

INVERSE DESIGN OF HIGH LIFT SYSTEMS

David Jones, Ian Fejtek
 djones@dehavilland.ca, ifejtek@dehavilland.ca
 Bombardier Aerospace, 123 Garratt Boulevard,
 Downsview, Ontario, M3K 1Y5
 Canada

Keywords *Inverse Design. High-Lift Systems. Flap Design*

Abstract

The design of wings for new aircraft is a time-consuming and challenging task. This is particularly the case when the design needs to incorporate high lift systems which also require design. This paper presents a method that was originally developed for the design of wings in cruise configuration. This method has been extended to the design of the individual components of high lift systems, in particular the flap, using an inverse design code based on a transonic small potential method.

1 Introduction

Computational Fluid Dynamics is now an essential tool in the design of modern aircraft. CFD methods are used to compliment wind tunnel testing, and in many cases, can be used to reduce the number of wind tunnel tests required in the design of new configurations, thus considerably reducing the cost of developing these new designs. The need to design new aircraft quickly and reliably, in order to beat the competition to the market place, puts increasing demand on the CFD methods that are used to design the wing, and other aircraft components. One CFD tool that can be used to assist in the design of wings is the inverse design method. An inverse design method is one that, given a set of target pressures, calculates the geometry that would produce such a pressure distribution.

There are many inverse design and optimization methods that are currently used to design the wing to meet specified lift and drag criteria. Many of these methods have been developed to design the wing at cruise conditions. Attention

is now being turned to the design and optimization of the high lift systems that are deployed during landing and takeoff [1,2]. The design of these lifting elements usually can only occur over a small part of the surface, the rest of the surface shape being dictated by the design of the wing at cruise, with the high lift elements stowed. Thus it is particularly important that the inverse design and optimization methods be sufficiently accurate in the region of the leading edges, for example, of the flap and wing-under-slat surfaces.

Another important aspect to wing inverse design methods is that they should be capable of designing the wing in the presence of other aircraft components, since it is well known that the effect of other aircraft components can have significant influence on the wing. This paper describes an inverse design method that can design the wing in the presence of other components. Thus, this method is ideally suited for the design of individual components in a high-lift system, such as the flap.

The inverse design method presented here is a modular program, in that it can quickly and easily be coupled to alternate flow solvers. The method has previously been used for the design of transonic wings [3], and wing-mounted engine nacelles [4]. In this paper, the use of the inverse design method for high-lift component design is investigated. This investigation has been done in both 2-D and 3-D using a 2-D Navier Stokes flow solver and a 3-D Euler solver.

2 Description of Method

2.1 The Inverse Design Method

The inverse design method used in this investigation is a technique that creates the surface shape from a prescribed pressure distribution and an initial design shape. The method was originally developed and implemented by Takanashi[5]. This code was developed for 3-D wing design, and is an iterative method based on the solution of a set of integral equations derived from the transonic small disturbance equation.

The implementation of this method at Bombardier Aerospace is called INDES. The inverse design method is actually a suite of three programs consisting of INDES, together with a pre- and post-processor. The

