GRANT
IN-02-CR
26647
P.17

ANALYSIS AND DESIGN OF PLANAR AND NON-PLANAR WINGS FOR INDUCED DRAG MINIMIZATION

SEMI-ANNUAL STATUS REPORT
December 1990 – June 1991

NASA Grant NAG-1-1198

ATTN: Raymond E. Mineck
Technical Monitor

Dennis M. Straussfogel
and
Mark D. Maughmer

Department of Aerospace Engineering
The Pennsylvania State University
University Park, PA 16802

(NASA-CR-188656) ANALYSIS AND DESIGN OF PLANAR AND NON-PLANAR WINGS FOR INDUCED DRAG MINIMIZATION Semiannual Status Report, Dec. 1990 - Jun. 1991 (Pennsylvania State Univ.) 17 p CSCL 01A G3/02

N91-27131

Unclas
0026647

July 1991

TABLE OF CONTENTS

I. SUMMARY

The research activity thus far has concentrated on looking at available methods for predicting induced drag, and exploring how planform and non-planar geometries influence these predictions. Thus far, only methods applicable to incompressible flow have been investigated; however, since the ultimate goal of this research is to develop design methodologies suitable for compressible (transonic) flow, some consideration is being given to more sophisicated methods. Calculations using lifting-line theory, Eppler's modified lifting-line theory, vortex lattice methods, and panel methods have been made. For each of these methods the influence of the freely deforming wake has been investigated. In considering the trade-off between predictive capability and computational intensity, it appears likely that the most suitable design methodology will be a hybrid method which initially uses an incompressible flow method to get as close as possible to an optimum design, and then shifts to an Euler equation solver to account for compressibility and to refine the design.

A significant obstacle in designing wings which take advantage of unconventional geometries for lowering the induced drag is that the gains expected are of the same magnitude as the accuracy of the methods used to predict them. In addition to the errors introduced by discretization, as well as the inherent difficulties due to the singular behavior at the tips, a real question exists with regard to the influence of the relaxed wake. Clearly, it is the effect of the relaxed wake, neglected in classical linear methods, which is responsible for much of the predicted reduction in induced drag for non-conventional wing geometries. The computed wake shapes obtained by iterating to make the trailing filaments tangent to the computed local streamlines, however, are not in close agreement with actual wake shapes. Such non-physical wake shapes may or may not produce predictions which are in better agreement with the actual induced drag than can be obtained with rigid wake models. It remains to more fully explore this question.

As the goal of this research is to provide methods for the design of optimal wing ge-

ometries, the current activity centers on evaluating the suitability of available design tools. Given that the designer of the optimal wing must also consider profile drag, wave drag, weight, bending moment, and so forth, it is planned to pursue optimal designs using available methods, and, by assessing the magnitude of the possible errors, determine whether or not the expected induced drag reductions are real and large enough to justify additional refinement of induced drag prediction techniques. Toward this end, the immediate direction of the research is to use available methods to predict the performance of the planar wings used in the recent wind-tunnel tests at NASA LaRC. In addition to calibrating the design methods, insight will be gained with regard to the errors which might be expected. Unfortunately, this effort will not be conclusive due to the difficulty in predicting the contribution of the wing/body juncture. In any case, if the predictions are in reasonable agreement with the experimental results, the design of the wind-tunnel experiment to explore the potential benefits of non-planar wing geometries will be undertaken. As a direct comparison between a planar and a non-planar wing is not possible, it is planned to design an optimum planar and an optimum non-planar wing for the same mission requirements. In this way, it is hoped that a definitive answer will be obtained to the question of whether or not any significant benefits can be achieved with non-planar wing geometries.

If the low-speed results are promising, the research will next consider compressibility effects. While it is not expected that compressibility will directly influence the induced drag results, it may do so indirectly, as do Reynolds number effects. For example, the amount of area increase that can be used for a non-planar geometry before a reduction in induced drag is offset by increased drag due to additional wetted area is strongly dependent on the profile drag. If compressibility causes the profile drag to increase due to wave drag or the loss of laminar flow, then the amount of area increase that can be tolerated will be reduced.

