

CFD-Based Shape Optimization for Optimal Aerodynamic Design

Asya Gabbasa

Badih Ali Jawad
Lawrence Technological Univ.

Evangelos Koutsavdis
ANSYS Inc.

ABSTRACT

Increased energy costs make optimal aerodynamic design even more critical today as even small improvements in aerodynamic performance can result in significant savings in fuel costs. Energy conscious industries like transportation (aviation and ground based) are particularly affected.

There have been a number of different optimization methods, some of which require geometrically parameterized models. For non-parameterized models (as it is the case often in reality where models and shapes are very complex). Shape optimization and adjoint solvers are some of the latest approaches.

In our study we are focusing on generating best practices and investigating different strategies of employing the commercially available shape optimizer tool from ANSYS/CFD solver Fluent. The shape optimizer is based on a polynomial mesh-morphing algorithm. The simple case of a low speed, airfoil/flap combination is used as a case study with the objective being the lift to drag ratio.

A number of different built-in optimization algorithms and the way to best employ them are investigated.

CITATION: Gabbasa, A., Jawad, B. and Koutsavdis, E., "CFD-Based Shape Optimization for Optimal Aerodynamic Design," *SAE Int. J. Passeng. Cars - Electron. Electr. Syst.* 5(1):2012, doi:10.4271/2012-01-0507.

INTRODUCTION

Computational fluid dynamics is an application of computer aided analysis for analyzing fluids and their properties. It utilizes different approaches of algorithms and fluid mechanics formulae for the same purpose. There is different software like ANSYS which supports the feature of fluid mechanics so that a fluid interaction with the surfaces and given boundary conditions can be easily simulated and predicted in the analysis phase of a design. There are different equations like Navier Equation, RANS equation and different meshing techniques like finite difference and finite volume that are used in CFD for fluid analysis. Other meshing schemes are also there like FEM and BEM but they are not used for fluid analysis (Anderson, 1995).

The flow behavior in terms of velocity field for a *non-turbulent, Newtonian fluid* is governed by the Navier-Stokes equations. These equations are valid only for the *moving flow* as it describes the flow field. The Navier-Stokes equations are obtained by combining the fluid kinematics and constitutive

relation into the fluid equation of motion, and eliminating some higher order tensors. Navier-stroke equation is applicable for any laminar or transient and non viscous flow passing through passage of any geometry (Tannehill, 1997).

Generally;

For Compressible fluid

$$\nabla p + \mu(\nabla^2 \cdot v) + \frac{1}{3} \cdot \mu \cdot (\nabla(\nabla \cdot v)) + \rho b = \rho \cdot \dot{v}$$

For Incompressible fluid

$$-\nabla p + \mu(\nabla^2 \cdot v) + \rho b = \rho \cdot \dot{v}$$

Reynolds-averaged Navier-Stokes (RANS) equations are the oldest methodology to *model turbulent flow*. For the turbulent flow it introduces an additional apparent stresses called *Reynolds stresses*. These equations are time-averaged equations but are equally valid for the flows with a time-varying mean flow.



$$\rho \frac{\partial y_{ij} u_i}{\partial x_j} - \rho f + \frac{\partial}{\partial x_j} \left[-p \frac{\delta_{ij}}{1} + M \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] - \rho u_j^2 u_i^2$$

Zingg *et al* (2002) exclaims Computation fluid dynamics as moving towards considerably minimizing user proficiency needs. The outcomes of the RANS equations, acronym of Reynolds-averaged Navier-Stokes equations, in the arena of computational fluid dynamics (CFD) to aircraft proposal simulation & design, comprises of issues in the subsequent three areas. These are computational efficiency, human efficiency and accuracy of physical modeling.

Computational Efficiency: There has to be lessening in the computing time needed to attain suitably determined solutions. This is an imperative requirement in sleek aerodynamic and multi-purpose design customization, as a result of the drift focusing a concurrent product and process advanced manufacturing. For comprehensive incorporation into the design procedures, the time necessary for solution of the RANS equations should be limited to only some minutes, over three-dimensional simulations, which is two orders of degree quicker than capacity, a fresh improvement at the beginning of this era. Even though there is a specific benefit of parallel architectures and augmented computer processing velocity, there is a requirement for enhancement in algorithms. In design customization as well, algorithm trustworthiness has larger implication. An explicit narration of Zingg *et al* (2002) is articulated as “Modern design optimization an algorithm, such as adjoint methods cannot be effective if the flow solver does not converge in relevant areas of the design space”.

