AN INVERSE METHOD FOR THE DESIGN OF AIRFOILS

By Tsuneari Nagaoka and Etsuo Morishita
Department of Aeronautics and Astronautics, School of Engineering, the University of Tokyo

Abstract
An inverse method for designing airfoils, in which final profiles are determined from prescribed pressure distributions, is researched. First, an inverse method using the panel method is briefly introduced. Then, a method using a non-linear equation as the governing equation is proposed. Due to the treatment of the boundary condition on the airfoil surface, modification of the airfoil geometry is complicated, so an iterative direct-inverse approach is applied. Residual of the calculated pressure distribution by the direct method solving compressible Navier-Stokes equation and the prescribed pressure distribution will be minimized by a numerical optimization method. The procedure is repeated until good convergence is attained in the final profile. A brief result of a non-lifting case is shown.

1. Introduction
Due to the cost of wind tunnel testing and improvement in the performance of computers, designing components of airplanes using computers is playing an important part in the design stage today. Direct method which analyze the flow around an airfoil whose shape is prescribed is used as a tool for designing airfoils in the industrial stage. Inverse method, which determines the airfoil shape satisfying the prescribed flow condition, however, is still in the experimental stage.

Although the inverse method has a possibility of not having a solution to the prescribed condition, it also has a possibility to obtain a breakthrough to the performance which may be difficult to attain using the direct method.

Inverse method can be divided into two categories: analytical method and computational method. The analytical method which transforms an airfoil into a circle whose solution is known using conformal mapping was used by Lighthill(1) and Sato(2). This method is limited to only solving a two dimensional problem.

One of the most simple ways to solve an inverse problem computationally is to assume the flow as a potential one, and use the Laplace equation as the governing equation. For example, the method by Bristow and Grose(3) can cover a wide range of inverse problems, including designing a portion of an airfoil, with the remaining portion already designed.

2. Inverse Panel Method by Shigemi
The panel method is one of the most widely used tools to
solve a potential flow problem. Although the flow solved by the panel method is inviscid and incompressible, it approximates the flow around an airfoil very well in case the flow is in a low Mach number region and has no large separations. The panel method, which was originally developed to solve the direct problem, can also be applied to the inverse problem.

We briefly introduce the panel method which was modified by Shigemi\(^{4,5}\). First, we give the principle of the panel method as a solution to the direct problem, then we apply it to the inverse problem.

An airfoil is discretized to \(m\)-1 nodes, starting counter clockwise from the trailing edge, so the trailing edge point is counted again as the \(m\)-th node. The density of the distributed vortex is \(\gamma_i\) at the \(i\)-th node, and varies linearly from \(\gamma_i\) to \(\gamma_{i+1}\) along the \(i\)-th panel.

At the \(i\)-th control point \((x_i, y_i)\) of the \(i\)-th panel (typically the midpoint), the velocity component normal to the surface \(v_{i,n}\) is given as follows:

\[
v_{i,n} = -U_{\infty} \frac{X_i \sin \alpha - Y_i \cos \alpha}{l_i} - \frac{1}{2\pi l_i} \sum_{j=1}^{m-1} \frac{A_{ij} \gamma_j + B_{ij} \gamma_{j+1}}{l_j} (i = 1, 2, \ldots, m-1)
\]

where

\[
A_i = I_1 Y_i + I_3 X_i
\]

\[
B_i = I_2 Y_i + I_4 X_i
\]

\[
X_i = x_{i+1} - x_i
\]

\[
Y_i = y_{i+1} - y_i
\]

\[
l_i = \sqrt{X_i^2 + Y_i^2}
\]

and \(l_i\) are coefficients which are functions of \(x_i\) and \(y_i\), \(U_{\infty}\) and \(\alpha\) are the free stream velocity and angle of attack, respectively. Boundary condition of zero normal velocity at the body surface is given by setting the left hand side of Eq. (1) to zero, which will be rewritten as follows:

\[
\sum_{j=1}^{m-1} \frac{A_{ij} \gamma_j + B_{ij} \gamma_{j+1}}{l_j} = 2\pi U_{\infty} (Y_i \cos \alpha - X_i \sin \alpha) (i = 1, 2, \ldots, m-1)
\]

The Kutta condition at the trailing edge will be given in the form of

\[
\gamma_1 + \gamma_m = 0
\]

\(\gamma_i\) can be obtained by solving the system of Eqs. (7) and (8).