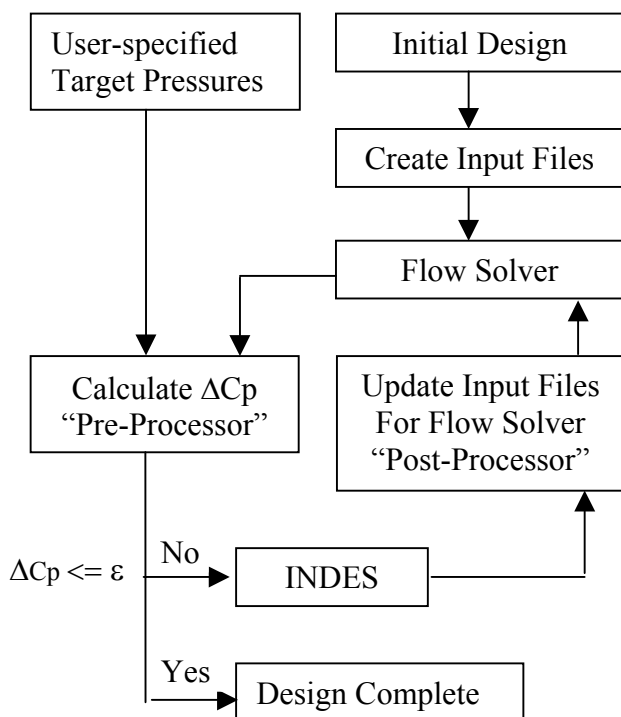


Fig. 1. Flow Chart illustrating the Design Process

pre-processor is used to extract the pressures on the design component from the flow solver output, and to calculate the difference between these computed pressures and the target pressures. The INDES code is given this

pressure difference together with the current geometry. INDES then computes corrections to the current geometry, based on the solution of the discretized integral equations. These geometric corrections take the form of perturbations to the chordwise camber and thickness distributions at each wing design station. The post-processor code is used to create new input files for the flow solver based on this modified geometry.

Various improvements have been made, at Bombardier, to the original inverse design method including the use of a smoothing algorithm for both chordwise and spanwise perturbations to the geometry. This smoothing method is based on de Casteljau's algorithm which uses a Bezier curve fitting routine[6]. A dissipation term has also been added to the governing equations for high subsonic Mach number cases. These improvements were originally suggested in a paper by Bartelheimer[7]. Of course, in this investigation, the dissipation term for high subsonic Mach number cases was not required and therefore not used, although the smoothing algorithm was utilized. It has been shown [3,8] that the use of this smoothing algorithm is very beneficial.

Figure 1 illustrates the iterative design process, starting with an initial design and culminating in a wing that produces the required target pressures. Typically in previous investigations with the INDES code, 5-10 iterations of this design process have proven sufficient to obtain the final geometry that produces pressure distributions matching the target pressures to within a pre-specified tolerance. It should be noted that, although the inverse design process will modify the sectional shape at each spanwise station, it will not modify the planform, the location of the leading edge or the blunt trailing edge thickness.

The inverse design method has been coupled to several different flow solvers including a 3-D Navier Stokes flow solver, LANS3D, a 3-D Euler code, MGAERO[3,9], a transonic small perturbation method, KTRAN[8], a 3-D panel

code, VSAERO, a 2-D Navier Stokes flow solver, and a 3-D full potential code, FLO22. In this investigation, 2 flow solvers will be used, the 3-D Euler Code MGAERO, and the 2-D Navier Stokes code TORNADO.

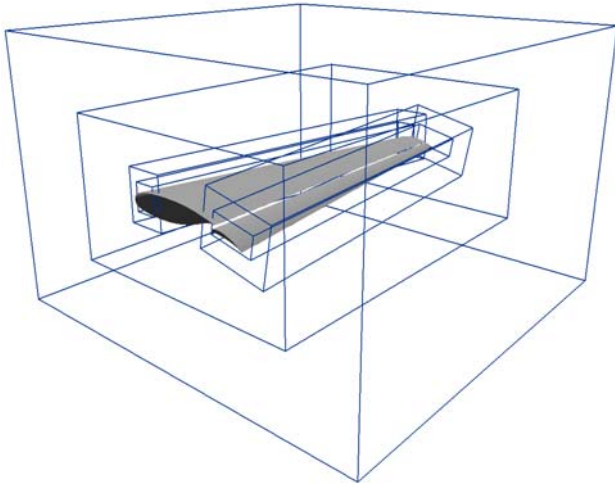


Fig. 2. Illustration of MGAERO Grid Blocks around Tapered Wing/Flap Configuration

2.2 The Flow Solvers

The 3-D flow solver used in this investigation was MGAERO[10]. MGAERO is a commercial CFD code currently used by a number of aircraft companies throughout the world. MGAERO solves the 3-D Euler equations using an explicit, finite difference method applied to multiple Cartesian meshes. The discretized equations are solved using a modified 4-stage Runge-Kutta scheme. Second-order/fourth-order dissipation and multigrid convergence acceleration are employed. MGAERO also includes a boundary layer method that has been coupled to the flow solver, although this feature was not used in this exercise. Improvements to drag prediction have also been implemented at Bombardier[11].

