A STUDY ON ROLE OF CAMBER AND THICKNESS RATIO ON THE AIRFOIL CHARACTERISTICS USING CFD SOFTWARE

Dhatchanamurthy.P[1], Gopinath.M[2], Karthikeyan.L.M[3]

[1]*Department of Aeronautical Engineering, Kathir College of Engineering (India)*

[2]*Deputy Manager (Aero), ARDC, Hindustan Aeronautical Limited (India)*

[3]*Department of Aeronautical Engineering, Techno Global University (India)*

ABSTRACT

Airfoil is the most important element of an aircraft, forming the basis for shape of any wing. Traditionally airfoils were being development by empirical methods taking considerable support form wind tunnel testing. The main purpose of this study is to adequately acquaint, as a first step, with standard commercial software for computation of viscous flow over an airfoil. In the second step we study in detail the influence of the position of the maximum camber and of the thickness-to chord ratio on the airfoil characteristics, choosing the NACA (National Advisory Committee for Aeronautics) 23012 as a candidate. Validation of the commercial software to be used is a crucial step to this study. Here we have used the commercial software, the FLUENT, for the two-dimensional viscous flow over an airfoil, along with GAMBIT for generation of suitable grids. NACA 23012 airfoil has been used in the validation of the software since large body of literature exists for this classical airfoil along with reliable wind tunnel results. We continue with the study with this airfoil and carry out a study in which we have moved the position of maximum camber from its standard value of 15 percent to forty percent, in order to study its influence on the stall pattern.

Keywords- Airfoil, Chamber, FLUENT, GAMBIT, NACA

I. INTRODUCTION

A well designed airfoil should possess adequate and reliable lift, along with low drag values under sustained operating conditions. Anderson. J [1] explains fundamentals of airfoils. Slawomir Koziel and Leifur Leifsson [2] has discussed the multi-fidelity high-lift aerodynamic optimization of single-element airfoils. Cody Lafountain, Kelly Cohen and Shaaban Abdallah [3] designed a camber controlled airfoil design for morphing unmanned aerial vehicles. It is also essential that the airfoil possesses low pitching moments. Good low-speed characteristics are essential for good aircraft handling qualities. Moore M, Wilson P A, and Peters A.J [4] has discussed the ground effect in airfoil geometry of a wing. In order to ensure this the airfoil should have a large value of lift coefficient and, importantly, mild stall pattern. Maximum value of lift coefficient is influenced by thickness ratio and generally this is ensured by thickness ratios in the range of 12 to 15 percent, the maximum value of lift coefficient diminishing on either side of this range. Ranzenbach.R. and Barlow.J [5] has studied the two-dimensional airfoil with ground effect. The stall pattern is governed by the location of the maximum camber on the chord. While the location of maximum camber close to the leading edge results in reduced drag, these airfoils have sharp stall. Also the maximum lift is sensitive to the contamination of the leading edge, for example, dirt sticking to the airfoil or ice packing up around the stagnation point around the leading edge under icing atmospheric conditions. Akira Oyama and Kozo Fujii [6] explained airfoil design optimization for airplane in mars exploration. Ranzenbach, R. and Barlow, J. [7] has studied the ground effect of cambered airfoils in wind tunnel and road conditions. It is owing to the great efforts mounted in the United States of America (USA) at National Aeronautics and Space Administration (NASA) during the 1930s that a systematic body of data exists on a number of airfoils. Indeed it was this study that brought about a standardization of nomenclature for the airfoils. The characterizations of these airfoils were established mainly by careful experimentation in well-designed wind tunnels [8].

Over the last forty years considerable concerted effort all over the world has seen considerable growth and development in the area of Computerized Fluid Dynamics (CFD) in the form of developments in flow algorithms, numerical iterative schemes for the solution of the conservation laws of fluid flows. Mohammad Kouhi, Gabriel Bugeda, DongSeop Lee and Eugenio Oñatehas [9] used adaptive remeshing in CFD for aerodynamic shape optimization. All this has resulted in comprehensive computer codes, for the analysis of three-dimensional flows over quite complex aircraft shapes close to real-life shapes including viscous effects, so much so that at the present stage CFD can virtually deal with very complex real life situations such as a complete aircraft with all its control surface deflections in various flow regimes. Frederik Zahle, Christian Bak, Niels N. Sørensen Tomas Vronsky and Nicholas Gaudern [10] has explained the numerical optimization set up using two dimensional CFD.

In this study we have chosen to use the commercial software to try and examine the capability of CFD as a tool for the analysis of flow characteristics of an airfoil. K.M.Pandey, S.Chakraborty and K.Deb [11] analyzed the flow through compressor cascade using FLUENT in CFD. We have used the flow code FLUENT along with the grid generation code GAMBIT. The study includes the validation exercise in which one of the most standard airfoils namely the NACA 23012 is used as a standard. A comparison is made against well established wind tunnel experiments, generally accepted as standard experiment results [12]. The validation includes examination of several aspects on numerical parameters which actually dictate the results. They include the size of computational domain, the pattern of grids as well as their distribution and of the placement of grid points closest to the solid boundary in order to have a grid suitable for handling viscous flows. Computations have been carried out for each airfoil case for an incidence range between -2 to 16^0 or more till the convergence pattern in the residuals breaks down. It is realized that the code is capable of resolving complex flow pattern over airfoils close to the point of separation and even slightly beyond. After this validation study we have studied thickness ratio as a parameter to bring out the role of thickness ratio on the value of maximum lift coefficient. The pattern of lift behavior in the neighborhood of stall is a very important criterion in the design of airfoils and it is believed that this pattern is governed by the position of max chamber. It is reported that the position of maximum chamber as it moves closer to the leading edge results in sharp stall pattern. Q.H.Nagpurwala, S.Subaramu, M.D.Deshpande and S.R.Shankapal [13] has studied the performance of miniature propellers in MAVs. Thomas R. Barrett, Neil W.Bressloft and Andy J.Keane [14] has explained the airfoil design and optimization using multi-fidelity analysis. We have obtained pressure distribution patterns and the streamline flow pattern as an aid to study characteristics of flow over airfoils and to bring out the physical features of the flow.

II. VALIDATION OF THE FLUENT CODE

Since the number of flow parameters in the flow codes is large it is essential to choose a set of values for these parameters, either from experience or initial judgment and carry out a validation exercise to clearly study the capability and accuracy of codes to handle both flow and geometry of concern. A. Firooz and M. Gadami [15] have analyzed the ground effect of NACA 4412 airfoil in fixed and moving ground conditions. For validation we have chosen baseline NACA 23012 airfoil as a candidate since our study in this project concerns the same airfoil with parametric variation of position of camber and thickness ratio. This study contains comparison of basic airfoil lift and drag vs. angle of attack as a basis of validation. We have chosen standard experimental results available in literature to establish validation of the software. Based on this study we fix up the range of parameters to be used for our further investigation of flow over different airfoils.