An equivalency between the tangential velocity \(v_{i,t}\) and the vortex density is held on the surface of the airfoil can be proven.

\[
\nu_{i,t} = \gamma_i
\]

The principle of solving the inverse problem using the panel method is to specify a set of coordinates \((x_i, y_i)\) of nodes of the polygon which satisfies Eq. (7), where \(\gamma_i\) is already given. If the prescribed condition is given as velocity distribution, it can be converted into vortex distribution by Eq. (9), and if the prescribed condition is given as pressure distribution, it can be converted into velocity distribution by the following equation:

\[
C_p = \frac{p - p_{\infty}}{\frac{1}{2\rho U_{\infty}^2}} = 1 - \frac{u^2 + v^2}{U_{\infty}^2}
\]

It is sufficient to set only \(\gamma_i\) (or \(\gamma_{i,t}\)) as unknowns. Although Eq. (7) is a linear equation of \(\gamma_i\), it becomes nonlinear with regard to \(\gamma_i\), so the solution is attained only iteratively, for example, by using the Newton-Raphson method. Eq. (7) can be rewritten as follows:

\[
\nu_{i,t} = \gamma_i
\]
where
\[ a_i = (a_i, a_i, ..., a_i) \]
\[ \gamma = (\gamma_1, \gamma_2, ..., \gamma_m) \]
\[ b_i = 2\pi U_{\infty} (Y_i \cos \alpha - X_i \sin \alpha) \]
and \( a_i, b_i \) are function of \( Y = (Y_1, Y_2, ..., Y_m) \).

The initial guess \((Y_1^0, Y_2^0, ..., Y_m^0)\) to the solution of Eq. (11) can be expanded into a Taylor series, neglecting higher orders.

\[ f_i(Y) = \sum_{k=1}^{m} \left( \sum_{j=1}^{m} \frac{d a_{ij}}{d Y_j} Y_j + \frac{d b_i}{d Y_k} \right) \delta Y_k \]

(12)

When \( \delta Y \) is obtained as the solution of Eq. (12), \( Y_i^0 \) improves to \( Y_i^0 + \delta Y_i^e \), and the same procedure is repeated until the solution converges.

Eq. (12) represents \( m-1 \) equations, while the number of unknowns is \( m \). To order to close the system of equations, one of the values of \( \delta Y_i^0 \) must be fixed. It is natural to fix the leading edge point, and so the shape of the whole airfoil is improved by moving the relative position of each node with respect to the leading edge point.

The flow chart of this procedure is given in Fig. 1, and a result using this procedure is shown in Fig. 2. The number at the left side indicates the number of iteration and 0 corresponds to the initial profile. In this case, the final profile is attained after 5 iterations. Target pressure distribution is NACA 4415, which is represented with a solid line, obtained from experimental data, with the Reynolds number of \( 1 \times 10^6 \) and \( \alpha = 0^\circ \). The difference at the upper surface near the trailing edge is thought to be the effect of the boundary layer.

The procedure for obtaining the solution using Eq. (12) is shown in Fig. 2. The solution of Eq. (12) is the initial guess \((Y_1^0, Y_2^0, ..., Y_m^0)\) to the solution of Eq. (11), which is the velocity distribution uncorrected for the boundary layer. The solution of Eq. (11) is obtained using Eq. (12) as the initial guess until the solution converges.

When \( \delta Y \) is obtained as the solution of Eq. (12), \( Y_i^0 \) improves to \( Y_i^0 + \delta Y_i^e \), and the same procedure is repeated until the solution converges.