MGAERO does not use body-fitted grids; instead it uses Cartesian mesh blocks which are relatively quick and easy to generate. This

makes MGAERO an excellent tool in an aircraft design office, since new models can quickly be modeled, and the time to create the grid is very short. It is therefore also a very useful tool to use in conjunction with INDES. At each iteration, the geometry will change, but since the grids are not body-fitted, the same grid can be used at each iteration. A typical grid can be seen in Fig. 2. Embedded meshes of higher grid point density are used to resolve areas of high curvature - eg. the leading edges. Computations of complete aircraft configurations can be performed overnight on an SGI ORIGIN 200 workstation.

The 2-D flow solver used in this work was TORNADO[12] which is a multi-block Navier Stokes code that was developed under a collaboration between Bombardier Aerospace and the University of Toronto Institute for Aerospace Studies (UTIAS). TORNADO was developed from ARC2D, and thus uses the Beam and Warming implicit approximate-factorization algorithm.

The current version of TORNADO has three separate turbulence models: Baldwin-Barth, Spalart-Allmaras and the Mentor SST model. It also includes matrix dissipation, which leads to more accurate drag prediction, and multi-grid convergence acceleration, which can lead to speed-ups in convergence by a factor of 3. However, for the purposes of this investigation, the multi-grid option was not used. The primary reason is that the multi-grid option is more sensitive to grid quality, and since the grids were created totally automatically, it was decided that it would be prudent not to use this acceleration technique at this stage.

A suite of codes called AMBER2D was also developed by UTIAS to accompany TORNADO. The AMBER2D codes are used to automatically generate 2-D multi-block grids for TORNADO, and the suite consists of three programs, "kmS" which creates the grid topology based on a panel solution, "agrid" which creates an initial algebraic grid, and "megrid" which is used to smooth the algebraic

grid. During the inverse design, a new multi-block grid is produced at each design iteration.

3 Discussion of Results

3.1 2-D Flap Design

This test case used the NLR 7301 takeoff wing/flap configuration[13]. This configuration is illustrated in Fig. 3 below. This configuration consists of a wing with flap at 13.9 degrees deflection angle. The flap is modeled with a small blunt base of thickness 0.17% local flap chord. The objective of the test case is to “re-create” the NLR 7301 geometry from the target pressures. The target pressures are generated by running TORNADO on the NLR 7301 geometry. An H-H topology grid is used to model the configuration and this grid is created automatically using the AMBER2D suite of codes. The grid contains 14 blocks with a total of 64000 grid points. The grid spacing normal to the surface uses Navier Stokes spacing of $1.0e-06c$. The TORNADO solution uses the

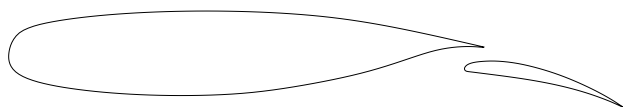


Fig. 3. NLR 7301 Takeoff Wing/Flap

Spalart-Allmaras turbulence model, with scalar dissipation. The TORNADO solution required 7500 iterations to converge, and took 8800 CPU seconds.

Flow conditions for this model were Mach 0.2, incidence 4 degrees, and Reynolds number 2.51 million. Transition was set on the main element at 3% upper, and 78% lower surface, and on the flap at 58.8% upper and 95% lower surface, according to the experimental report[13].