III. CHARACTERISTICS OF NACA 23012 AS VALIDATION EXERCISE

Fig.1 shows 23012 airfoil as per NACA four digits series profile generator [16]. The standard coordinates are the airfoil is imported to GAMBIT from NACA resource [17] and quadrilateral grids are created over sub-domains into which the main domain of fluid flow is divided. We have taken the various dimensions for the numerical flow domain as shown in Fig.2. The airfoil is placed between (0,0) and (1,0). The chord of the airfoil is thus unity. Fig. 3 shows the quadrilateral grids within the flow domain, with suitable clustering.

Fig.1: The NACA 23012 Aerfoil

J. P. Vandoormaal, and G. D. Raithby [18] explains a simple method for calculate incompressible Flows. In our study we have chosen Spalart-Allmaras one equation model for dealing with turbulence. We have retained the default options for the code parameters in this model. We use SIMPLE algorithm as flow model and we have used second order upwind finite differencing. There are several iterative methods available in FLUENT for solving simultaneous equations that result from discretisation of equations of conservation of motion for the finite volume grid chosen. It has been our experience that converging results are obtained with relaxation factor for continuity, for x and y moment and for numerical turbulent viscosity all set at 0.3 and we also have chosen implicit iterative scheme available as an option in FLUENT. Fig. 4 and fig.5 shows the close up view of the airfoil showing clustering of grids near the leading and the trailing edges.

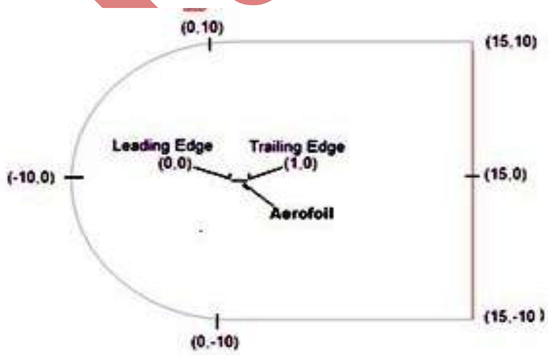


Fig. 2: Computational domain chosen domain, with suitable

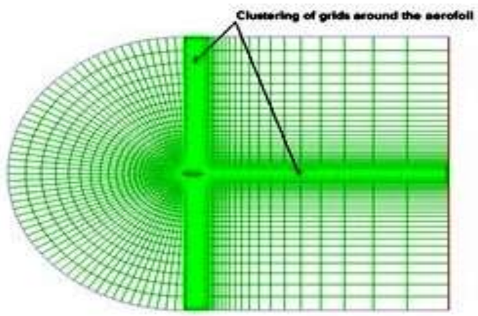


Fig. 3: Quadrilateral grids within the flow clustering

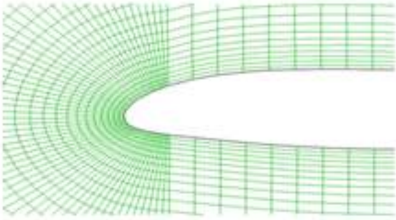


Fig.4: Clustering of grids near the leading edge

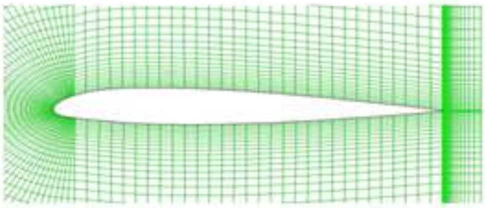


Fig.5: Clustering of grids near the leading and trailing edges

IV. FLOW SIMULATION VERSUS PREDICTION CAPABILITIES OF CFD

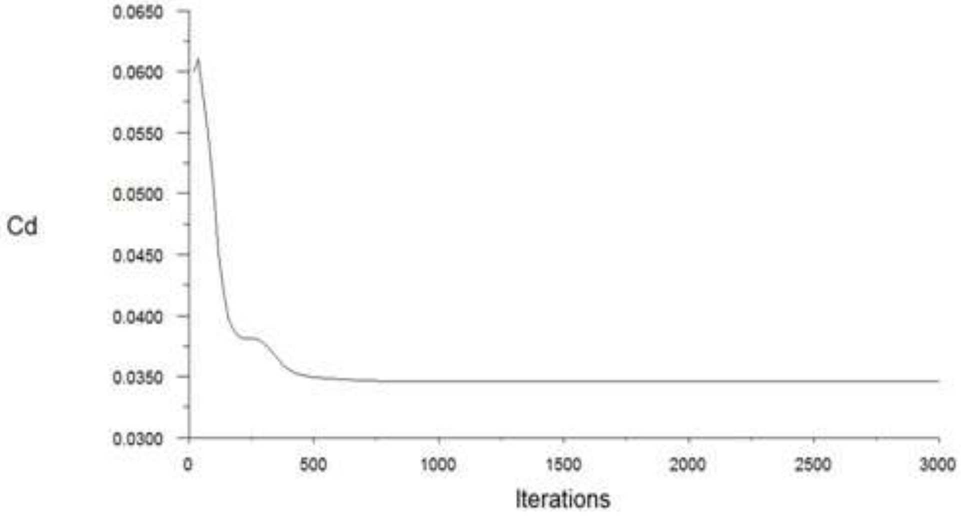
Considerable progress has been made in the field of computational fluid dynamics by the way of excellent algorithms, of flow models for particularly turbulent flows as well as the equations of conservation laws of fluid dynamics, very many software are available in literatures. It is the nature of computational fluid dynamics in general and fluid mechanics in particular that there is a large number of extraneous parameters peculiar to algorithm or to flow model, turbulence etc. Further, one of the essential steps in CFD namely the grid generation also would involve several parameters peculiar to geometry; some of these would imply a need for special grids around specific flow regions where flow gradients are high as well as the peculiarity of grids for capturing boundary layers and wakes. We would like to distinguish the simulation capabilities from predictive capabilities of a given software or of a flow model. We mean by the simulation capability the possibility of obtaining any experimental results considered to be accurate but known as the time of computation. The results may not yield comparable results in the first attempt or run of the code. It may require certain efforts to alter or tune in some of the floating parameters or fixes in the code recommended by the code developers. But with this effort one might get eventually reasonably comparable results.

Often this may take some time. But in itself this capability also is of practical importance because after having fixed up the parameter space and after assessing that these will give the best results one may continue to use the code along with the tuned parameters for generating data for geometries quite close to the geometry used in simulation exercise to generate useful practical results.

