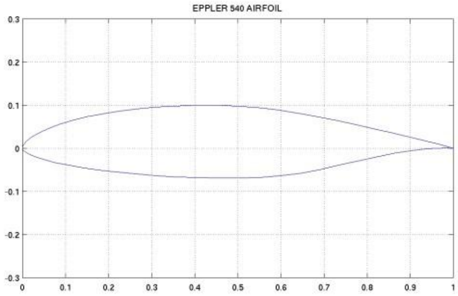
Figure: 1 Dual radius airfoil profile

The GACC-DR airfoil was analyzed at the 10 psf experimental condition where a majority of the PIV and hot wire data were taken. This dynamic pressure corresponds to a freesteam Mach number of 0.0824 and a chord Reynolds number of 0.47 million. The majority of the data were taken at a 0! geometric angle of attack. Note that for all computations in this study, the computational blowing coefficient is computed using the same definitions as the experimentally derived value. All calculations are performed with the SA model unless otherwise noted. Although no angle of attack corrections were applied to the experimental data, past experience with CC airfoil data led the authors to initially investigate the effects of induced angle of attack from the wind tunnel wall/model juncture vortices. The following formula based on geometric angle of attack and the experimentally measured sectional lift coefficient, Clexp, The test estimates an effective angle of attack. of experimental chordwise pressure coefficients with the FUN3D calculations at the geometric and estimated effective angles of attack for the no blowing condition, Cμ = 0. Figure: 12 shows a similar comparison for Cμ = 0.09. The corresponding experimental and computational lift coefficients are also included in the plots. The computational results indicate that the angle of attack correction provides a rational level of adjustment to achieve agreement with the measured upper surface pressure peaks and sectional lift coefficient. The variation of experimental and computational sectional lift coefficient, Cl, with blowing coefficient for the GACC-DR is shown in with the computations being performed at the estimated effective angles of attack.

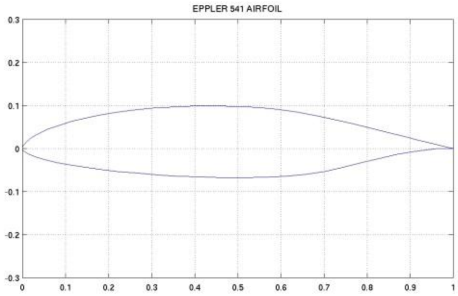
Even with the estimated angle-of-attack corrections, the computed sectional lift coefficient at Cμ = 0.09 are significantly higher (by about 30%) than the experimental value. also indicates that with jet blowing the CFD is predicting too high a level of circulation, yielding pressure levels that are somewhat low over the entire upper airfoil surface. Computations were made to investigate the sensitivity of the sectional lift coefficient and pressure distributions to some of the experimental parameters. FUN3D computations to assess the sensitivity of the lift coefficient to mass blowing rate indicate that a drop of 10% in mass flow will result in a 10% drop in lift coefficient. FUN3D computations investigating the effects of jet slot expansion under pressure indicate that a 0.003 inch expansion of the slot resulting in a 30% increase in slot width, which will reduce the lift coefficient to 3.76 (an 8.25% descrease from the baseline configuration). Computatilonsmodeling the upper and lower wind tunnel walls indicated a negligible effect on the airfoil lift coefficient and pressure distributions. Computations were also made to investigate the sensitivity of the sectional lift coefficient and pressure distributions to some of the numerical/computational parameters. shows a comparison of the chordwise pressure distributions between the three CFD codes with no blowing for the experimental angle of attack. All codes tend to prematurely separate on the lower surface of the airfoil. This may be due to a difference between the location of transition in the experiment and the delay in the development of turbulent eddy viscosity that that occurs for ”fully turbulent” low Reynolds number computations. compares CFD results from the three codes for a blowing coefficient of Cµ = 0.09 and a corrected angle of attack of −4.62!. Like the no-blowing case, the 3 codes predict very consistent, similar results compared to each other. Compares results using two different turbulence models for the same case. Although SST predicts a somewhat a lower lift coefficient than SA, the de overall circulation is still significantly high compared to experiment. As with the GTRI-DR when the Cµ values increase, the maximum y+ values on the upper flap surface also increase in the jet region over the flap. The effect of the larger y+ values was studied on the GACC-DR by generating a new unstructured grid with the same surface point distributions but with the wall spacing reduced by a factor of six. For the new mesh with refinement in the normal wall spacing, the maximum y+ value on the upper flap for the FUN3D solution at Cµ = 0.09 is reduced to just below one. The sectional lift coefficient did not change from Cl = 4.00 with the normal grid spacing refinement, and there were no significant changes in the chordwise pressure coefficient.