II. INTRODUCTION

Improvement in the aerodynamic efficiency of commercial transport aircraft will reduce fuel usage with subsequent reduced cost, both monetary and environmental. To this end, the current research is aimed at reducing the overall drag of these aircraft, with specific emphasis on reducing the drag generated by the lifting surfaces. The ultimate goal of this program is to create a wing design methodology which will optimize the geometry of the wing for lowest total drag within the contraints of a particular design specification. The components of drag which must be considered include profile drag, induced drag, and wave drag. Profile drag is dependent upon, among other things, the airfoil section and the total wetted area. Induced drag, which is manifested as energy left in the wake by the trailing vortex system, is mostly a function of wing span, but also depends on other geometric wing parameters. Wave drag of the wing, important in the transonic flight regime, is largely affected by the airfoil section, wing sweep, and so forth. The optimization problem is that of assessing the various parameters which contribute to the different components of wing drag, and determining the wing geometry which will generate the best overall performance for a given aircraft mission.

The primary thrust of the research effort to date has been in the study of induced drag. Recently reported finding have indicated that induced drag may be reduced by utilizing unconventionally shaped and/or non-planar wing planforms.[1-3] These findings are being investigated and the effect on the overall drag of these geometries, including drag components other than induced drag, are being assessed. Other design considerations, such as structural weight, total span, wing root bending moment, and so forth, will also be evaluated. In assessing the cost/benefit of these wing designs, it is necessary to predict the induced drag with a reasonable amount of reliability and precision. Work is being done to create analysis tools which can perform this function.

III. BACKGROUND

In order to calculate the induced drag generated by a lifting surface it is required that all, or at least part, of the local velocity field be determined in the vicinity of the wing. Potential flow methods generally solve for the velocity over only a small part of the flow field and thus save a tremendous amount of computation time. The induced drag is calculated in these methods by either applying the Kutta-Joukowski law to the bound vorticity or by integrating the streamwise component of pressure on the surface of the wing. In this way potential flow methods require solution of the velocity field only at points defining the idealized lifting surface as opposed to points defining the entire flow field. The potential flow methods which employ the Kutta-Joukowski law determine the downwash velocity at the wing either by direct calcuation or by analyzing the flow in the far-wake where the flow is assumed to be two-dimensional (i.e. in the Trefftz plane) and relating that solution to the flow at the wing. The latter technique assumes the wake of the wing to be rigid and aligned with the free-stream velocity. A more computationally intensive approach to calculate the induced drag is to solve the governing equations over the entire "region of influence" in the flow field. The induced drag is then determined by integrating the resulting distributed pressure force on the wing surface, or directly from the cacluated vorticity shed into the wake. The amount of computer time required to solve the governing equations makes this approach impractical as a preliminary design tool.

The following is a brief explanation of available methods for calculating induced drag and a discussion of the strengths and weaknesses of the potential flow methods, as well as methods which numerically solve the Euler or Navier-Stokes equations.

Lifting-Line Theory (Prandtl-Lanchester)

The lifting-line theory of Prandtl analyzes the flow field as a potential field with the wing modeled as a singularity in the form of a line vortex of varying strength located at the wing quarter-chord point.[4] Helmholtz's theorem requires that the spanwise change in vorticity of the lifting line be shed into a sheet of distributed trailing vorticity. The trailing vorticity is assumed to be aligned with the free-stream velocity and to extend downstream to infinity. The strength of the trailing vortex sheet at any point is equal to the spanwise change in vortex strength at the corresponding point on the lifting line. In this model, the sheet of trailing vorticity is assumed to not deform under its own induced velocity (i.e. a rigid wake). The velocity that the trailing vortex sheet induces on the lifting line is used to calculate the induced drag of the wing. Munk used the lifting-line theory to calculate the optimum spanwise lift distribution for minimum induced drag, within the context of the given assumptions.[5] For minimum induced drag the induced velocity normal to the lifting line must be proportional to the cosine of the local dihedral angle. For a straight lifting line (dihedral angle equal to zero everywhere along the span) the lift distribution that generates this induced velocity distribution is elliptical. For a curved lifting line, which models a non-planar wing with spanwise varying dihedral angle, the optimum lift distribution for minimum induced drag is well defined, again, within the limits of the modeling assumptions.[6] Several questions arise, however, regarding these assumptions. The lifting-line model ignores the effect of the chordwise distribution of vorticity on the downwash distribution since it collapses all the vorticity generated at a given spanwise location to a single point. Also, the effect that the deforming wake might have on wing performance is neglected.

