A98-31508

ICAS-98-2,5,2

UNSTEADY PARALLEL AIRFOIL DESIGN FOR ROTARY WING APPLICATIONS

Luis Carlos de Castro Santos
Departamento de Matemática Aplicada - Instituto de Matemática e Estatística
Universidade de São Paulo, Rua do Matão 1010, São Paulo, SP, 05508-900, BRAZIL

Abstract

The fast changes in aerospace industry require a more accurate and flexible design structure, taking into account limitations on resources. In the under developed countries several industries failed in completing projects not only due to the lack of resources, but mostly due to the bad management of the available resources. An important issue in helicopter preliminary design is the choice of the blade geometry. The experimental data available in the literature severely limits the choices of design, and wind tunnels are not an option in a short budget context. Therefore a methodology is presented, by coupling a Navier-Stokes 2D code, taking into account compressible and viscous effects, to classical blade element and momentum theory in order to provide a more flexible preliminary design tool, while minimizing the computational requirements in terms of hardware (which is more realistic for under developed countries). A design test case is performed confirming the validity of the technique.

Introduction

The aerodynamic environment of rotary wings is rather complex entailing to a very demanding design problem. The aerospace market increasingly demands that the development costs must be kept to a minimum, and in order to do so, as much of the flow complexities must be taken into account in the preliminary design phase. Therefore it is important to start with a good choice of airfoil in order to minimize the redesign of the rotor in the development phase. To accomplish that, a particular combination of Computational Fluid Dynamics (CFD) and Design Optimization methods is proposed.

Since the main objective is to detect and minimize problems in the preliminary design phase, the turn-around solution time must be minimal. Therefore an unsteady Navier-Stokes code is used to simulate the aerodynamic environment faced by a rotor blade profile. The numerical airfoil data, is combined with blade element theory in order to estimate the full rotor aerodynamic forces and moments. This approach is validated by comparison with literature results and is proven accurate enough for preliminary rotor design purposes.

The airfoil shape can be modeled by a series of parameters and the optimization method can be used to minimize the required power on certain pre-determined flight conditions. In any optimization procedure the bottleneck is the evaluation of the sensitivities, which usually requires N number of flow solver runs, where N is the number of design variables. Although effective this technique leads to high computational costs.

The introduction of computational fluid dynamics (CFD) tools in engineering design has been restricted to the highly developed companies and businesses due to the necessity of an adequate balance between hardware, software and support. For the less developed, CFD is still limited to some critical calculations and is far from being an everyday tool. Even though, preliminary design problems can be more efficiently computed with the addition of CFD tools in the design cycle without the necessity of great changes in the design methodology, which is in general more guided by management style than technical aspects. The coupling of classical and CFD techniques can add flexibility and turn-around time without major investment in computational resources. A method is proposed, taking into account severe restrictions in computer hardware and without major allocation of resources to algorithm development (typical of large CFD codes).

To reduce the optimization cycle time, with a penalty on accuracy, the design problem can be represented as an interpolation problem in several dimensions. The designer has the choice of using a classical optimization technique, once the design space is mapped or, using his experience and based on the data, select another portion of the design space to analyze.

This technique can be easily implemented in parallel allowing the whole procedure to be accomplished in a matter of hours, which is compatible with today's design demands.

Problem Statement

A typical practice in rotor performance estimation is to combine Blade Element and Momentum Theory in order to compute aerodynamic coefficients [6-7]. Considering a helicopter rotor with a number of blades equal to b the thrust and power coefficients (based on rotor disk area) are given by:

$$C_T = \sigma/2 \int a\, [\, \theta\, \underline{r}^2 - \lambda\, \underline{r}\,]\, d\underline{r} \qquad (1)$$

$$C_P = \sigma/2 \int a\, [\lambda\theta\, \underline{r}^2 - \lambda^2\, \underline{r}\,]\, d\underline{r} + \sigma/2 \int C_{d0}\, \underline{r}^3 d\underline{r} \qquad (2)$$

where a is the profile lift slope, σ is the rotor solidity , θ the collective pitch angle, λ the inflow ratio and C_{d0} is profile base drag. The integration is made from $\underline{r}_{min}$ to $\underline{r}_{max}$ representing the effective aerodynamic regions of the blade. The position along the span $\underline{r}$ = r/R, is non-dimensional, since R is the rotor radius. The inflow is given by:

$$\lambda = -\kappa + (\kappa^2 + 2\kappa\theta\underline{r})^{1/2} \qquad (3)$$

where $\kappa = a\sigma/16$.