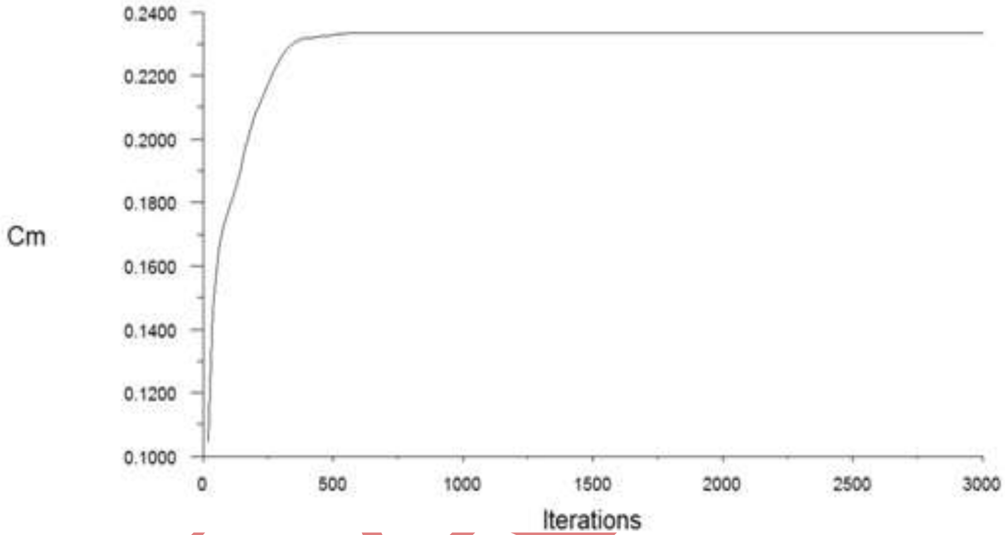
In contrast to this, if the software were to have predictive capability, one should be able to obtain good comparison with standard results with some default options for parameters in the code in a so-called blind shot, that is to say without the prior knowledge of the results. While more and more software are maturing for achieving predictive capability, there still will be some room for improving the codes in direction of predictive capability.

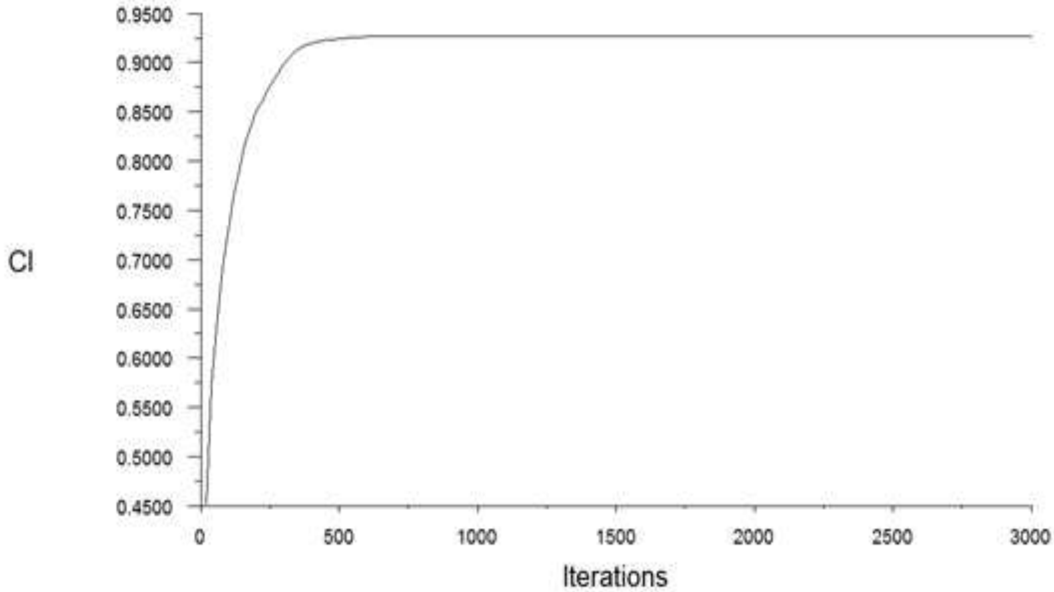
V. INFLUENCE OF THE FLOW DOMAIN

One of the most important parameters is the extent of the flow domain: for flow over an airfoil for example what should be the extent of the computational domain in x- and in y-directions. Of course the larger the domain the better are the chances of obtaining results for free stream flow conditions. But larger the computational extent more will be the execution time. Though this may not be critical for two dimensional flow simulation on modern computers, large grids may lead to problems of storage and execution times for three-dimensional flows. It is always good to choose the minimum domain that would offer interference-free results.

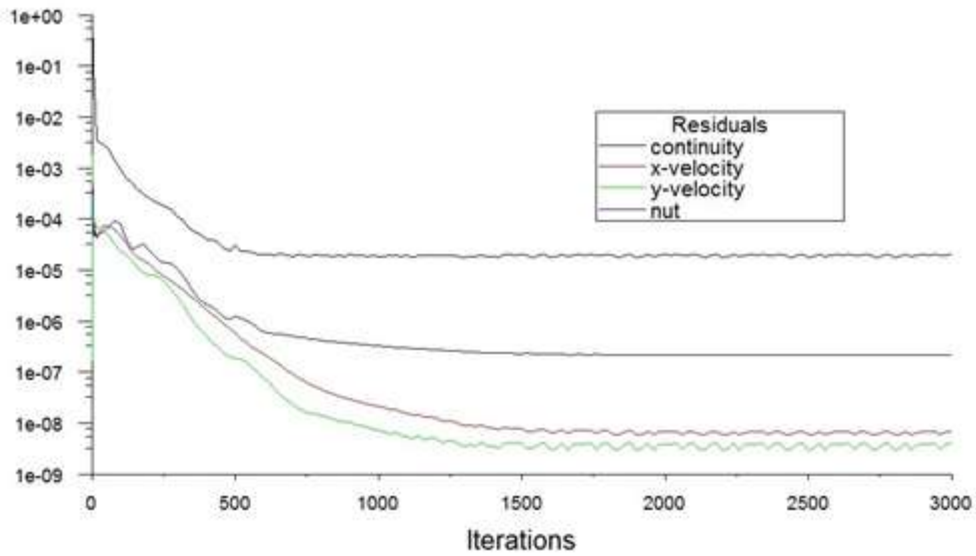

Cd
0.0650
0.0600
0.0550
0.0500
0.0450
0.0400
0.0350
0.0300
0
500
1000
1500
2000
2500
3000
Iterations

23015


Cm
0.2400
0.2200
0.2000
0.1800
0.1600
0.1400
0.1200
0.1000
0
500
1000
1500
2000
2500
3000
Iterations


Cl
0.9500
0.9000
0.8500
0.8000
0.7500
0.7000
0.6500
0.6000
0.5500
0.5000
0.4500
0
500
1000
1500
2000
2500
3000
Iterations

23015



23015

Fig. 6: Comparison of Numerical Results Obtained In FLUENT With the Standard Experimental Results

Fig. 6 shows a comparison of numerical results obtained with FLUENT with the standard experimental results of 23015. In Fig 7-9 we have shown the convergence history in terms of the fall of residuals in continuity, in x- and y-momentum and in numerical viscosity, and for lift drag and pitching moment.