The original target flap geometry (NLR 7301)

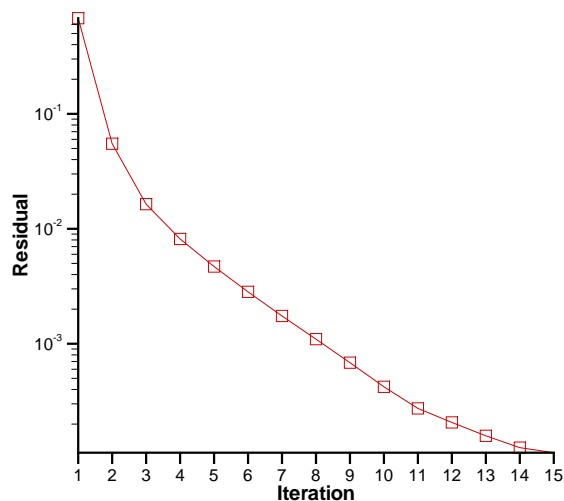


Fig. 4. Convergence History of 2-D NLR Flap Design

was replaced by a NACA0012 section with the same trailing edge thickness, and orientation, and this was used as the starting design. At each iteration, the basic grid topology does not change, and thus each new TORNADO solution can be restarted from the solution for the previous geometry, thus reducing the runtimes.

Initial tests with this model revealed that INDES initially had a tendency to modify the geometry too much in the vicinity of the leading edge, thus producing a rather pointed shape, which, not surprisingly, gave convergence problems for TORNADO, and even occasionally caused problems with the gridding program. It was therefore necessary to apply a damping factor to the thickness change calculated within INDES. This took the form of a maximum thickness change per design iteration of 0.001c at the leading and trailing edges, and 0.005c in the vicinity of the mid-chord.

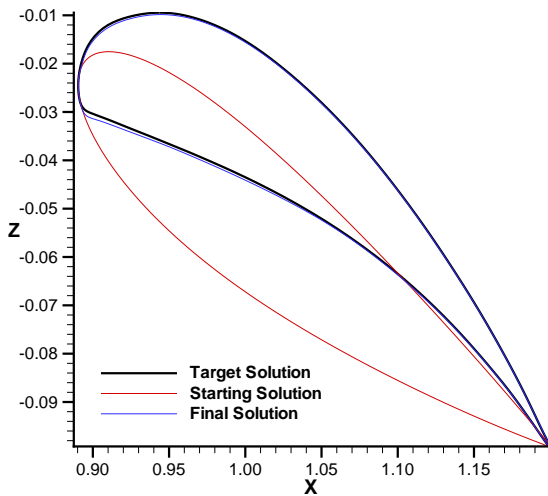


Fig. 5. Comparison of Geometry for 2-D NLR Flap

The inverse design convergence history is shown in Fig. 4 where the residual is defined by:

$$Residual = \left[\sum_{i=1}^N \int_{x_{le}}^{x_{te}} (Cp_{calc} - Cp_{target})_i^2 \right] / N$$

where N is the number of design stations, $(Cp_{calc} - Cp_{target})$ is the difference between the calculated and target pressures, and x_{le} , x_{te} are the leading edge and trailing edges respectively.

The inverse design was run for 15 iterations. For the purposes of this evaluation exercise, no convergence checking is performed. Thus, although the inverse design ran for 15 iterations, the results were satisfactory after about 12 iterations. Of course, the thickness damping applied within INDES has a negative impact on the speed of convergence.

Fig. 5 illustrates the final design geometry compared to the target shape. As can be seen, the target geometry has been well reproduced starting with the NACA0012 geometry. Fig. 6 shows the comparison between the starting pressure, the final pressure after 15 design iterations, and the target pressures. The

agreement is very good, although some small differences can be seen at the leading edge.

All the inverse design runs were done on a SGI ORIGIN 200 workstation 270MHz processor. The first run with the initial design took about 6000 iterations to converge sufficiently, and took about 7400 CPU seconds. Subsequent TORNADO runs used the solution from the previous design iteration as a restart, and took about 5000 iterations to converge in 6000 CPU seconds.

