Adjoint based optimization of NACA 4412 Aerofoil using ANSYS Fluent

Kaushal Shrivastava[1] Prof. Sandeep Jainb[2] Dr. Lokesh Bajpaic[3]
[1]M. Tech. Student [2]Associate Professor [3]Head of Dept.
[1,2,3]Department of Mechanical Engineering
[1,2,3]Samrat Ashok Technological Institute, Vidisha 464001, (Madhya Pradesh) India

Abstract— Optimization of design for aeronautical problems by the help of computational fluid dynamics (CFD) is one of the active area of research in modern era. This paper is focussing on optimization of various design parameters like drag force, lift force, lift to drag ratio. The pressure variation and velocity variation over the aerofoil are also studied. Aerodynamic characteristics of National Advisory Committee for Aeronautics (NACA) 4412 aerofoil is taken here for optimization. The whole analysis is based on finite element method (FEM) and computational fluid dynamics. Adjoint theory is used here for optimization present in Fluent module. The airfoil of plane impacts more on its aerodynamic efficiency, so main objective of this paper is to optimize for high lift to drag ratio to maximize aerodynamic efficiency.

Key words: Adjoint, Fluent, CFD, Aerofoil, Shape Optimization

I. Objective

The objective of the present paper is to optimize a basic asymmetric airfoil NACA 4412 using Adjoint solver interfaced with the Ansys FLUENT analysis software. The coefficients of lift, drag and the lift to drag ratio of the original and optimized profile for same angle of attack is compared by using ANSYS CFX software.

II. Introduction

Computational fluid dynamics (CFD) models are used everywhere in aerodynamic design to make design more robust and efficient. Adjoint based shape optimization process is one of the best methods for optimization process. Practically, Adjoint is used to find the change in any engineering quantity like drag force, lift force, total pressure drop or any other parameter with respect to all the input parameters like the geometry or shape of the profile[1].

The adjoint solver calculates[2] the derivative of a single engineering quantity or observation with respect to a large number of input parameters simultaneously using a single computation. The engineering observation measures system performance, like the drag or lift on an aerofoil, or the total pressure drop in a system. In short, the derivatives with respect to the geometrical shape of the system are calculated by the use of adjoint.

The derivatives obtained, defines the degree of change of observable with respect to geometry which is the sensitivity[3] of the system. The most sensitive observable change more as compare to less sensitive observable on deforming the geometry of the profile. To optimize the profile, any one of the observable is chosen and solve for optimization.

III. Workflow of Adjoint

The solutions after simulations are found out, then a parameter is chosen which is of engineering interest and fed into the set-up which further solve the adjoint problem. Then, mesh is morphed on defining the solution control; after certain iterations a morphed mesh is obtained. Then post processing is done to get the shape sensitivity, various contours, plots and graphs. The flow chart of workflow of adjoint is shown below.

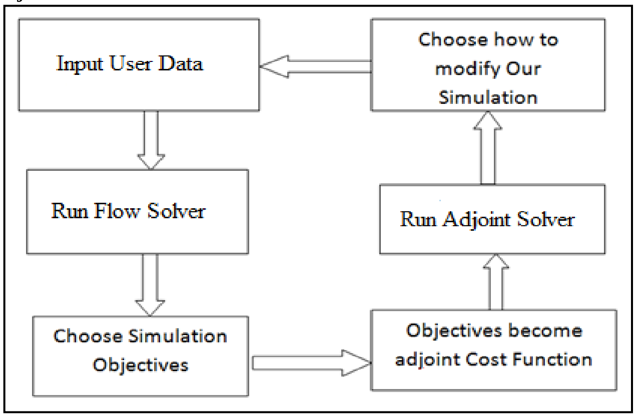

Fig. 1: Flow Chart of Adjoint Solver

IV. Ansys Fluent

CFD involves the use of computers to solve the system of governing equations that define the physics of the system. In this case, the Navier-Stokes equations[4] which simulate the dynamic interaction between the surface and fluid, is defined by prescribed boundary and initial conditions. Fluent software is one of many commercial solvers that uses finite element discretization to solve the system of equations.