The design of the blade profile determines its lift slope and base drag coefficient. High Performance techniques [4] although very representative require a considerable amount of computational resources. An alternative is to represent the blade by a series of section, in the strip theory sense, but computing the aerodynamic coefficients with respect to their dependency on local Mach and Reynolds numbers. In this manner the prediction can take into account the compressible and viscous effects, namely shock wave and separation, associated to the rotor's operation. This technique has been compared with a full simulation[4] and has proven to be accurate in order to obtain a preliminary design estimate.

The Navier-Stokes Flow Solver

The flow phenomena of interest is modeled by the Reynolds-averaged Navier-Stokes equations, here presented in a two-dimensional body-fitted coordinate system:

$$\partial_t q + \partial_\xi E + \partial_\eta F = 1/Re\, (\partial_\xi R + \partial_\eta S) \qquad (4)$$

where q is vector of conserved quantities: mass, momentum in x and y directions and energy.

$$q = 1/J\, \{\, \rho\, ,\, \rho u\, ,\, \rho v\, ,\, \rho e\, \}^T \qquad (5)$$

The energy is given by:

$$e = \rho\, [\, C_v\, T + (u^2 + v^2)/2\,] \qquad (6)$$

where T is the temperature and C_v the specific heat at constant volume.

The vectors E and F are the inviscid fluxes of the conserved quantities in ξ and η directions:

$$E = 1/J\, \{\, \rho U,\ \rho u U + p\xi_x\, ,\ \rho v U + p\xi_y\, ,\ U(e+p)\, \}^T$$

$$F = 1/J\, \{\, \rho V,\ \rho u V + p\eta_x\, ,\ \rho v V + p\eta_y\, ,\ V(e+p)\, \}^T \qquad (7)$$

where U and V are the contravarient components of velocity:

$$U = u\, \xi_x + v\, \xi_y \quad ,\quad V = u\, \eta_x + v\, \eta_y \qquad (8)$$

The vectors R and S are the viscous fluxes, respectively in ξ and η directions:

$$R = 1/J\, \{0,\ \tau_{x\xi}\, ,\ \tau_{y\xi}\ 0,\ \Phi_R\, \xi_x + \Phi_S\, \xi_y\, \}^T$$

$$S = 1/J\, \{0,\ \tau_{x\eta}\, ,\ \tau_{y\eta}\, ,\ \Phi_R\, \eta_x + \Phi_S\, \eta_y\, \}^T \qquad (9)$$

and the stresses:

$$\tau_{x\xi} = \tau_{xx}\, \xi_x + \tau_{xy}\, \xi_y$$
$$\tau_{y\xi} = \tau_{xy}\, \xi_x + \tau_{yy}\, \xi_y$$
$$\tau_{x\eta} = \tau_{xx}\, \eta_x + \tau_{xy}\, \eta_y$$
$$\tau_{y\eta} = \tau_{xy}\, \eta_x + \tau_{yy}\, \eta_y$$

with the fluid being considered as Newtonian and Φ_R and Φ_S given as viscous dissipation terms[7].

The equations are numerically approximated by central differences resulting in:

$$(I+\Delta t\, J\delta_\xi A^n + \varepsilon_I D_{I_\xi})\,(I+\Delta t\, J\delta_\eta B^n + \varepsilon_I D_{I_\eta})\{\Delta q\} = \{C^n - \varepsilon_E D_E\} \qquad (10)$$

where A = $\partial E/\partial q$ and B= $\partial F/\partial q$ are the Jacobian matrices and the residual:

$$C^n = -\Delta t\, J\,(\delta_\xi E+\delta_\eta F) + \Delta t\, J/Re(\delta_\xi R+\delta_\eta S) \qquad (11)$$

The terms D_I and D_E are artificial dissipation terms[8] required to stabilize the numerical scheme. In addition to that, the algebraic Baldwin-Lomax model is used, to take into account turbulence effects. The problem is decoupled in two penta-diagonal problems and the computational code developed by Sankar Huff and Wu[7]. is used to obtain the solution. The domain is discretized as shows Figure 1.