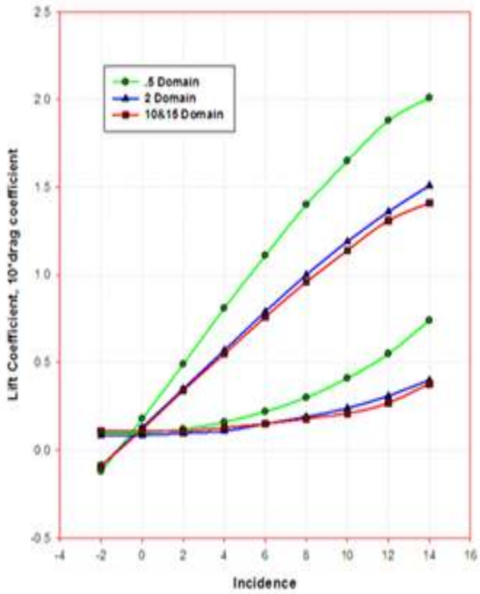


Fig.7: Influence of the Extent of Flow Domain on the Lift and Drag Characteristics of the Aerofoil

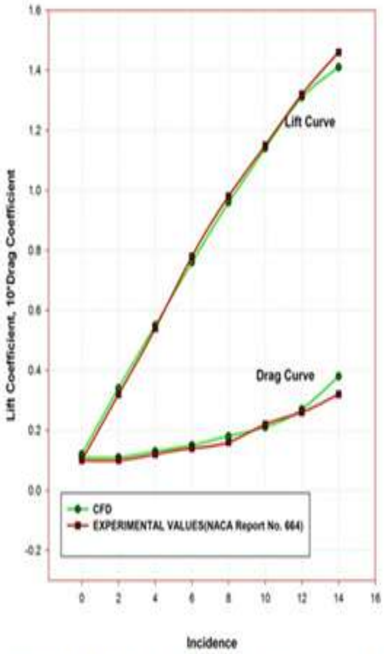


Fig. 8 Computed Lift and Drag Characteristics of basic NACA 23012 Aerofoil in Comaprison with Standard Experimental Results

VI. INFLUENCE OF MAXIMUM THICKNESS RATIO ON THE MAXIMUM VALUE OF LIFT COEFFICIENT

Fig.9 to fig.12 give the lifting Computed variation of lift with incidence for selected airfoils with the thickness to chord ratios of 12, 15, 18 and 21 percent respectively. In figures we have summarized the values of maximum lift coefficient as function of the airfoil thickness to chord ratio.