There are the following basic steps[5] for obtaining the flow field in ANSYS FLUENT:

- Definition of geometry and discretization of the fluid volume around the geometry.
- Definition of physical model which states the governing system of equations.
- Definition of boundary and initial conditions.
- Solution the governing systems of partial differential equations and visualization of the results.

The governing equations are defined by the physics of the problem. In CFD analysis of the winglets, Fluent solve three governing equations[6] which are the equations for conservation of mass, momentum and energy.

V. Literature Review

Trejo-Vargas et al. [7] study about method for reduction in control points with same accuracy. The main theme of their study is creating a realistic method to define surface of wings according to parameters that are related to its performance. In this research, a B-spline curve for representation of airfoil geometry is used. By using a least square fitting method the control over a curve by using a reduced number of control points is being done. The weighted fitting method for representation of aerofoil is tested for different characteristics of aerofoil and this gives a good result while analysing.

Nitin Teguria et al. [8] worked on the generation of design of blade, based on Gluert Annulus momentum equation. Chord as parameter and twist distribution, type of air foil, tip speed ratio is selected. NACA airfoil of four and five series is chosen and analysed with diagram and different parameters, hence power co-efficient of airfoil NACA 4412 and NACA 23012 are higher than other airfoils can be concluded as conclusion. It is also concluded that co-efficient of power is dependent on the local tip speed ratio, axial interference faction, rotation interference factor, angle of relative wind from rotor plane, drag to lift ratio.

Amritpal Singh compared S1223 and NACA 4412 profiles at different angles of attack between 0^0 to 15^0 for different parameters by the use of CFD. The objective of this study is to find the coefficient of lift, drag, velocity and pressure variations at surrounding conditions with inlet velocity of 2.5 m/s at 30,000 as Reynolds number. The theories are explained and comparisons showed excellent agreement with the result.

Alex E Ockfen presented the NACA4412 aerofoil with flap in extreme ground effect using CFD FLUENT software. The objective of this study is to find the influence of flap in the viscous ground-effect.The turbulence model used is Spalart-Allmaras for steady state incompressible surrounding condition. Effects of Reynolds number, angle of attack, flap deflection and ground height are presented for split and plain flap. The conclusion of this study includes the use of flap for small angle of attack is advantageous for small flap deflection for 5% length of chord.

VI. FLUENT ANALYSIS FOR NACA 4412:

The FLUENT software is used to get the optimized profile for the NACA 4412. A notepad file with the some numerical coordinates of the NACA profile is fed as the input for the software and the result is obtained (shown in Fig. 2-Fig. 7). The Reynolds number of 1 million is used for analysis. The nodes and elements are 7282 and 3393 respectively in numbers and are generated in close vicinity of the profile so that focus remains on specific area. Pressure-Velocity scheme used is coupled with gradient based solver of least square cell based having momentum of second order upwind and turbulent kinetic energy and turbulent dissipation of first order upwind as spatial discretization. FLUENT solver is set at some spefic parameters which are tabulated in table 1.

Pressure velocity scheme	Coupled
Solver gradient	Least square cell based
Momentum	Second order upwind
Turbulent kinetic energy	First order upwind
Turbulent dissipation	First order upwind
Velocity	50 m/s
Density (kg/m^3)	1.225
Viscosity (kg/m-s)	1.7894