Geometrical Description

A family of symmetrical airfoils can be defined using the relationship:

$$y(x) = t\,(a_0 \sqrt{x} + a_1 x + a_2 x^2 + a_3 x^3) \qquad (4)$$

where t is the profile thickness relative to the chord, and x is the percentage of chord. Therefore y is also normalized. The coefficients are not very intuitive therefore it is better to describe the airfoil using more practical parameters. If we defined t as the thickness (relative to the chord), x_m as the percentage of chord corresponding to the maximum thickness and τ as the profile trailing edge angle, we can relate these parameters to the coefficients by the system:

$$[A]\,\{a_0,\ a_1,\ a_2,\ a_3\}^T = \{\tfrac{1}{2},\ 0,\ 0,\ -\tan(\tau/2)\}^T$$

where $[A]$ is given by:

$$\begin{vmatrix} \sqrt{x_m} & x_m & x_m^{\,2} & x_m^{\,3} \\ t & t & t & t \\ t/2\sqrt{x_m} & t & 2t\,x_m & 3t\,x_m^{\,2} \\ t/2 & t & 2t & 3t \end{vmatrix}$$

This system is obtained by applying conditions on the trailing closure and angle, slope and position of the maximum thickness. This family of profiles is therefore defined by only three parameters which span the design optimization space. An example can found on Figure 2.

The lift slope and profile drag must be computed for the operation conditions as a function these design parameters.

A Typical Design Problem

The main design problem of aerodynamic is to find an adequate geometry to achieve a prescribed performance. Consider a helicopter rotor in hover, in a certain operating condition. Using the family of airfoils described in the previous section one could which to estimate the level of thrust generated and power required by the rotor for different choices of the design parameters, which in this case are the airfoil´s thickness and position of maximum thickness. The designer should provide an initial guess of the design parameters and run the Navier-Stokes code for a certain region of the design space. In this case t was varied from 10% to 12%, x_m from 15% to 25% and the Mach from 0.5 to 0.7. The trailing edge angle is kept fixed to 5°.

The results where obtained in parallel using an IBM SP1 machine with 14 nodes composed of RS6000 model IBM370 processors. The peak performance of those processors is on the 36 MFLOPS range, which is quite low compared with the international supercomputing standard. The parallel capability allows a representative improvement in performance.

A 157 x 40 grid was used and all computations where converged until the residual (eq. 11) reached 10^{-7}. The nine cases considered where obtained after approximately 35 minutes of CPU time, which in terms of turnaround time is quite reasonable. The results for the lift slope and profile drag are displayed on Tables 1 to 6.

	$x_m = 15\%$	$x_m = 20\%$	$x_m = 25\%$
t = 10%	7.529	7.540	7.540
t = 11%	7.609	7.632	7.597
t = 12 %	7.666	7.689	7.643

Table 1 : Lift Slope coefficient (a) for M = 0.5

	$x_m = 15\%$	$x_m = 20\%$	$x_m = 25\%$
t = 10%	8.285	8.319	8.285
t = 11%	8.388	8.411	8.377
t = 12 %	8.491	8.468	8.411

Table 2 : Lift Slope coefficient (a) for M = 0.6

	$x_m = 15\%$	$x_m = 20\%$	$x_m = 25\%$
t = 10%	9.717	9.694	9.626
t = 11%	9.843	9.878	9.752
t = 12 %	9.855	9.981	9.855

Table 3 : Lift Slope coefficient (a) for M = 0.7

	$x_m = 15\%$	$x_m = 20\%$	$x_m = 25\%$
t = 10%	0.0090	0.0080	0.0074
t = 11%	0.0096	0.0085	0.0077
t = 12 %	0.0103	0.0091	0.0082

Table 4 : Profile Drag coefficient (C_{d0}) for M = 0.5

	$x_m = 15\%$	$x_m = 20\%$	$x_m = 25\%$
t = 10%	0.0101	0.0088	0.0080
t = 11%	0.0109	0.0095	0.0085
t = 12 %	0.0119	0.0101	0.0090

Table 5 : Profile Drag coefficient (C_{d0}) for M = 0.6

	$x_m = 15\%$	$x_m = 20\%$	$x_m = 25\%$
t = 10%	0.0132	0.0106	0.0093
t = 11%	0.0153	0.0116	0.0101
t = 12 %	0.0179	0.0128	0.0109

Table 6 : Profile Drag coefficient (C_{d0}) for M = 0.7

Second degree interpolating polynomials were used for each pair of thickness - position of maximum thickness of the table above:

$$a = a_0 + a_1 k + a_2 k(k-1)/2$$

$$C_{d0} = c_0 + c_1 k + c_2 k(k-1)/2$$

with the index:

$$k = (M-0.5)/0.1.$$

The interpolation is limited to the range of validity of the data and is kept constant to the respective extreme outside. These polynomials are used to compute the thrust and power coefficients for the rotor.