3.3 Inverse Method Using Navier-Stokes Equation

Using the panel method, the final profile can be obtained easily; however, due to the governing equation, the application is restricted to a low Mach number region and has no large separations. In order to solve transonic flow problems, using Navier-Stokes equations, the governing equation would be a natural approach. Due to the treatment of the boundary condition on the airfoil surface, modification of the airfoil geometry is complicated, so what is called an iterative direct-inverse approach will be applied.

3.1 Grid Generation

Computational grid system is generated using Poisson equation.

\[ \xi_x + \xi_y = P(\xi, \eta), \quad \eta_x + \eta_y = Q(\xi, \eta) \]

(13)

The actual solution of Eq. (13) is carried out in the computational \((\xi, \eta)\) domain. In this domain, Eq. (13) will be transformed to

\[ \alpha x_{\xi_x} + 2\beta x_{\xi_y} + \gamma x_{\eta_y} = -J^{-1} (P\xi_x + Q x\eta) \]

\[ \alpha y_{\xi_x} + 2\beta y_{\xi_y} + \gamma y_{\eta_y} = -J^{-1} (P\eta_x + Q y\eta) \]

(14)

where

\[ J^{-1} = x_{\xi_x} y_{\eta_y} - y_{\xi_y} x_{\eta} \]

\[ \alpha = x_{\xi_x}^2 + y_{\eta}^2, \quad \beta = x_{\xi_x} y_{\eta_y} + y_{\xi_y} x_{\eta}, \quad \gamma = x_{\xi_x}^2 + y_{\xi_y}^2 \]

and \( P, Q \) are functions used to control interior grid clustering.

\[ P(\xi, \eta) = p_1(\xi)e^{-\eta^2} + p_2(\xi)e^{-\eta (\eta_{max} - \eta)} \]

\[ Q(\xi, \eta) = q_1(\xi)e^{-\eta^2} + q_2(\xi)e^{-\eta (\eta_{max} - \eta)} \]

(15)

where \( a, b, c, d, e, f, g, h, i, j, k, l, m, n, o, p, q, r, s, t, u, v, w, x, y, z \) are constants and \( p, q, r, s, t, u, v, w, x, y, z \) are functions of \( \xi \).

The type of grid which is said to be most suitable to calculate the flow around an airfoil is what is called a
C-type. In this type of grid, treatment in the vicinity of trailing edge is most important, however, it is quite difficult to maintain the orthogonality.

Orthogonality in the region of the trailing edge can be obtained using the method proposed by Catherall which adds the following doublet term to Eq. (15).

\[ P_D = \alpha P e^{-r n^r}, \quad Q_D = \alpha Q e^{-r n^r} \quad (16) \]

where

\[ r = \sqrt{(\xi - \xi_{TR})^2 + (\eta - \eta_{TR})^2} \]

\( \alpha \) and \( \alpha \) are parameters, \((\xi_{TR}, \eta_{TR})\) are the coordinates of the trailing edge in the computational domain, and \( \gamma_0 \) is used to restrict the extent of the region surrounding the trailing edge which is affected. The results are shown in Fig. 3(a) and (b).

3.2 Governing Equations

The governing equation is compressible Navier-Stokes equation (3). Two dimensional Navier-Stokes equation can be written in generalized coordinates as follows:

\[ \frac{\partial \hat{Q}}{\partial t} + \frac{\partial (\hat{E} - \hat{E}_x)}{\partial \xi} + \frac{\partial (\hat{F} - \hat{F}_x)}{\partial \eta} = 0 \quad (17) \]

where

\[ \hat{Q} = \frac{1}{J} \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ e \end{bmatrix}, \quad \hat{E} = \frac{1}{J} \begin{bmatrix} \rho U \\ \rho u U + \xi \rho \xi \\ \rho u v + \xi \rho \eta \\ \rho v U + \eta \rho \eta \end{bmatrix}, \quad \hat{E}_x = \frac{1}{J} \begin{bmatrix} 0 \\ \xi \xi \tau_{xx} + \xi \tau_{xy} \\ \xi \xi \tau_{xx} + \xi \tau_{xy} \\ \xi \xi \beta_x \end{bmatrix} \]