Table 1 FLUENT Solver Description

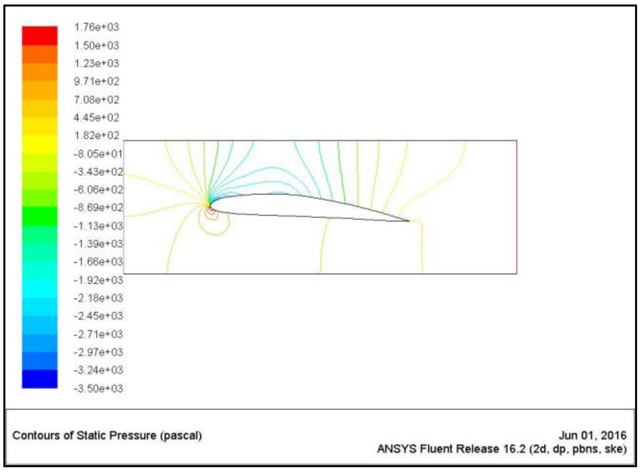

Fig. 2: Contours of Static Pressure

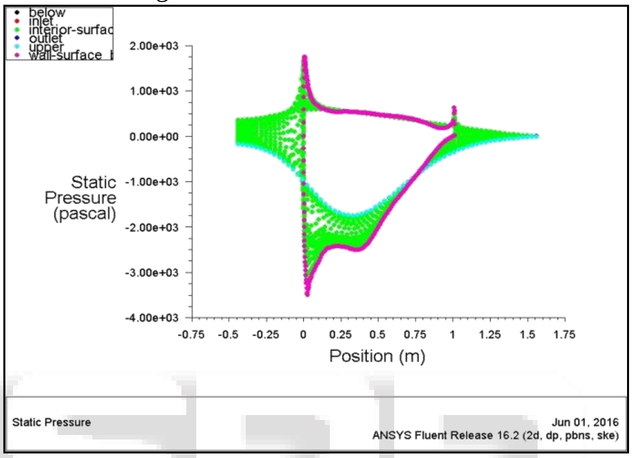

Fig. 3: Graph of Static Pressure wrt Position

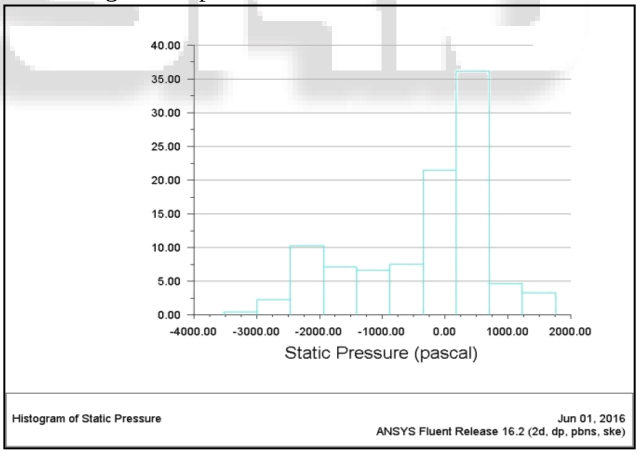

Fig. 4: Histogram of Static Pressure

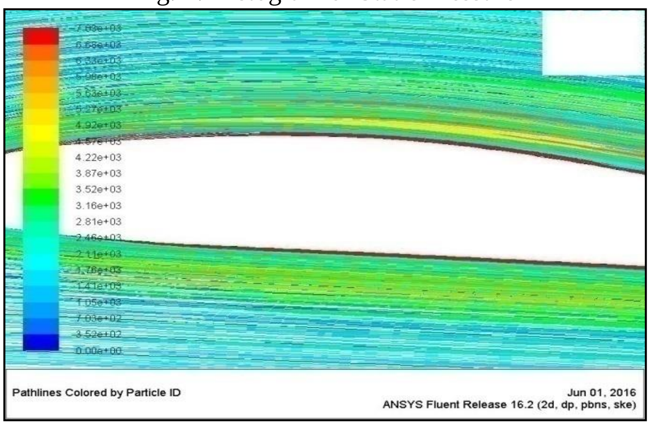

Fig. 5: Pathlines of Fluid Particles around airfoil

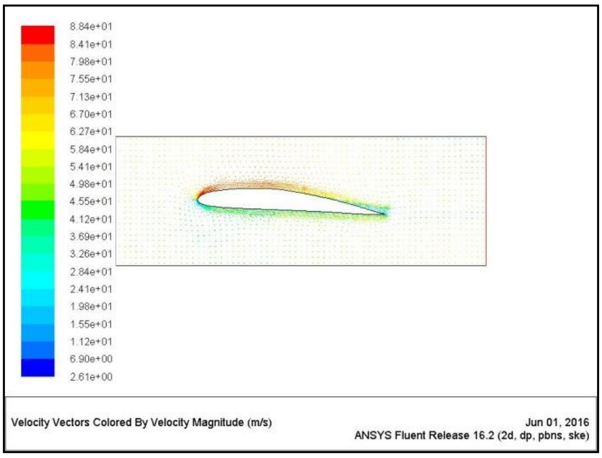

Fig. 6: Velocity Vectors around airfoil

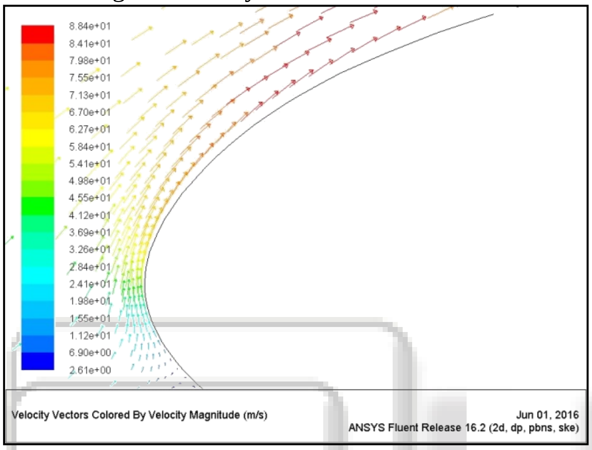

Fig. 7: Magnified velocity vectors

VII. Results and Discussions

From the comparison of analysis results for the NACA 4412 and the optimized profile, an increase of 0.3011 N and 0.4395 N in the lift force is obtained for 4^0 and 8^0 AOA respectively. There is very negligible difference in the drag coefficient values. The decrement in drag force is 0.00861 N and 0.009148 N for 4^0 and 8^0 AOA respectively. The lift to drag ratio, which is a cumulative parameter for determining the performance of the airfoil, has increased from 43.270 to 45.634 and from 38.777 to 40.490 for 4^0 and 8^0 AOA respectively.

VIII. Conclusion

From the pressure contour in Fig. 2, it can be noted that there is a low pressure region at the top of the airfoil which assists lift as air moves from a region of higher pressure to a region of lower pressure, hence pushing the airfoil on its way. The static pressure given by the ANSYS Fluent in Fig. 2 is the sum of gauge pressure and operational pressure with respect to positions. Therefore, negative pressure means the gauge pressure is 113 Pa below the operational pressure. The peak static pressure is 176 Pa which is at the leading edge of the airfoil is called stagnation point. At this point, the kinetic energy is