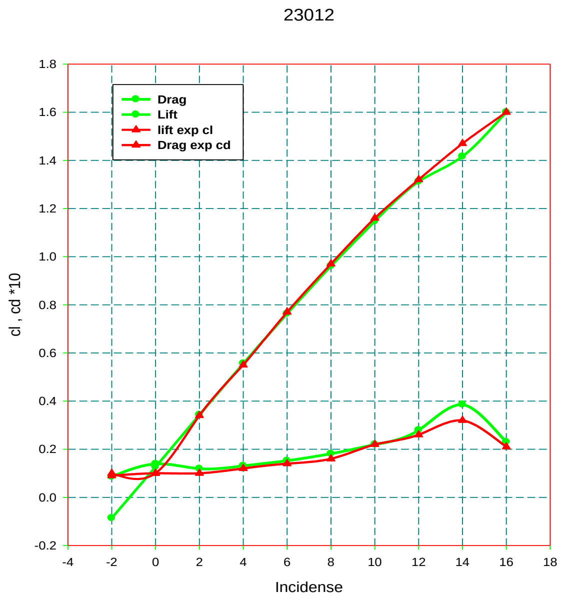


Fig 9: Variation for NACA 23012 airfoil

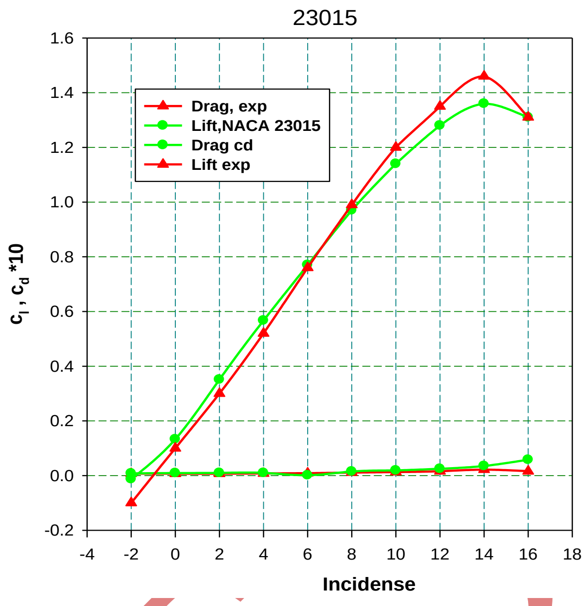


Fig 10: Variation for NACA 23015 airfoil

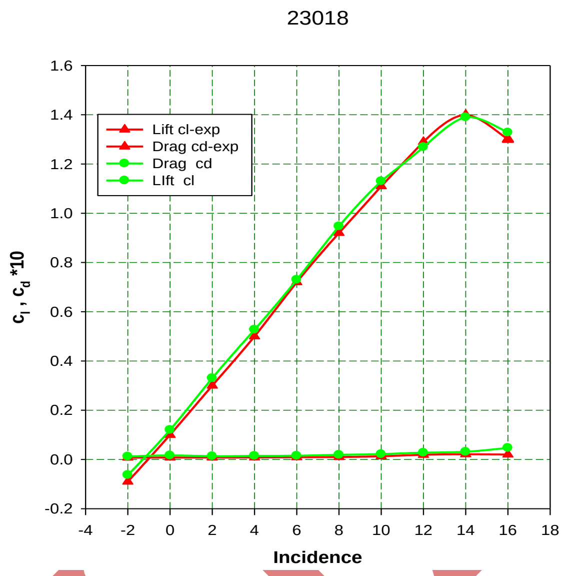


Fig 11: Variation for NACA 23018 airfoil

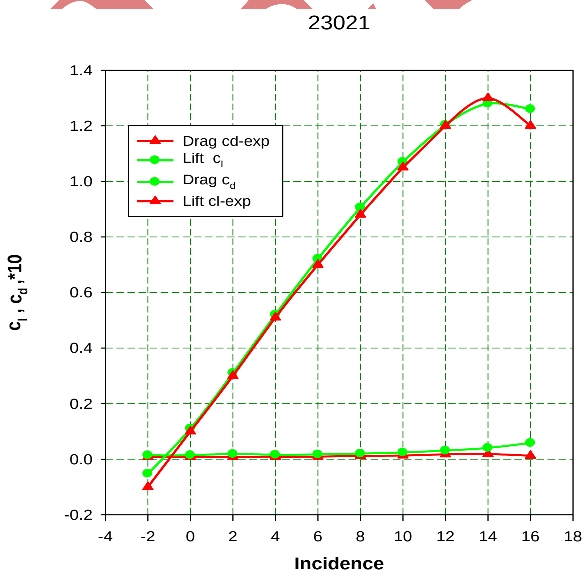


Fig 12: Variation for NACA 23021 Airfoil

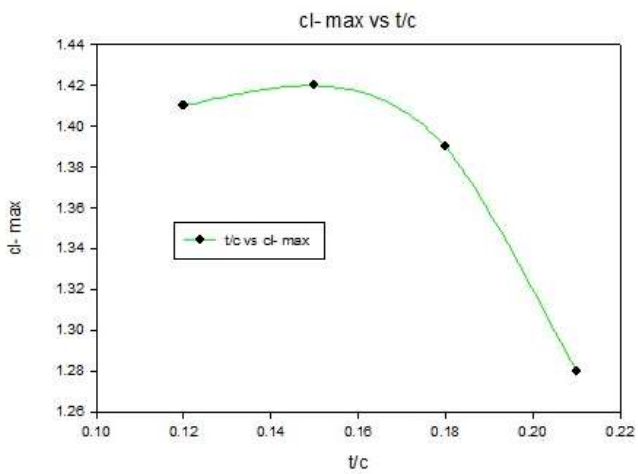


Fig 13: Computed Effect Of Thickness-To-Chord Ratio On Maximum Lift Of NACA 23012,15,18,21 Series Airfoils

It is instructive to compare fig.13 with fig.14 taken from Abbott and Doenhoff [19]. We notice from this graph that the maximum lift coefficient is obtained with thickness chord ratios on the order of 15 to 17 percent chord. This the conventional experience often quoted in open literature. Thus is established that the code FLUENT can indeed bring out the effects of airfoil thickness on its maximum lifting capability very well. In order to bring out the features of flow around the condition of maximum lift we have put together plots of pressure - coefficient (Cp) distributions in fig.14 to fig.17 and the corresponding flow patterns are shown in fig.19 and 20.

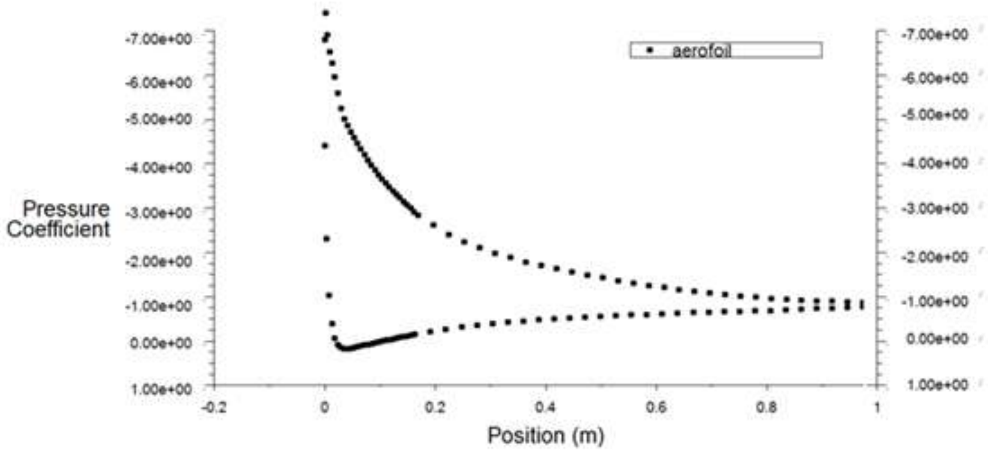


Fig 14: Computed Pressure Distribution for the NACA 23012 Airfoil for 14 Degree Incidence

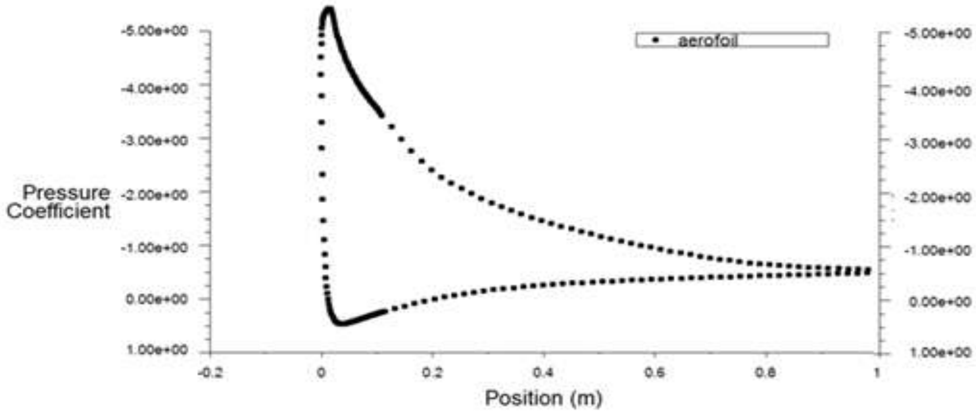


Fig 15: Computed pressure distribution for the NACA23015 airfoil for 14 degree incidence

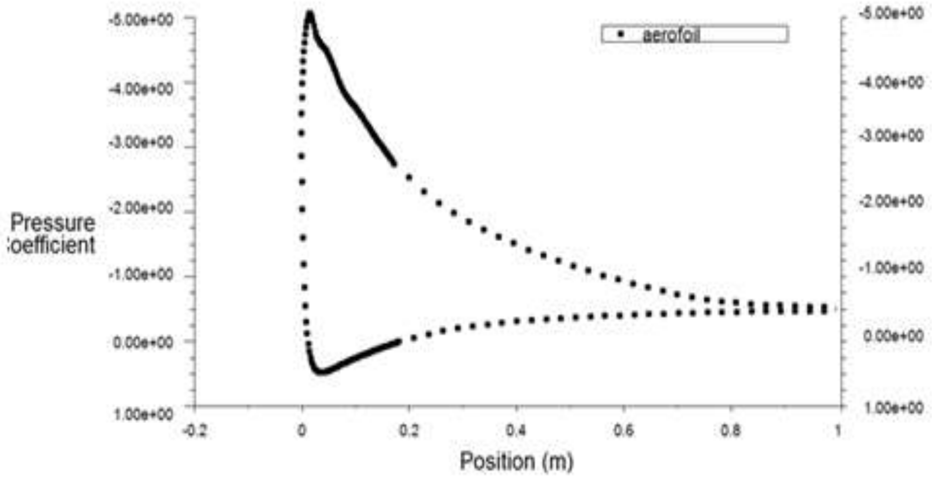


Fig 16: Computed pressure distribution for the NACA23018 airfoil for 14 degree incidence