\[ \hat{F} = \frac{1}{J} \begin{bmatrix} \rho V \\ \rho v U + \eta \rho \xi \\ \rho v v + \eta \rho \eta \end{bmatrix}, \quad \hat{F}_x = \frac{1}{J} \begin{bmatrix} 0 \\ \eta \eta \tau_{xx} + \eta \tau_{xy} \\ \eta \eta \tau_{xx} + \eta \tau_{xy} + \eta \beta_x \end{bmatrix} \]

\[ \frac{1}{J} = x_{\xi} \eta - x_{\eta} \xi \]

\[ U = \xi u + \xi v, \quad V = \eta u + \eta v \]

\[ \tau_{xx} = \frac{2}{3} \mu (2u_x - v_x), \quad \tau_{xy} = \frac{2}{3} \mu (2u_v - v_x) \]

\[ \tau_{yx} = \tau_{xy} = \mu (u_x + v_x) \]

\[ \beta_x = \tau_{xx} u + \tau_{xy} v + \kappa T_x, \quad \beta_y = \tau_{yx} u + \tau_{yy} v + \kappa T_y \]

3.3 Updating Algorithm

The designing procedure is as follows. First, we assume the target pressure distribution and initial geometry. Then the flow around initial geometry is calculated using the direct method. The updating of geometry is done by changing the surface curvature, \( \Delta C \), which is related to the difference between target and calculated pressure distribution, \( \Delta C_r \), using the algorithm developed by Campbell.

For subsonic and low-supersonic Mach numbers (\( M \leq 1.1 \)), the following equation is used.

\[ \Delta C = \Delta C_r \cdot A(1 + C^2)^B \quad (18) \]

where \( A \) is a relaxation factor that is positive for the upper surface and negative for the lower surface. The exponent \( B \) may vary between 0 and 0.5, with higher values yielding a faster convergence rate but less stability in the leading edge region.

When the local Mach number is above 1.1, the equation initially used is

\[ \Delta C = \frac{d (\Delta C_r)}{d x} A \sqrt{M_x^2 - 1} \left[ 1 + \left( \frac{d y}{d x} \right)^2 \right]^{1.5} \quad (19) \]

\[ \frac{d (\Delta C_r)}{d x} = \frac{1}{2} \frac{d M_x}{d x} \]
When the streamwise slopes of the calculated pressures are close to the corresponding slopes of the target distribution, Eq. (18) is used in combination with Eq. (19) to obtain faster convergence. Since Eq. (19) is not technically valid when the free stream Mach number $M_\infty$ less than 1.0, an effective free stream Mach number of 1.01 is applied for the subsonic cases.

These equations are applied at each point along the airfoil surfaces, marching from the leading edge to the trailing edge. The local curvature changes are made by shearing the points aft of the current through a given angle (see Fig. 4(a)). This approach results in minimal changes to the curvatures at the other points, however, at the end of the design sweep, the airfoil will typically have either an open or crossed trailing edge. To remedy this situation, the surface is rotated about the leading edge back to the original trailing edge location (see Fig. 4(b)). Smoothing is applied to both the airfoil surface and the nose camber line to ensure that a reasonable airfoil geometry is maintained throughout the design process.

Once a new surface is obtained, a new grid system is developed and the same procedure is continued until convergence is obtained. The flow chart of this procedure is given in Fig. 5.

3.4 Computational Result

Fig. 6 shows the result obtained by the procedure in the former section. At this level of development, a subsonic and non-lifting case is the case that can be calculated. The present iterative procedures converge well in this numerical experiment.

4. Conclusions

An inverse problem of obtaining an airfoil which satisfies a given pressure distribution is researched. The present research can be summarized as follows:

1. Using the inverse panel method, the final profile can be obtained easily, however, the application is restricted to low Mach number region and has no large separations.

2. A non-lifting subsonic airfoil inverse design is studied. An inverse method is introduced, and the present method is found suitable to be employed in this case.

The Navier-Stokes equations have a possibility of solving the inverse problems where shocks and boundary layer separation can be seen. The future work is to apply the present procedure to solving the high Mach number problems, as well as high angle of attack problems. Shortening of calculation time is also expected.

References