Human Efficiency: There is a vital want for a diminution in the human endeavor and proficiency needed for computing flows over complicated configurations in view of the fact that humans are not administered by Moore's law (Zingg *et al*, 2002), Kaku (1999) demonstrated that computer power increased to two times the prior one after each eighteen months, and the speedy boost in power on this level is a single and marvelous development in the account of technology. However proficiency in CFD and aerodynamics is essential for moving out with the computations, there should be a decrement in the proficiency needed in the assortment of solver and grid constraints. One more problem to be addressed is the judgment of global numerical error.

Accurateness of Physical Modeling: In the course of technique for example suitable grid resolution, numerical errors can be cautiously manipulated. On the other hand, errors coming out from physical models, as well as turbulence models and calculation of laminar turbulent alteration are supplementary tricky to calculate and administer (Zingg *et al*, 2002). The eddy-viscosity turbulence models utilized at the moment for processing aerodynamic flows are not capable to envisage with accuracy, slight occurrence for instance Reynolds number or flap gap effects on high-lift customizations.

The main finest methodology emerged to be Reynolds stress models or second moment closures. It is vital to

significantly augment the speed with which similar models are included into aerodynamic flow solvers, for manufacturing utilized at this point, and to judge and direct the manufacturing of such models. Additionally manufacturing of turbulence models needs diminution of time required processing three dimensional flows; moreover there is a want for further elevated class tentative datasets which comprise all the boundary condition data required for processing. To augment the computational effectiveness of RANS solvers, two fresh algorithms are taken into account: an imprecise Newton-Krylo algorithm which minimized the time required for processing & computing in order to attain a steady state solution; and the other is an upper order spatial discretization which minimizes the grid resolution requirements for a certain point of arithmetical accuracy (Zingg *et al*, 2002), as a result in addition minimizing computing expense. Furthermore, objective procedures of algorithm presentation are needed.

RESEARCH STRATEGY PLAN

The plan of this research strategy is to produce an airfoil design and optimization mechanism that can adapt an airfoil shape (section geometry) delivering enhanced aerodynamic performance in terms of maximizing lift to drag ratio under landing and takeoff flight situation.

Processing the airfoil optimization development can significantly reduce manufacturing product time duration and fetch into view better designs in contrast with the traditional planned design modification methodologies. A well-organized and efficient optimization method is initiated by an amalgamation of high-fidelity commercial CFD tools with numerical optimization methodologies. This organism not only presents a steady, computerized optimization device for design engineers, but furthermore significantly minimizes the expenditure and the production time for a design process. The plan will be to unite the high fidelity commercial CFD tools (Fluent) with mathematical optimization systems to morph high lift organization. In research plan as shown in figure (1) we carry out morphing (grid deformation) openly within the fluent code devoid of reconstruction of geometry and the mesh with a peripheral utensil. Direct search method algorithms for example the Simplex, Compass, and Torczon...etc. is examined. The designer can prefer one of those to use in optimization of aerodynamic shapes distinctiveness.

FORMULATION

A. FLOW SOLVER

FLUENT is a computational fluid dynamics (CFD) solver. It is a software package to simulate fluid flow problems. It is based on a finite-volume method to solve the governing equations for a fluid. It provides the capability to use different physical models such as incompressible or compressible, inviscid or viscous, laminar or turbulent, steady- state or transient analyses.



In the CFD formulation as well, the conservative forms of continuity and Navier-Stoke's equations in integral form for incompressible flow of constant viscosity were solved by the built-in functions of the Fluent 13 CFD software. The current work used two equation turbulent models: the realizable k-e model that solves one transport equation to allow the turbulent kinetic energy, and its dissipation rate to be independently determined. The realizable k-e model is particularly suitable for our model, as the model uses enhanced wall treatment based on the law of the wall.

