NASA Contractor Report 165894

A Prescribed Wake Rotor Inflow and Flow Field Prediction Analysis — User's Manual and Technical Approach

T.A. Egolf
A.J. Landgrebe

UNITED TECHNOLOGIES RESEARCH CENTER
East Hartford, CT 06108

NASA

National Aeronautics and
Space Administration

Langley Research Center
Hampton, Virginia 23665
AC 804 827-3966

A Prescribed Wake Rotor Inflow and Flow
Field Prediction Analysis –
User's Manual and Technical Approach

by

T. Alan Egolf
Research Engineer

and

Anton J. Landgrebe
Manager, Aeromechanics Research Section

United Technologies Research Center
East Hartford, Connecticut

June, 1982

Provided to the Structures Laboratory, U.S. Army
Research and Technology Laboratory (AVRADCOM),
NASA Langley Research Center, Hampton, VA

Under Contract NAS1-16058,
Coupled Rotor Fuselage Vibration Analysis

PREFACE

This computer program is provided to the government by the United Technologies Research Center (UTRC) under Contract NAS1-16058, "Coupled Rotor Fuselage Vibration Analysis", for which the Sikorsky Aircraft Division of the United Technologies Corporation was prime contractor. Earlier versions of the program have been provided to the government under Contracts NAS2-6463, Contract DAAJ02-74-C-0027, and DAAJ02-77-C-0027. The second contract formed the original basis for the single rotor, forward flight module of the Rotorcraft Wake Analysis recently provided to the Applied Technology Laboratory, Ft. Eustis.

The UTRC Prescribed Rotor Wake Inflow Program was originally developed under UTRC funding by A. J. Landgrebe during the period 1963 to 1965. Following that period, it has been updated and refined by Mr. Landgrebe, and since 1973 by T. A. Egolf, who is now the principal engineer responsible for the program at UTRC. The Prescribed Wake Rotor Inflow Program formed the foundation for the more comprehensive Rotorcraft Wake Analysis which has been provided to the Government and used by the Army and NASA at Ft. Eustis, the Langley Research Center, and the Ames Research Center. Description of this analysis, application, and correlation thereof are presented in the References. The version described within is the most advanced level of the analysis.

It is recommended that, for each new user application of the program, the authors be contacted at UTRC.

T. A. Egolf, Research Engineer, Aerodynamics Group of Aeromechanics Research Section – (203) 727-7188.

A. J. Landgrebe, Manager, Aeromechanics Research Section – (203) 727-7358.

This will provide the opportunity to provide guidance in the use of the program, input preparation, and output checkout. It will also provide UTRC with feedback on the results and identification of potential refinements and required expanded capabilities.

A Prescribed Wake Rotor Inflow and Flow
Field Prediction Analysis -
User's Manual and Technical Approach

TABLE OF CONTENTS

LIST OF FIGURES

ABSTRACT

A user's manual is provided which includes the technical approach for the Prescribed Wake Rotor Inflow and Flow Field Prediction Analysis, an advanced single rotor version of the UTRC Rotorcraft Wake Analysis. The analysis is used to provide the rotor wake induced velocities at the rotor blades for use in blade airloads and response analyses. A flow field prediction capability is included to provide induced velocities at arbitrary field points such as at a tail surface.

This analysis calculates the distribution of rotor wake induced velocities based on a prescribed wake model. Section operating conditions are prescribed from blade motion and pitch angle data determined by a blade response analysis. The analysis represents each blade by a segmented lifting line and the helical wake of the rotor by discrete segmented trailing vortex filaments. Blade loading and circulation distributions are calculated based on section operating conditions and section lift coefficient data. The strength of the trailing vortex elements are determined based on the spanwise variation in bound circulation. The contributions of each of the trailing vortex segments to the induced velocity distribution are calculated. Using the same relationships and wake model, the capability is also provided to calculate the induced flow field at arbitrary field points about the rotor.

The User's Manual consists of descriptions of the technical approach, the computer program structure, modes of program operation, and the program input and output.

INTRODUCTION

This is a user's manual and brief technical approach for the operation of the most advanced version of the UTRC Prescribed Wake Rotor Inflow Program. This program is a single rotor version of the Rotorcraft Wake Analysis (RWA) described in Ref. 1. This version has all of the fundamental features of the RWA program for forward flight application of single rotors. The major features and capabilities of the Rotorcraft Wake Analysis are shown in Fig. 1. This analysis can be run as an independent computer program or coupled in an iterative manner to an existing aeroelastic blade response analysis based on force and modal integrations (G400), Ref. 2, used in the "Coupled Rotor Fuselage Vibration Analysis" (NAS1-16058), Ref. 3, activity to provide a direct transfer of rotor induced inflow distributions to the blade response analysis for more accurate angle of attack definitions. In this coupled mode most of the required program input is transferred directly to and from the analyses to avoid manual interaction by the user. This manual describes the input and output for the inflow program only.

This computer program can be adapted to interface with other programs which require the rotor wake induced velocities to predict the azimuthal variation of the blade spanwise airload distribution for steady forward flight conditions (e.g., rotor performance, aeroelastic blade response, blade loads, and rotor stability analysis).