Airfoils chosen for experimentation

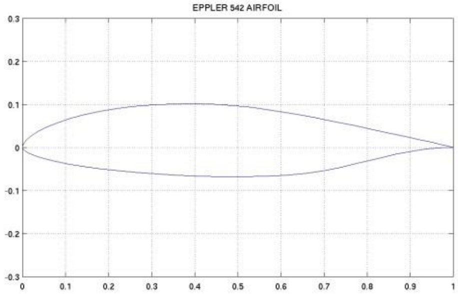
E540: Eppler E540 general aviation airfoil Max thickness 16.9% at 45.5% chord.
Max camber 1.6% at 35.7% chord



E541: Eppler E541 general aviation airfoil Max thickness 16.6% at 46.2% chord.
Max camber 1.7% at 36.3% chord



E542: Eppler E542 general aviation airfoil Max thickness 16.9% at 41.3% chord.
Max camber 1.9% at 31.5% chord



Conceptual design

The main concept of the experiment is the alternative design of the three airfoils shown above.Theseepplerairfoils are mostly used in small planes and RC models.The prime idea of the experimentation is to apply the dual radius to the above three airfoils and determine the result analysis using the Solidworks software version 2010.

Solidworks

SolidWorks Premium is a comprehensive 3D design solution that adds powerful simulation and design validation to the capabilities of SolidWorks Professional, as well as ECAD/MCAD collaboration, reverse engineering, and advanced wire and pipe routing functionality.
Handle all aspects of your part and assembly modelingwithSolidWorks 3D designsystem. Effective product design involves a wide range of tasks that demand flexibility in your software. 3D solid modeling offers several advantages over traditional 2D design, but you want 3D CAD tools that you can use every day while being powerful enough to handle all the aspects of your design process.

Solidworks Simulation

Ensure product robustness using the range of powerful structural simulation capabilities in SolidWorks Simulation Premium. It goes beyond SolidWorks Simulation Professional and includes additional tools for simulating nonlinear and dynamic response, dynamic loading, and composite materials.

References

1. Rumsey, C. L., Gatski, T. B., Sellers III,W. L., Vatsa, V. N., and Viken, S. A., “Summary of the 2004 CFD
2. Validation Workshop on Synthetic Jets and Turbulent Separation Control,” AIAA Paper 2004–2217, 2004.
3. Jones, G. S. and Joslin, R. D. e., Proceedings of the Circulation Control Workshop March 2004, NASA CP-2005213509, 2005.

4. Slomski, J. F., Gorski, J. J., Miller, R.W., and Marino, T. A., “Numerical Simulation of Circulation Control Airfoils as Affected by Different Turbulence Models,” AIAA Paper 2002– 0851, 2002.
5. Paterson, E. G. and Baker, W. J., “Simulation of Steady Circulation Control for Marine-Vehicle Control Surfaces,” AIAA Paper 2004–0748, 2004.
6. Chang, P. A., Slomski, J. F., Marino, T. A., and Ebert, M. P., “Numerical Simulation of Two- and ThreeDimensional Circulation Control Problems,” AIAA Paper 2005–0080, 2005.
7. Swanson, R. C., Rumsey, C. L., and Anders, S. G., “Progress Towards Computational Method for Circulation Control Airfoils,” AIAA Paper 2005–0089, 2005.
8. Englar, R. J., Smith, J. J., Kelly, S. M., and Rover III, R. C., “Application of Circulation Control to Advanced Subsonic Transport Aircraft, Part I: Airfoil Development,” Journal of Aircraft, Vol. 31, No. 5, Sept. 1994, pp. 1160–1168.
9. Lui, Y., Sankar, L. N., Englar, R. J., Ahuja, K. K., and Gaeta, R., “Computational Evaluation of the Steady and Pulsed Jet Effects on the Performance of a Circulation Control Wing Section,” AIAA Paper 2004–0056, 2004.
10. Jones, G. S., Viken, S. A., Washburn, A. E., and Cagle, C. M., “An Active Flow Circulation Controlled Flap Concept for General Aviation Aircraft Applications,” AIAA Paper 2002– 3157, 2002.