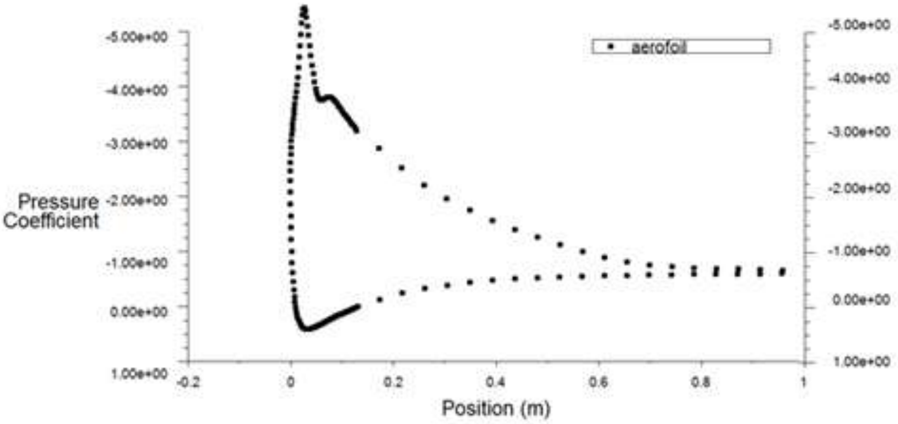
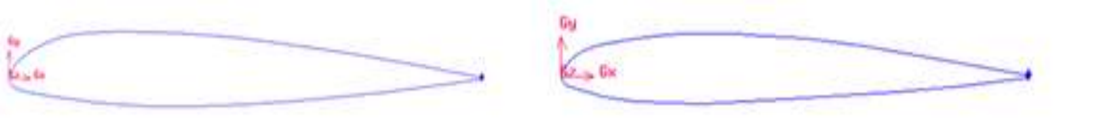


Fig 17: Computed pressure distribution for the NACA23021 airfoil for 14 degree incidence

VII. STUDY OF THE ROLE OF POSITION OF MAXIMUM CAMBER ON THE LIFTING CHARACTERISTICS OF NACA 23012 AIRFOIL

As mentioned before the lifting characteristics around the region of stall are important from the point of view of handling qualities of the aircraft. A stall that is abrupt is indeed dangerous in the handling of aircraft under slow speed conditions typial of the landing phase of an aircraft. The proximity of the ground requires that the aircraft does not suddenly loose the height due to abrupt stall of its wings. The wing characteristics follow by and large the characteristics of the airfoil that goes to make the wing. We desire an airfoil with docile or mild stall pattern to ensure safe handling characteristics for the aircraft. On the role of the position of maximum camber on the stall characteristics of the airfoil we quote the following from Kermode [20]. On page 93 of his well-known book he writes: "Generally speaking we get good all-round characteristics when the maximum camber is situated about 40 per cent of the chord back. Airfoils with the maximum camber well forward, say, 15 percent to 20 per cent of the chord, may have low drag but are apt to have poor stall characteristics – a rather sudden breakaway of the airflow". In order to verify this observation computationally we have taken the NACA 23012 airfoil and moved the position of its camber from its position at 15 per cent chord to 40 percent chord. In order to do this , first we extracted the camber line of NACA 23012 using GAMBIT. We started with the original airfoil and split its upper and lower 'edges' with constant percentage lines along the airfoil chord. We obtain the camber line as the locus of points located midway between the upper and the lower edge points. Next we moved the maximum camber point to a position at 40 per cent of the cord, thereafter we moved the other poits of the camber to produce a smooth camber curve with its ordinate at 40 per cent of the chord. After this step we superposed the thicness distribution typical of NACA 23012 airfoil over this new camber line to obtain a smooth new airfoil with its camber now at 40 per cent of the chord. The new airfoil is contrasted with the original airfoil as shown in fig.18.



Baseline NACA 23012 Aerofoil with its Maximum Chord Location at 15 Percent of Chord

Modified Aerofoil with the Maximum Camber Point Shifted to 40 Percent positon from its 15 Percent Position from NACA 23012

Fig 18: Comparison of the Basic NACA 23012 with a Derived Airfoil

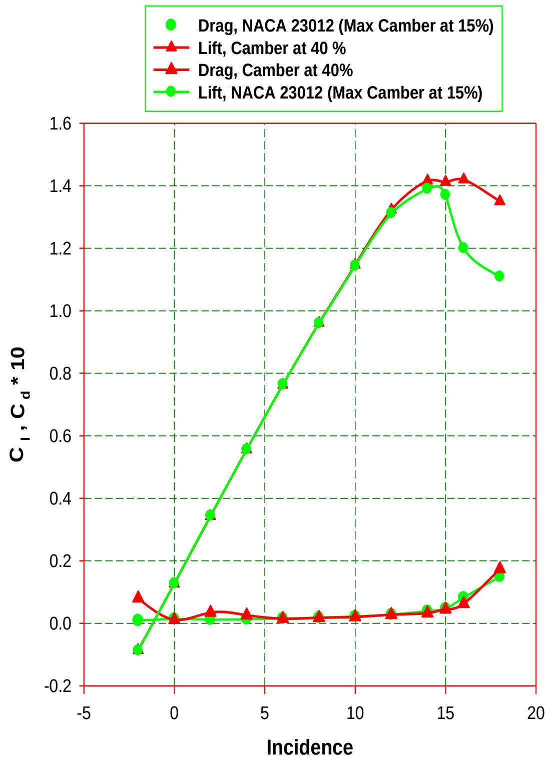


Fig.19: Lift & Drag characteristics of basic NACA airfoil Compared with modified 40% camber airfoil values