In this paper, steady-state, incompressible two dimensional flow was assumed. The numerical simulations were carried out by solving the conservation equations for mass and momentum by using structured grid finite volume methodology.

The sequential algorithm, semi-implicit method for pressure linked equation (SIMPLE), was used in solving all the scalar variables. For the convective terms of the continuity and momentum equations, and also for the turbulence equations, the first order upwind interpolating scheme was applied in order to achieve more accurate results.

B. DESIGN VARIABLES

As revealed from [figure \(2\)](#), the overlap and the gap flanked by the flap and main airfoil are utilized as the design variables. The control points as shown in [figure \(3\)](#) are situated around the area of the gap and the overlap section. Thus, the shape optimization, i.e., expansion, retraction, etc., are imposed in these area in order to achieve optimal values of L/D ratio.

C. OBJECTIVE FUNCTIONS

A single-objective function can be stated as:

Find X which

$$\text{Minimize } f(X) = [f_1(X)/f_2(X)]$$

The symbols f_1, f_2 are drag and lift correspondingly, X is known as the design vectors.

The vector of design variables, X , principally includes factors that manipulate the shape of the airfoil. Relying on the problem of concern, supplementary design variables may comprise of the angle of attack, the parallel and perpendicular transformation design variables that administer the place of flaps in multi-element configurations as shown in [figure \(2\)](#).

In this work, the objective function is processed by C++ language encoding & programming and constructed on the User Defined Function (UDF) library which considers control points and angle of attack as input factors, and value of drag to lift ratio is output result.

D. DIFFERENT STRATEGIES IN OPTIMIZATION

There are different strategies of optimization studied in this paper. The strategies are the following:

1-. At different algorithms

2-. At different position of control points

3-. At different number of control points

Simulations are executed using each of these strategies. The resulting L/D ratios are closely monitored. Comparisons between before and after optimization scenarios are noted. Each strategy is evaluated of its effectiveness in improving the performance of the system. Also, analogous strategies are compared to each other.

RESULTS AND DISCUSSION

AT DIFFERENT ALGORITHMS

In this part we demonstrate the persuance of algorithms on assessment of L/D ratio at different angle of attack. As shown from [figure \(4\)](#), all data sets point out a virtually indistinguishable lift to drag ratio numerical range at (-4 to 0) degrees angle of attack and the data carries on to be identical fairly fine at (6 to 20) degrees. There are little deviances ranges at angle of attack from (0 to 6) degrees. The [figure \(4\)](#) below shows that the Torczon and simplex algorithms are yielding the utmost lift to drag ratio, at the same time as the compass algorithm is yielding the lowest value.

Those deviations occur from the particular performance for every algorithm; the disparity between the various algorithms of the direct search method are mostly in the choice of the step length and search direction; such as at compass algorithm, first select an initial point (x_0, y_0) for 2 dimension, a preliminary step length Δ , compute the objective function at preliminary point then at east, west, north, south correspondingly, if among these phases of input produces a lesser $f(x, y)$ for 2D, there is new iteration (x_1, y_1) as revealed in [figure \(5\)](#). The computation phases carry on by the similar preliminary step length and search direction until converge optimal shape, at the same time as the step length and search direction are variables relied on objective function numerical equivalent at Torczon and simplex algorithms.

As it was identified previously in description the compass algorithm has a small number of steps if related with remaining algorithms for example the simplex and Torczon algorithms. They require additional steps to move toward a lowest amount of objective as revealed. This reason the compass algorithm simple to explain, trouble-free to put into practice and may speedily move toward a solution, but may be sluggish to perceive it, if the step size is outsized.

This appears evident from current work, the amount of designs approach at compass algorithm are below to relative algorithms; for instance at angle of attack (0 to 4) degrees the maximum number of designs were thirty one & thirty two correspondingly at compass algorithm whilst at Torczon were forty & forty one at 0 and 4 degrees correspondingly and at simplex algorithm were forty & forty four for (0 & 4) degree correspondingly. This explicates the meeting of the solution for compass algorithm is quick.