As a numerical example the tip Mach number of the rotor Mt is set to 0.7, the range of integration varies from 20% to 90% of the blade span. The rotor solidity is set to 20%. For a constant 5 degree pitch we obtain:

	$x_m = 15\%$	$x_m = 20\%$	$x_m = 25\%$
t = 10%	0.0318	0.0318	0.0318
t = 11%	0.0320	0.0320	0.0319
t = 12 %	0.0320	0.0320	0.0320

Table 7 : Thrust Coefficient (C_T) for $\theta = 5°$

	$x_m = 15\%$	$x_m = 20\%$	$x_m = 25\%$
t = 10%	0.0031	0.0029	0.0028
t = 11%	0.0032	0.0030	0.0029
t = 12 %	0.0034	0.0031	0.0030

Table 8 : Power Coefficient (C_P) for $\theta = 5°$

The results from Table 7 indicate a constant level of thrust around the region of analysis while Table 8 reveals a clear direction of minimization of power requirements. The region could be subdivided or gradients could be computed by finite differences to support an optimization procedure, since the data for the lift slope and profile drag have already been modeled.

Blade twist can be easily introduced and optimized without the need of re-running the CFD code. Different planforms can be analyzed, although limited to blade area effects.

At this point the designer has the choice of attempting to find an optimum using a classical optimization procedure, or even by visual inspection of the chart restart the process choosing another region of the design space.

The accuracy of the whole procedure can be adjusted without any change in the philosophy. The number of design parameters, the degree of the interpolating polynomials can be chosen according to the available resources. This growth capability can turn to a competitive advantage since no change in the methodology is necessary when the equipment is upgraded.

Conclusions

The coupling of CFD tools with classical blade element-momentum theory in a parallel computing environment can lead to an effective design methodology for preliminary rotor design. This philosophy could extended to take into account wake effects and can also be easily modified for forward flight analysis which is also being considered for a near future.

Acknowledgements

The author wishes to acknowledge the support from the National Center of High Performance Computing (CENAPAD-SP) for the use of the IBM-SP1 machine.

References

1 - Srinivasan, G.R., McCroskey, W.J.; "Navier-Stokes Calculations of Hovering Rotor Flowfields", Journal of Aircraft, vol. 25, no. 10, 1988.

2 - Gray, R.B., McMahon, H.M., Shenoy, K.R., Hammer, M.L.; "Surface Pressure Measurements at Two Tips of a Model Helicopter Rotor in Hover", NASA CR 3281, 1980.

3 - Wey, T.C., Gray, R.B.; "A Numerical Investigation of Blade Tip Loadings on Thick-Bladed Helicopter Rotors", Computers and Mathematics with Applications, vol. 12[a], no. 1, pp.53-83, 1986.

4 - Webster, R.S., Chen, J.P., Whitfield, D.L.; "Numerical Simulation of a Helicopter Rotor in Hover and Forward Flight" , AIAA paper 95-0193, 33rd Aerospace Sciences Meeting and Exhibit, January 9-12, Reno, NV, USA, 1995.

5 - Huff, D.L., Wu, J., Sankar, L.N.; "Analysis of Viscous Transonic Flow over Airfoil Sections", AIAA paper 87-0420, 25th Aerospace Sciences Meeting, Reno, NV, USA, 1987.

6 - Stepniewski, W.Z., Keys, C.N.; *Rotary Wing Aerodynamics*, Dover Publications, New York, USA, 1984.

7 - Seddon, J.; *Basic Helicopter Aerodynamics*, AIAA Educational Series, 1990.

8 - Pulliam, T.H.; "Artificial Dissipation Models for the Euler Equations ", AIAA paper 85-0438, 23th Aerospace Sciences Meeting, Reno, NV, USA, 1985.