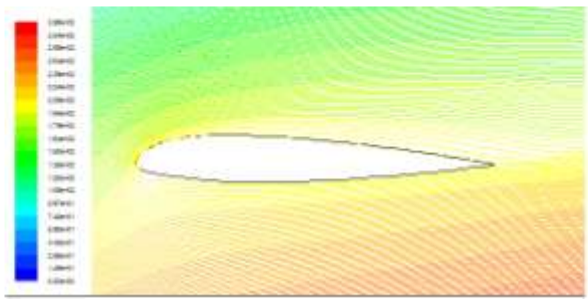


Fig. 20 (a): NACA 23012 steam flow lines

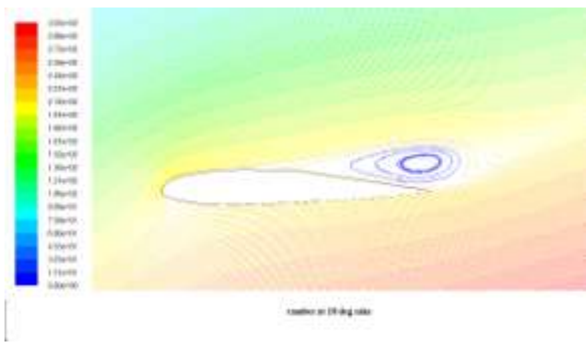


Fig 20 (b): 40% cambers at 18 deg

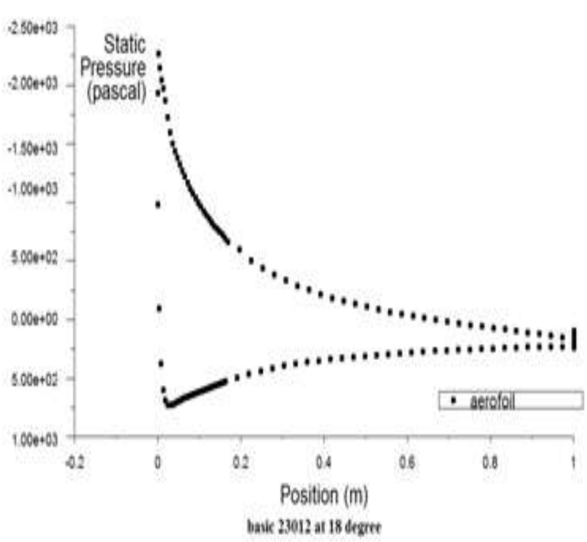


Fig. 20 (c): Cp for NACA 23012 at 18 deg

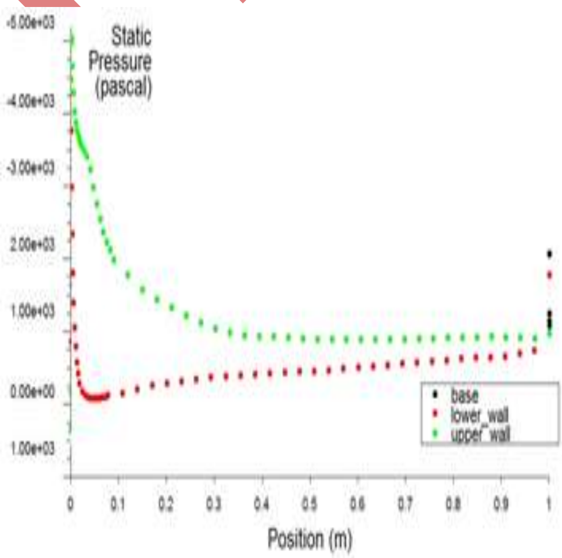


Fig. 20(d): Cp of 40% camber at 18 deg

VIII. STUDY OF NACA 64-110 AIRFOILS

Fig. 21 shows NACA 64110 airfoil. As mentioned above, the coordinates are imported to gambit and quadrilateral grids are created over sub-domains in to which main domain of fluid flow is divided to be specific. We have taken 10*15 dimensions for this domain.

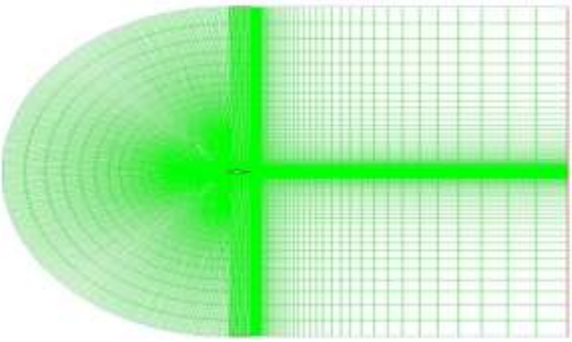


Fig 21(a): shows rectangular grids obtained using gambit

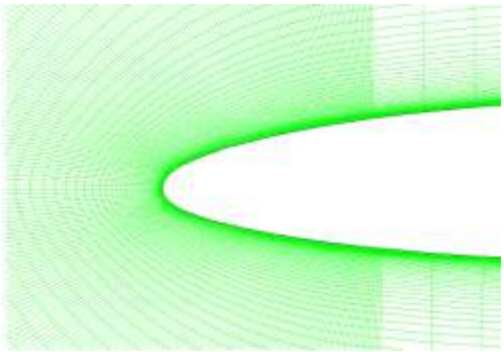


Fig 21(b): Close view of the grid at the leading edge

Fig 21(b) shows closely spaced grid around leading edge close to the body in order to pick up rapid changes taking place in this region. In our study we have chosen spalart allmaras one equation code for model. We have retained default options for parameters in this model. We use simple algorithm as flow model and we have used second order upwind finite differencing. There are several iterative methods available in fluent for solving simultaneous equations that results from discretisation of equation of conservation of motion from finite volume grid chosen. It's been our experience that converging results are obtained with relaxation factor continuity x and y and turbulence viscosity set as 0.3 and we also had chosen implicit choice as an available option in fluent. Fig 22 shows the lift and drag (L/D) characteristics of basic NACA 64110 airfoil compared with experimental value.

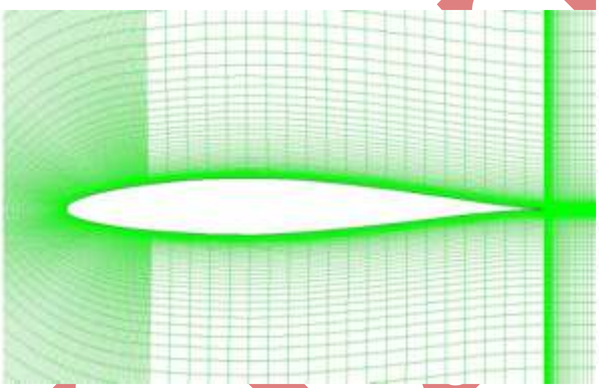


Fig 21(c): Close viewof the grid around NACA 64110 aerfoil

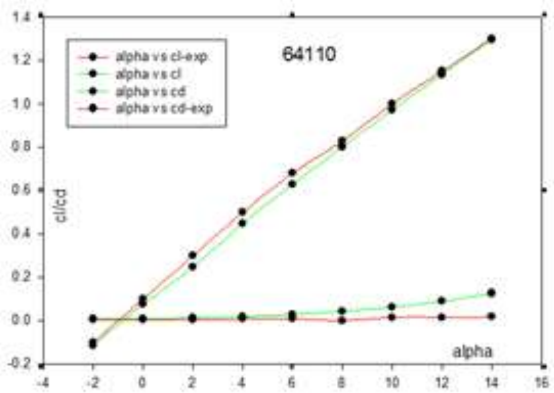


Fig 22: L/D characteristics of NACA 64110 airfoil compared with experimental value