As it is identified earlier the objective of present work was to enhance the numerical value of L/D ratio at landing and take-off flight situations by aerodynamic shape optimization by utilizing grid deformation.

As shown from [figures \(6,7,8\)](#), appears evident that the optimization mechanism has been victorious in attaining the objective. When the contrast between L/D ratio before and after optimization, it is seen obviously how augmented values of L/D ratio throughout the angles of the attack differ from (-4 to 20) degrees devoid of exemption. It was the uppermost percentage of enhancement of 33.919% for the duration of angle of attack four degrees at simplex algorithm, and the minimum percentage of enhancement was 6.925% for the duration of 0 degree angle of attack at compass algorithm, as shown in [appendix](#).

[Figures \(9,10,11\)](#) reveal the final outform for airfoil after optimization manipulations.

[Figures \(9,10,11\)](#) reveal the manipulation of diverse algorithms on ultimate shape optimization. As it is identified earlier that each algorithms has particular performance and search method therefore the compass algorithm is bounded search direction and length size, while the simplex and Torczon algorithms are unbounded length size and search direction that is appeared clearly from [figures \(10-11\)](#) the remaining both algorithms are produced just about the identical optimal shape.

AT DIFFERENT POSITION OF THE CONTROL POINTS

This strategy employed in the design is changing the position of the control points. The control points are critical in the design of an aerodynamic structure since they determine the factors in the system to be measured. Three locations of the control points are examined in the study. The position (A, B and C) are illustrated in [figure\(12\)](#).

Location A is directly in front of the flap, location B is above the flap directly below the overcamber of the flap and location C is above the flap and is quite far from the airfoil. These locations are chosen strategically so as to achieve different measurements of L/D ratio. Also to check the critical positions where in manipulations of the parameters in these positions could possibly translate to changes in the performance of the structure.

[Figure \(13\)](#) shows the L/D ratio graph against the angle of attack. Measurements are extended up until angle of attack=16 degrees. The results of the three locations are all plotted in one graph.

This figure shows that the L/D ratio of locations A, B, and C across the majority of the angle measures, are virtually the same, that means the changing of control points location does not affect on values of L/D ratios.

AT DIFFERENT NUMBER OF CONTROL POINTS

The third strategy in this study is changing the number of control points. Intuitively, this criterion is affected the performance of the airfoil structure since more system points are manipulated throughout the process ([Gill,1981](#)).

There are three cases. The first case is using (4) points; the second case is using (8) points; the third case is using (12) points. The L/D ratio as shown in [figure\(14\)](#) is plotted against the angle of attack up to 20 degrees.

The [figure \(14\)](#) shows the results for different numbers of control points. At negative angles, the values of the L/D ratio of the three cases are virtually the same. At angles in the range [2,8] degrees, the third case (12 points) is slightly higher than the other two, also at this range, the third case achieved L/D ratio greater than 25. At angles above 10 degrees, the L/D ratios of the three cases are virtually the same.

The plot shows a pattern implies that the L/D ratio is slightly increased when the number of control points is increased. This pattern may be further verified by testing larger number of control points. The changes may be insignificant in value, but these miniscule changes would translate to higher energy efficiency. Thus, it can be concluded that the number of control points would affect the performance of the aerodynamic design.

SUMMARY

In this optimal aerodynamic design study, a number of different analyses are performed. The report initiates with the introduction of computational fluid dynamics, where the case object is also a bit discussed along with different strategies. Then for the problem set up we analyzed different equations that will be used by the Ansys software for the analysis of the given aerofoil optimized case. This includes three different strategies at different algorithms, different location of control points and different number of control points to assess and optimize the L/D ratio.

This study has tested and proven the effectiveness of three different optimization strategies. Changing the number of control points affects the L/D ratio of the system. Compass, simplex, and Torczon algorithms are even more effective in optimizing the design. Combinations of these strategies could be a possible venture in the future of further optimization of aerodynamic systems. Finally through the merger of different strategies utilized the target is achieved.

NOMENCLATURE

CFD

Computational Fluid Dynamic

μ

Flow viscosity

P

Pressure

ρ

Air density



Documents

Document Sharing Hub