Vortex-Lattice Methods

The vortex-lattice method uses an array of horseshoe vortices with spanwise segments bound to the wing and streamwise segments trailing downstream from the trailing edge parallel to the free-stream velocity. The strength of each vortex is determined by satisfying the condition that the flow be tangent to the mean camber line of the wing at a number of control points equal to the number of vortices used. This constraint defines a system of simultaneous linear equations which can be solved for the vortex strengths. The strengths of the streamwise trailing vortex filaments are taken as the sum of the strengths of the horseshoe vortices distributed over the chord at a given spanwise position.

Modeling the wing as a lattice of vortices attempts to capture the effect of the chord-wise loading on the overall wing aerodynamics. The vortex-lattice method does not capture any thickness effects in that it models the wing as a set of discrete line vortices located on the mean camber line. The traditional vortex-lattice method also does not account for the influence of the deformed wake (i.e. wake roll-up). Typically the wing wake is modeled as straight non-deforming vortex filaments aligned with the free stream.

Induced drag is normally calculated in the vortex-lattice method by applying the Kutta-Joukowski law on the spanwise bound vortex segments under the influence of the local downwash. Some research has been done regarding the way in which the lattice is constructed.[7] For example, the question arises as to whether the spanwise vortex segments should be aligned perpendicular to the free-stream velocity, aligned with the sweep angle of the wing, or aligned with some other direction depending on the wing planform shape. It is found that the choice of lattice shape can have a significant effect on the accuracy and order of convergence of the model.

Some researchers have attempted to model the wake deformation by calculating the local velocity at points in the wake, aligning the trailing vortex filaments with the local velocity, then iterating until convergence to a steady state wake shape.[8] The resulting computed wakes show some tendency towards a rolled-up wake shape, but fail to converge to the two discrete trailing vortices observed in experiment. More precise analysis of deforming vortex sheets have been attempted recently by other researchers and it has been noted that even for a simple two-dimensional vortex sheet problem "the calculation of the self-induced motion of vortex sheets has proved quite intractable and has resisted the best efforts of numerous investigators."[9] Consequently the results from a simple vortex-lattice analysis of a deforming three-dimensional wake should be considered with some skepticism. These models have shown that the deforming wake does have a significant effect on the lift distribution of the generating wing which in turn effects the induced drag.

So, it is reasonable to conclude that an error in the computed wake shape will indeed effect the predicted performance of the wing.

Panel Methods

Panel methods discretize the wing upper and lower surfaces into source, doublet, or vortex panels which induce a perturbation on the uniform (free-stream) velocity field. Low-order panel methods assume the panels to be flat and have constant source, doublet, or vortex strength over the entire panel.[10] Higher-order methods consider surface curvature and source, doublet, or vortex strength derivative effects. The strength of each panel is determined by satisfying the flow tangency condition at a number of control points equal to the number of panels used.[3] As in the vortex-lattice method, the flow tangency boundary condition produces a system of linear simultaneous equations that can be solved for the panel strengths. The shape of the freely deforming wake can also be computed in a way similar to the vortex-lattice method. The wake is discretized into panels and the flow velocity at each panel calculated. The wake is reoriented so that the panels are aligned with the local velocity vector and the computation repeated until a steady-state wake shape is converged upon.

Unlike the vortex-lattice method, panel methods take into account the effects of wing thickness. This should have little effect on induced drag calculations, but may be important in analyzing profile and wave drag.

For panel methods, induced drag can be calculated by taking the streamwise component of the product of surface pressure and panel area summed over all the wing panels. This method is extremely sensitive to errors in the calculated pressure distribution which are most pronounced near the leading edges and wing tips, even in higher-order methods.[3] Another means of calculating induced drag is to either assume a fixed (non-deforming) wake or attempt to compute the deformed wake shape, then numerically integrate over the velocity field far downstream where the flow is assumed to be two-dimensional. If a fixed wake is assumed, this method does not account for the deforming wake's influence on the lift distribution and induced drag. If a deformed wake is computed, the part of the wing which has the most profound influence on the shape of the deformed wake, namely, the tip region, is exactly where the solution is known to be least accurate. This is the fundamental problem in any free-wake analysis using the analytic or numerical methods currently available.