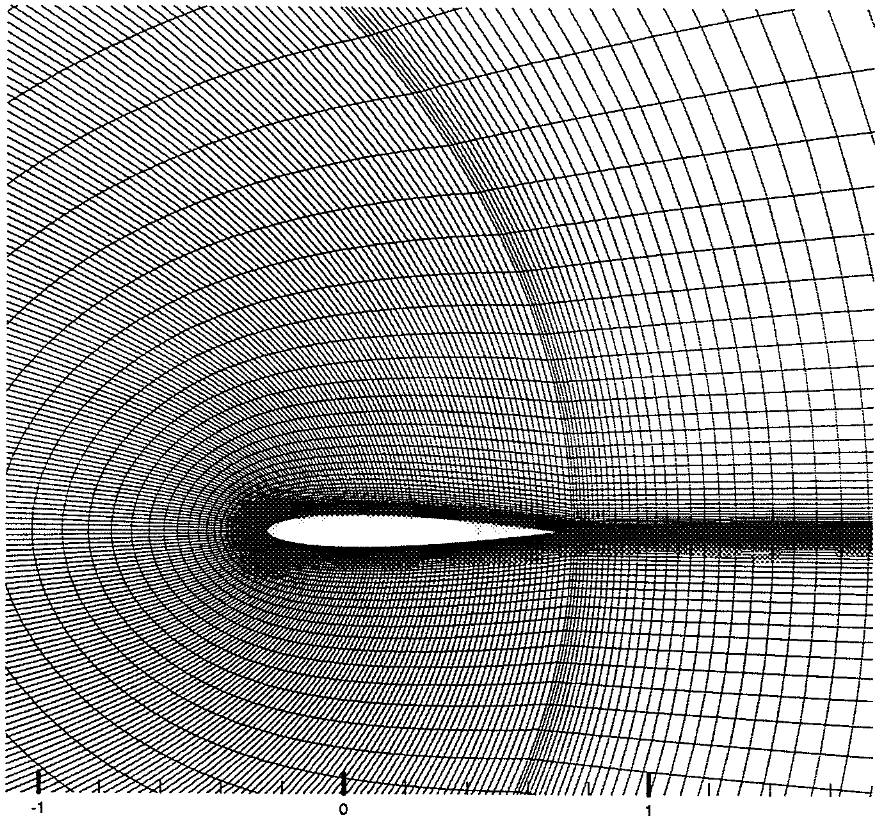

Figure 1. Body-Fitted Grid System for Navier-Stokes calculations

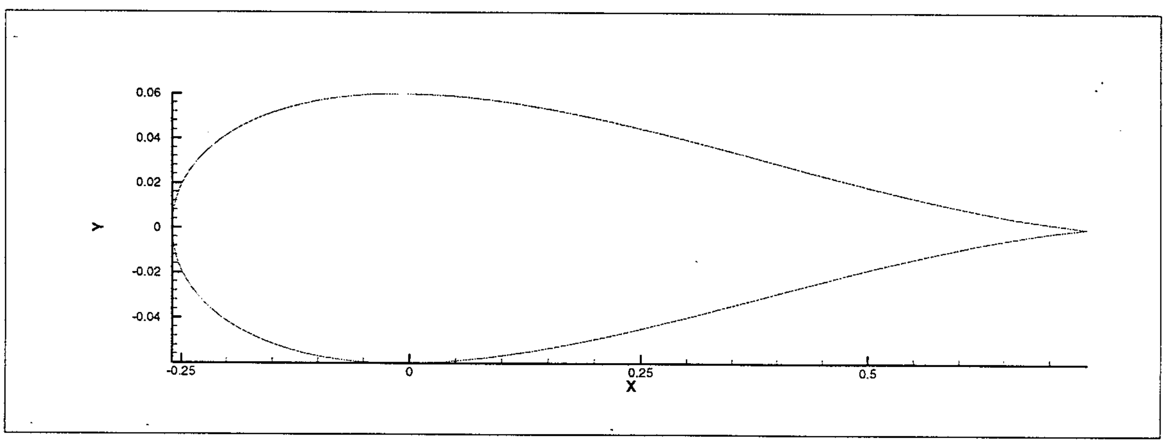

Figure 2. Airfoil Geometry for $t = 10\%$ and $x_m = 25\%$