completely converted into pressure energy. From the velocity vector in Fig. 7, it can be observed that the velocity at this point is the least. The velocity of the fluid is high above the airfoil, which is mainly due to the angle of attack. This aids the formation of a low pressure region at the top surface, which produces lift force in normal direction.

References

[1] Chalothorn Thumthae and Tawit Chitsomboon, "Optimal angle of attack for wing in subsonic flow for a fighter jet", Aeronautical Engineering, pp. 1279–1284. 2009.
[2] Dimitri J. Mavriplis, Brian A. Lockwood, "Adjoint-based Unsteady Airfoil Design Optimization with Application to Dynamic Stall", American Helicopter Society 68th Annual Forum, pp 1-3.2012.
[3] Ozge Polat and Ismail H. Tuncer "Aerodynamic Shape Optimization of Wind Turbine blade using a Parallel Genetic Algorithm", Procedia Engineering pp.28-31. 2013.
[4] Michael B. Giles and Niles A. Pierce, "An Introduction to the Adjoint Approach to Design". Flow, Turbulence and Combustion 65, Kluwer Academic Publishers, pp 393–415. 2000.
[5] Domenic Bucco, "Aerospace Applications of Adjoint Theory". Defence Science and Technology Organisation-Technical Report, pp 1-37. 2010.
[6] Xiongwei Liu, Lin Wang, Xinzi Tang "Optimized linearization of chord and twist angle profiles for fixed-pitch fixed speed wind turbine blades", Renewable Energy 57. pp 111-119. 2013.
[7] F. Pérez-Arribas and I. Trejo-Vargas. " Computer aided design of parametrically designed wing", Parametrical design of airplane 44. pp 252-260.2012.
[8] Nitin Tenguria, N. D. Mittal and Siraj Ahmed."Evaluation Of Performance Of Horizontal Axis Wind Turbine Blades Based On Optimal Rotor Theory",Journal of Urban and Environmental Engineering, v.5. pp 5-23.201