Modified Lifting-Line Theory (Eppler)

Another potential flow method being considered is one developed by Eppler in which the lifting line is located at the trailing edge of the planform instead of along the quarter-chord line.[11] As in Prandtl's lifting-line model, the effects of chordwise loading are not included; however, the influence of the trailing edge shape is taken into account. It is assumed in this method that the bound vorticity does not influence the induced velocity and is therefore not considered in the downwash calculations. Induced drag is calculated in this method by applying the Kutta-Joukowski law to the bound vorticity at the trailing edge. The Eppler method can be implemented with either fixed- or free-wake analysis, and can consider planar and non-planar wing planforms. The free wake analysis will suffer from the same inaccuracies discussed in regard to vortex-lattice and panel methods. The advantage of the Eppler method is that is possesses the simplicity of the Prandtl lifting-line model but includes some planform effects in the form of the trailing-edge shape. Intuitively, the Eppler model makes sense in that it places the bound vortex line where the vorticity is actually shed into the flow.

As developed, this model predicts the induced drag given a specified lift distribution. Using the method in the inverse problem of determining the optimum lift distribution, trailing edge shape, and spanwise camber for minimum induced drag is currently being explored. Some early results of this analysis have indicated that a nearly straight trailing edge has some performance advantages.

Euler and Navier-Stokes Equation Solvers

Potential flow methods do not include the effects of compressibility and are therefore inadequate for the transonic wing design problem. To handle these effects and to attain a higher level of accuracy in calculating induced drag, a numerical solution of at least the Euler equations with appropriate boundary conditions is required. The solution must be found over a large enough volume of the flow so as to capture the significant upstream and downstream effects on the wing performance. The effects of the freely deforming wake are captured in the solution, as long as the solution grid extends far enough downstream to include the region of significant influence. Once the velocity distribution on the wing is determined, the lift and drag on the wing can be found from a surface pressure integration similar to that used in panel methods. Determining lift and drag from a far-field wake-integration scheme has also been attempted[12]. Even though the Euler equations do not contain any viscosity terms, the numerical solution shows some viscous-like behavior because of the truncation error incurred in the finite-differencing process. By decreasing the gradients in the solution, this *artificial viscosity* will drive the mathematical solution in the same direction as, but at a different rate than, the real viscous effects.

In order to numerically solve the Euler equations over the number of grid points needed for reasonable accuracy, CPU time in the range of 3 CPU hours on a Cray Y-MP is required[12]. This amount of CPU time would be prohibitive for an iterative design process.

To include the effect of viscosity as well as compressibility in the wing design problem, the full Navier-Stokes equations must be used. Numerically solving these equations would require an amount of CPU time larger than that required for solving the Euler equations, and therefore would likewise make this approach impractical for a design procedure.

IV. RESULTS

The initial phase of the research effort has concentrated on the use of potential flow models for analysis of wing performance and the induced drag minimization problem. Even though future efforts will include real fluid and compressibility effects, it is believed that a firm grounding in the characteristics of the idealized flow is necessary before meaningful results for the more complex problem can be pursued. As already discussed, the analysis of the freely deforming wake is a difficult problem even for potential flow models. The only reasonable course of action is to attain an understanding of this simplified case before attempting to analyze the more difficult problem of transonic, viscous flow.

The types of wings being considered in the development of a wing design method include those with non-planar and unconventional planform geometries. The unconventional geometries include highly swept tips, straight trailing edges, and spanwise varying sweep angles. In analyzing these wings, the question of how the freely deforming wake effects performance must be considered. If the wake is assumed to remain rigid and aligned with the free-stream, there is no evidence that Munk's original solution with regard to the optimal lift distribution is not correct.[5] Likewise, the "stagger theorem" asserts that only the total circulation generated at a particular spanwise station is important to the induced drag calculation and dismisses any effect from the streamwise location of the lifting element (i.e. wing sweep). Also, the optimum lift distribution for non-planar geometries is specified as that which induces a velocity normal to the wing which is proportional to the local dihedral angle. Both of these results appear to be correct if wake deformation is ignored.