IX. CONCLUDING REMARKS

In this study, we developed adequate insight into the working of the modern CFD software. We gained adequate familiarity with two software tools, the FLUENT, for flow analysis and the GAMBIT which goes with FLUENT, for geometry handling and for grid generation. Also we study the influence of thickness on the maximum lift characteristics of the airfoil. This study is carried out with NACA 23015, NACA23018 and NACA 23021 airfoils which have thickness of respectively 15, 18 and 21 percent of chord. Using the standard experimental results for the much studied classical airfoil NACA 23012 as a case for validation of FLUENT for study of viscous flow over airfoils through the entire working range of the airfoil, we have established the efficacy of FLUENT in proper simulation of flow field details and of quantitative data for the airfoil characteristics of lift, drag and pitching moment. The excellent post processing afforded by FLUENT has been used to obtain detailed pressure distributions and pictures of flow stream lines and contour maps. These have been of great value in our understanding of the physics of the flow. Equipped with the detailed validation of the software, we have applied the software in the study of role of thickness ratio of airfoils and of the position of camber in influencing the high lift behavior of airfoils. On the realization that the high-lift flow regime over airfoils governs the low-speed lifting characteristics of wings close to the region of stall – which is of paramount significance in the safe low-speed handling of any aircraft – we have studied this regime in terms of effect on it of the maximum thickness to chord ratio of the airfoil and also of the position of camber on the stall and the post stall characteristics. Through this study we have been able to affirm the well known results: (i) the maximum lift is attained from a given airfoil when its maximum thickness-to-chord ratio is in the range of 15 to 17 per cent; (ii) the stall behavior is governed by the position of maximum camber. A camber position of around 40 per cent results in smooth or docile stall. On the contrary putting the maximum camber position closer to the leading edge, say at 15 per cent as in the case of classical NACA 23012 airfoil, results in abrupt or violent stall. These are very important results generally well known, but these have been confirmed using the state-of-the art CFD tools. Here lies the value of CFD tools which good validation can be used for generating practical data on aircraft components in much cheaper manner than in the case of traditional expensive experimental methods.

X. ACKNOWLEDGEMENT

The authors gratefully acknowledge the contributions of Prof. Dr.S.S.Desai for his work on the original version of this document.

REFERENCES

[1] Anderson, J. Jr. Fundamentals of aerodynamics, 3rd edition, McGraw-Hill, 2001.

[2] Slawomir Koziel and Leifur Leifsson, “Multi-Fidelity High-Lift Aerodynamic Optimization of Single-Element Airfoils”, 2nd International Conference on Engineering Optimization, Lisbon, Portugal, September 6 - 9, 2010.

[3] Cody Lafountain, Kelly Cohen and Shaaban Abdallah, Camber Controlled Airfoil Design for Morphing UAV, 47th AIAA Aerospace Sciences Meeting Including The New Horizons Forum and Aerospace Exposition, Orlando, Florida, AIAA 2009-1435, 2009.

[4] Moore M, Wilson P A, and Peters AJ, An Investigation into Wing in Ground Effect Airfoil Geometry, University of Southampton, UK, 2008.

[5] Ranzenbach, R. and Barlow, J., Two-dimensional airfoil in ground effect, an experimental and computational study, SAE Paper No. 94-2509, 1994.

[6] Akira Oyama and Kozo Fujii, Airfoil Design Optimization for Airplane for Mars Exploration, Proceeding in the third China-Japan-Korea Joint Symposium on Optimization of Structural and Mechanical Systems, Kanazawa, Japan, 2004.

[7] Ranzenbach, R. and Barlow, J., Cambered airfoil in ground effect: Wind tunnel and road conditions, *AIAA Paper 95-1909*, 1995.

[8] http://www.aerospaceweb.org/question/airfoils/q0041.shtml accessed on 18.12.2014

[9] Mohammad Kouhi, Gabriel Bugeda, DongSeop Lee and Eugenio Oñate, Aerodynamic shape optimization using adaptive remeshing, 2nd International Conference on Engineering Optimization, Lisbon, Portugal, 2010.

[10] Frederik Zahle, Christian Bak, Niels N. Sørensen Tomas Vronsky, Nicholas Gaudern, Design of the LRP airfoil series using 2D CFD, *Journal of Physics: Conference Series 524* (012020) 2014.

[11] K.M.Pandey, S.Chakraborty, K.Deb, CFD Analysis of Flow through Compressor Cascade, *International Journal of Soft Computing and Engineering, 2(1)*, 2012, 362-371.

[12] Carl J.Wenzinger and Thomas A.Harris, Wind Tunnel Investigation of an NACA 23012 Airfoil with various arrangements of slotted flaps, NACA Report No. 664, 423 – 453.

[13] Q.H.Nagpurwala, S.Subaramu, M.D.Deshpande and S.R.Shankapal, Parametric Studies on the Aerodynamic Performance of a Miniature Propeller for MAV Application, 16th Annual CFD Symposium, Bangalore, 2014.

[14] Thomas R. Barrett, Neil W.Bressloft and Andy J.Keane, Airfoil Design and Optimization using Multi-Fidelity Analysis and Embedded Inverse Design, 47th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics and Materials Conf, Newport, Rhode Island, 2006.

[15] A. Firooz, M. Gadami,"Turbulence Flow for NACA 4412 in Unbounded Flow and Ground Effect with Different Turbulence Models and Two Ground Condition: Fixed and Moving Ground Conditions, Int. Conf. on Boundary and Interior Layers, 2006.

[16] http://www.ppart.de/aerodynamics/profiles/NACA4.html accessed on 18.12.2014.

[17] http://www.ae.illinois.edu/m-selig/ads/coord_database.html accessed on 18.12.2014.

[18] J. P. Vandoormaal, and G. D. Raithby, Enhancements of the Simple Method for Predicting Incompressible Fluid Flows, *Numerical Heat Transfer, 7*(2) 1984, 147-163.

[19] Abbott, Ira H. and von Doenhoff, A. E.: Theory of Wing Sections, Dover Publications, Inc., New York.

[20] Kermode, A.C. Mechanics of Flight, 8th Edition, Pitman Publications, 1972.