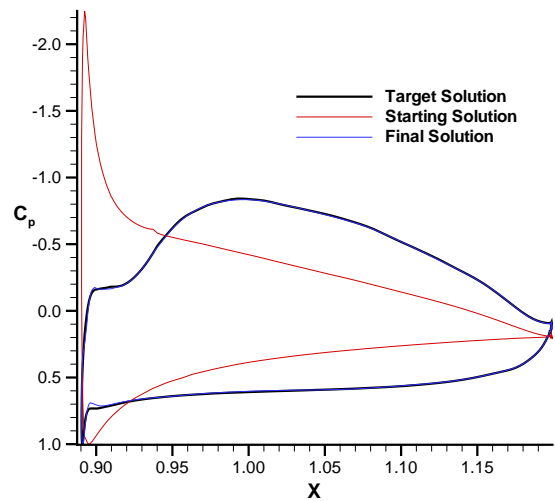


Fig. 6. Comparison of Pressures for 2-D NLR Flap

3.2 3-D Flap Design

The geometry used for the 3-D flap design investigation consisted of a tapered wing and flap with the NLR 7301 wing and flap used for the target sectional geometry.

Again, the objective is to “re-create” the tapered wing composed of NLR 7301 sections from the target pressures. The target pressures are created by running the flow solver, in this case MGAERO, on this target geometry and extracting the pressures at 9 linearly-distributed spanwise stations. The flow conditions again were Mach 0.2 and incidence 4.0 degrees. The 3-D grid consisted of 8 levels, and the

MGAERO pre-processor and solver together took 15000 CPU seconds on the SGI ORIGIN 200 workstation.

The starting geometry for the inverse design was created by replacing the target flap sectional geometry with a NACA0012 section. Each section was suitably scaled, such that the trailing edge thickness and chord matched the target.

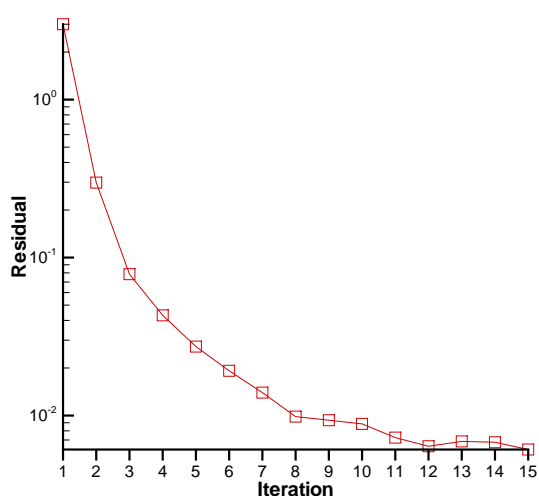


Fig. 7. Convergence History for 3-D Tapered Wing.

The inverse design was again run for 15 iterations. The damping function on the thickness perturbation calculated within INDES, was also used, but it was discovered that the use of this function is not as important as with the 2-D flow solver. It seems that MGAERO is more forgiving of the relatively large changes to the geometric shapes produced within INDES during the early stages of the inverse design.

The inverse design convergence was not quite as good as in the 2-D case as can be seen in Fig. 7. Additionally, although the target pressures were quite well replicated as can be seen in Fig. 8, there were some discrepancies at the leading edge, but overall the target geometry was well reproduced as can be seen in Fig. 9.

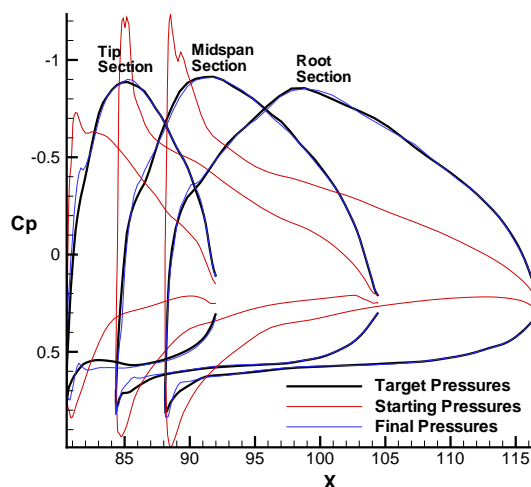


Fig. 8. Comparison of Pressures at Root and Midspan of Tapered Wing.