Recent studies[1-3] examine the question of optimum lift distribution and maximum wing efficiency. These studies use high- or low-order panel methods with various wake iteration schemes to analyze planar wings. Initially, there were questions raised concerning the validity of Munk's theory regarding optimum lift distribution and maximum attainable wing efficiency, but closer examination revealed that the wing planform which generates the most nearly elliptic lift distribution does indeed generate the least induced drag. In particular, the results show that an elliptic chord distribution does not necessarily generate an elliptic lift distribution when the lifting-line model is not used and the effects of a freely deforming wake included. The spanwise variation of sweep also affects the resulting lift distribution.

Previous research[13] using a fixed-wake analysis considers the influence of spanwise camber on the induced drag. Results from this study are presented in Figure 1. The fixed-wake analysis shows no difference between positive and negative dihedral. In contrast, the vortex-lattice and Eppler methods using a freely deforming wake show that positive (tip up) dihedral has a favorable effect, while negative (tip down) dihedral increases the induced drag. This result is presented in Figure 2. These methods predict approximately a 1% decrease in induced drag with positive spanwise camber.

Finally, the results of a sample wing design problem which includes the trade-off between induced and profile drag contributions in a non-planar wing are presented in Table 1. This design employs the same airfoil section as used in the recent wind tunnel tests at LaRC. As expected, the results show that a non-planar wing can be used to reduce the total drag at high lift coefficients while not causing a drag penalty at lower lift coefficients.

In conclusion, the results of the preliminary analyses performed so far indicate that the freely-deforming wake has only a small *direct* effect on induced drag, but does have a significant effect in modifying the lift distribution of the generating wing. In this way, the freely deforming wake has a significant *indirect* effect on induced drag. A design method optimizing planform and spanwise camber for maximum wing efficiency must therefore include an accurate determination of the free wake effects. As previously discussed, the current state-of-the-art in free wake analysis is of questionable accuracy. This is an area that must be addressed for the methodology of wing design to progress further.

V. REFERENCES

1. Van Dam, C.P., "Induced-Drag Characteristics of Cresent-Moon-Shaped Wings," *Journal of Aircraft,* Vol. 24, No. 2, February, 1987.

2. Kroo, I., and S.C. Smith, "The Computation of Induce Drag with Nonplanar and Deformed Wakes," SAE 901933, SAE Aerospace Technology Conference and Exposition, October, 1990.

3. DeHaan, M.A., "Induced Drag of Wings With Highly Swept and Tapered Wing Tips," AIAA-90-3062-CP, AIAA 8th Applied Aerodyanamics Conference, August 1990.

4. Prandtl, L., "Applications of Modern Hydrodynamics to Aeronautics," NACA Report No. 116, 1921.

5. Munk, M.M., "The Minimum Induced Drag of Aerofoils," NACA Report No. 121, 1921.

6. Cone, C.D., Jr., "The Theory of Induced Lift and Minimum Induced Drag of Nonplanar Lifting Systems," NASA TR R-139, 1962.

7. Schlichting, H. and H.H.B.M. Thomas, "Note on the Calculation of the Lift Distribution of Swept Wings," R.A.E. Report No. AERO 2236, December 1947.

8. Butter, D.J., and G.J. Hancock, "A Numerical Method for Calculating the Trailing Vortex System behind a Swept Wing at Low Speed," *The Aeronautical Journal of the Royal Aeronautical Society,* Vol. 75, August 1971.

9. Higdon, J.J.L., and E. Pozrikidis, "The Self-Induced Motion of Vortex Sheets," *Journal of Fluid Mechanics*, Vol. 150, 1985, pp. 203–231.

10. Maskew, B., "Prediction of Subsonic Aerodynamic Characteristics: A Case for Low-Order Panel Methods," *Journal of Aircraft,* Vol. 19, Feb. 1982, pp. 157–163.

11. Eppler, R., "Die Entwicklung der Tragflügeltheorie," *Z. Flugwiss*, November 1987, pp. 133–144.

12. Van Dam, C.P., Koorosh Nikfetrat, P.M.H.W. Vijgen, and C.M. Fremaux, "Calculation and Measurement of Induced Drag at Low Speeds," SAE 901935, SAE Aerospace Technology Conference and Exposition, October 1990.

13. Burkett, C.W., "Reductions in Induced Drag by the Use of Aft Swept Wing Tips," *Aeronautical Journal,* December 1989.

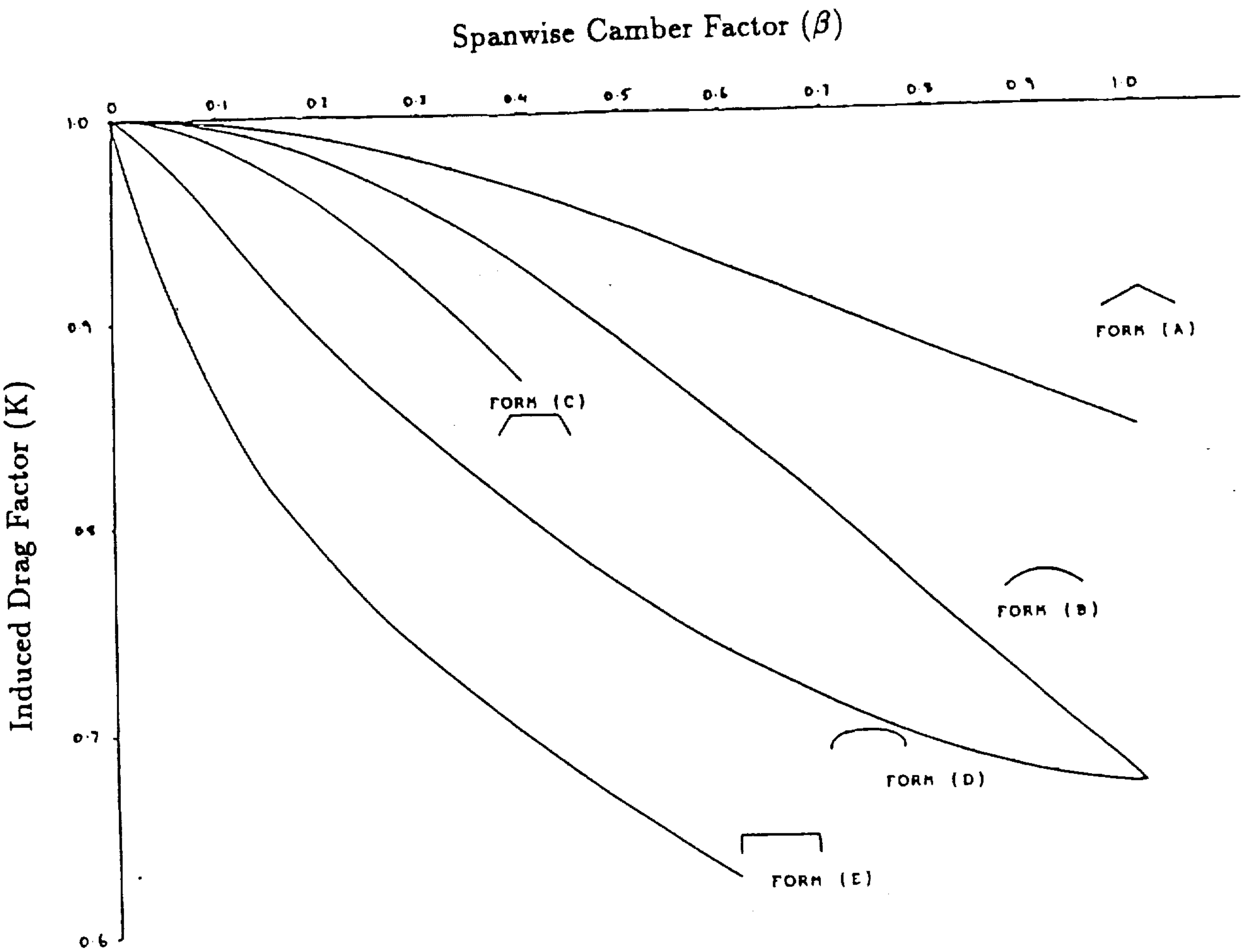

Figure 1: Variation of Minimum Induced Drag Factor with Camber Factor for 5 Forms of Vortex Arc.[13]