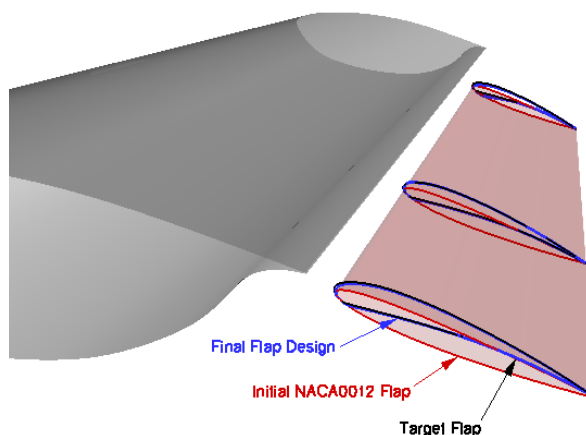


Fig. 9. Comparison of Geometry for 3-D Tapered Flap

4 Concluding Remarks

The inverse design method described in this paper has been used to design a flap, both in 2-D and 3-D, with respectively a Navier-Stokes code and a 3-D Euler code. The method

successfully replicated the target geometry within 15 design iterations.

In this investigation, the entire flap has been designed starting from a NACA0012 section. In practice, only the region around the leading edge of the flap would be eligible for design. Most of the flap surface is determined from the cruise wing design. Thus the practical use of this method would require the addition of a partial chord design. Some work has already been done at Bombardier Aerospace into extending the method to include a partial chord design option, but the development is still not complete.

The design of other components in the high lift system also remains to be investigated, in particular the wing under-slat surface (WUSS) design will be investigated further.

References

- [1] Besnard, E., Schmitz, A., Boscher, E., Garcia, N., and Cebeci, T., Two-dimensional aircraft high-lift systems design and optimisation, *36th AIAA Aerospace Sciences Meeting and Exhibit*, Reno, AIAA Paper 98-0123, 1998.
- [2] Milholen, W.E., II, Efficient inverse aerodynamic design method for subsonic flows, *38th AIAA Aerospace Sciences Meeting and Exhibit*, Reno, AIAA Paper 2000-0780, 2000.
- [3] Fejtek, I., Jones, D., Waller, G., Hansen, E., and Obayashi, S., A transonic wing inverse design capability for complete aircraft configurations, *19th AIAA Applied Aerodynamics Conference*, AIAA Paper 2001-2443, 2001.
- [4] Wilhelm, R., An inverse design method for designing isolated and wing-mounted engine nacelles, *40th AIAA Aerospace Sciences Meeting and Exhibit*, Reno, AIAA Paper 2002-0104, 2002.
- [5] Takanashi, S., Iterative three-dimensional transonic wing design using integral equations, *AIAA Journal of Aircraft*, Vol. 22, No. 8, August 1985.
- [6] Farin, G., *Curves and surfaces for computer-aided geometric design*, Academic Press, Toronto, 1988.
- [7] Bartelheimer, W., An improved integral equation method for the design of transonic airfoils and wings, AIAA paper 95-1688-CP, 1995.
- [8] Jones, D., Fejtek, I., and Leblond, D., Coupling of a wing inverse design code to a transonic small disturbance flow solver, *Proceedings of the 48th Annual CASI Conference*, Toronto, pp. 237-242, April 29 - May 2, 2001.
- [9] Fejtek, I., and Jones D., Development of a wing inverse design capability for complete aircraft configurations, *Proceedings of the 7th Annual Conference CFD99*, Halifax, May 30 - June 1, 1999.
- [10] Epstein, N., Luntz, A., and Nachshon, A., Multigrid Euler solver about arbitrary aircraft configurations, with cartesian grids and local refinements, AIAA Paper 89-1960, June 1989.
- [11] Waller, G., and Hansen, E., Drag prediction of engine installation effects on a twin-engine transport model, *Proceedings of the 48th Annual CASI Conference*, Toronto, pp. 25-36, April 29 - May 2, 2001.
- [12] Nelson, T.E., Zingg, D.W., and Johnston, G.W., Compressible Navier Stokes Computations of multielement airfoil flows using multiblock grids, *AIAA Journal*, Vol. 32, No. 3, 1994.
- [13] Van Den Berg, B., Boundary layer measurements on a two-dimensional wing with flap, NLR TR 79009 U, January 1979.