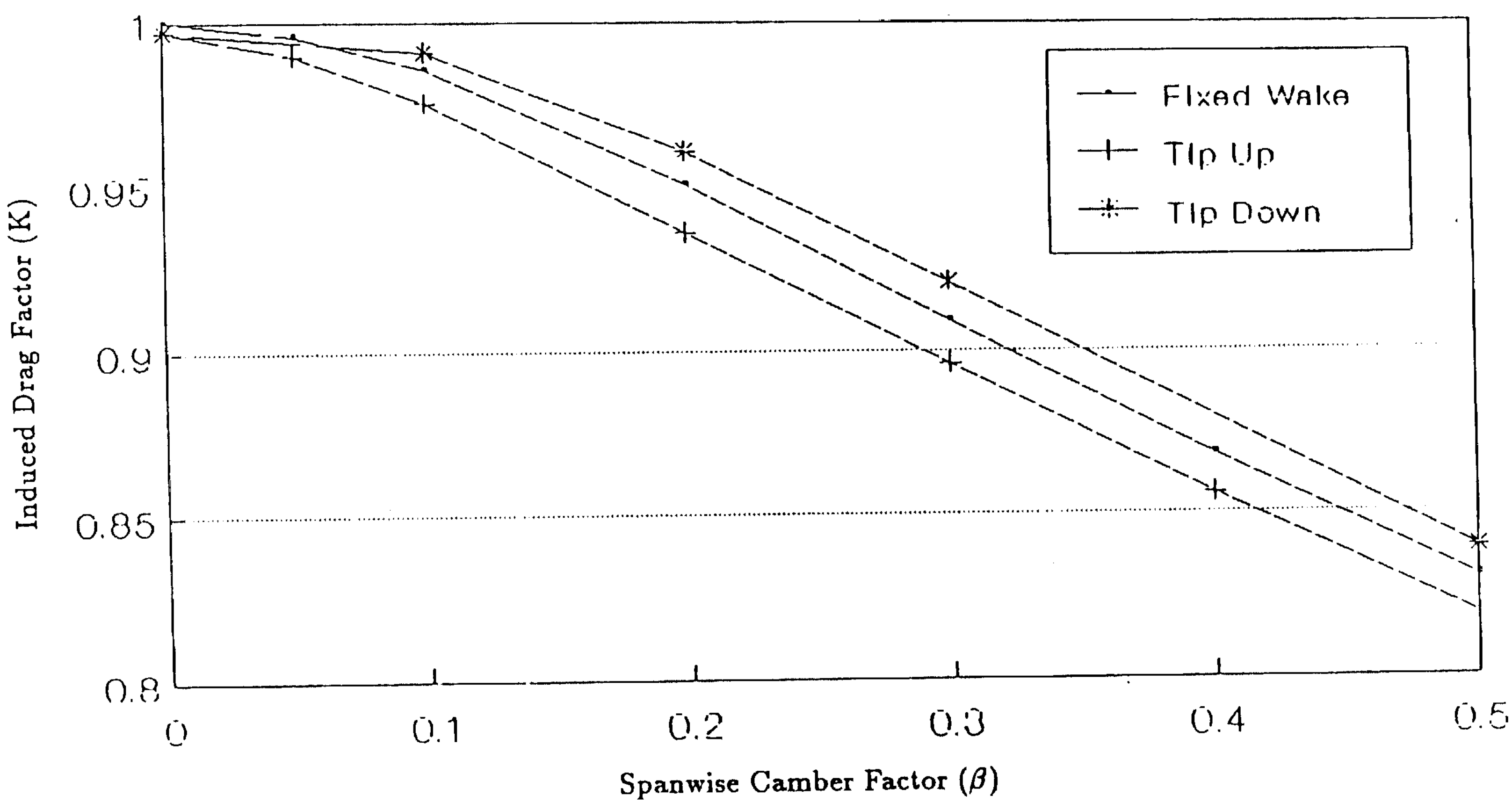

Figure 2: Effect of Relaxed Wake on Induced Drag Factor for Wings with Positive and Negative Spanwise Camber.

	C_L	C_{D_p}	C_{D_i}	C_D	L/D
Planar wing	0.4	.0125	.0072	.0197	20.3
Wing with winglet	0.4	.0131	.0065	.0196	20.4
Planar wing	0.9	.0145	.0362	.0507	17.7
Wing with winglet	0.9	.0149	.0328	.0477	18.9

Taper Ratio = 1/2
Straight Trailing Edge
Root Chord Reynolds No. $= 4 \times 10^6$
Tip Chord Reynolds No. $= 2 \times 10^6$
Airfoil Section: NASA NLF 0416
Section data with forced transition (NASA TP–1861, 1981)
Optimum lift distribution for each C_L

Table 1: Wing Design Trade-offs Using Fixed-Wake Analysis