2

LIST OF SYMBOLS

a	Blade section linearized lift curve slope, C_ℓ/radian
c	Blade section chord, ft
C_ℓ	Blade section lift coefficient
D	$8\pi R/ac$, radian/C_ℓ
GC	Geometric influence coefficient due to a trailing vortex filament segment (Eq. 7)
$\overline{GC}$	Trailing geometric influence coefficient summed over all blades and all like blade azimuth positions
GCB	Bound geometric influence coefficient for a particular bound segment
GCD	Total geometric influence coefficient associated with bound circulation at a particular radial and azimuthal position
r	Radial location along blade/tip radius
R	Rotor radius, ft
u	Non-induced velocity component at blade, fps
U	Blade section total velocity, fps
v	Rotor/wake induced velocity component at blade, fps
V_{MOM}	Momentum value of induced velocity, fps
w	Axial induced velocity due to a vortex filament segment, fps (Eq. 7)
W_1	Non-induced velocity term in linearized aerodynamic relationships, fps
α	Blade section angle of attack, radians
Γ	Blade bound circulation, ft^2/sec
μ	Rotor advance ratio (tip path plane)

θ	Blade section geometric angle (blade pitch), radians
λ	Rotor inflow ratio (tip path plane)
ϕ	Blade section aerodynamic angle (inflow angle), radians
ψ	Blade azimuth position, radians

Subscripted or Bracketed:

B	Due to bound vortex segment
R	Radial component
T	Tangential component
TPP	Referenced to tip path plane
Tr	Due to trailing vortex filament segment
Z	Axial component
(B)	Due to a particular blade
(I)	Associated with or due to a particular blade element
(J)	Associated with or due to a particular azimuth position
(k)	Due to a particular filament
(L)	Due to a particular azimuthal position
(m)	Due to a particular filament segment
(N)	Due to a particular blade element (radial position)

TECHNICAL APPROACH

General Description and Assumptions

The UTRC Prescribed Wake Rotor Inflow and Flow Field Prediction Analysis (herafter referred to as the rotor inflow program or inflow program) computes the bound circulation and induced inflow distributions along the rotor blades based on a prescribed wake model and a number of assumptions regarding the aerodynamics at each blade section. The blade section operating conditions are generally prescribed from blade motion and control information obtained from a separate blade response program or test data. The wake model is either generated internally in the computer program (undistorted wake) or prescribed from a separate wake geometry prediction program. Procedures have been developed for coupling the inflow program with an aeroelastic blade response program (G400), Ref. 2, in a linked manner as shown in Fig. 2. Descriptions and results pertaining to the development and application of the rotor inflow program in combination with wake geometry and blade response programs are presented in Refs. 1 through 11. In addition the feature to calculate the induced flow at user specified field points is included in this version. The program operates in either of two modes: the circulation solution or field point inflow mode.

Briefly, the mathematical model in the rotor inflow program consists of the representation of each blade by a segmented lifting line, and the helical wake of the rotor by discrete segmented vortex filaments consisting of trailing vorticity which result from the spanwise variation of bound circulation. The circulation of the wake for each blade changes with incremental azimuth position and is periodic for each rotor revolution. The blades are divided into a finite number of radial segments, and the induced velocity at the center of each selected blade segment is computed by summing the contributions of each bound and trailing wake segment. The contribution of each vortex segment (wake element) is obtained through use of the Biot-Savart equation which expresses the induced velocity in terms of the circulation strength of the vortex segment and its geometrical position relative to the blade segment at which the induced velocity is desired. The blade bound circulation distribution is determined by relating the wake element circulations to the bound circulations, expressing the wake induced velocities in terms of the unknown bound vortex strengths by means of the Biot-Savart equation, and developing a set of simultaneous linear equations relating the bound circulation and local blade angle of attack at each blade segment. These equations thus involve the known flight condition, wake geometry, airfoil lift-curve characteristics, blade motion and control parameters, and the unknown bound circulation values. Solution of these equations yields the desired bound circulation values and related wake circulation values, which, when combined with the appropriate geometrical relations in the Biot-Savart law, produce the required induced velocity distribution.

A simplified flow diagram showing the sequence of operations of the program in the circulation solution mode is presented in Fig. 3. The section parameters (airfoil characteristics and geometric angle of attack) are computed from input data. The blade-wake geometry is formulated. The wake influence coefficients (geometric coefficients) at the blades are calculated using the Biot-Savart law. Numerical techniques are used to solve the circulation matrix. Finally, the induced inflow is computed.

The fundamental background for a portion of the analytical formulation used in this single rotor version is similar to the technical approach described by Piziali and DuWaldt in Ref. 12. Although a new computer program was independently developed, several of the fundamental assumptions were retained from that reference.

The rotor inflow program is based on the following formulation and assumptions:

1. Each blade is represented by a lifting line (bound vortex) divided into a finite number of segments (blade segments) each having a different circulation strength (see Fig. 4). The aerodynamic characteristics at the centers of each segment are assumed to be representative of the entire segment over a finite azimuth interval. The use of lifting line theory and two-dimensional airfoil data rather than lifting surface theory results in the satisfying of the aerodynamic relationships at the quarter chord point of each blade segment.

2. The wake is represented by a finite number of vortex filaments trailing from the blade segment boundaries. Each filament is divided into straight segments, the lengths of which are determined by a specified wake azimuth interval which is equivalent to the azimuth interval of each blade (see Fig. 3). The circulation strength of each trailing vortex segment is constant along its length, and is equivalent to the difference between the circulation values of its adjacent bound vortex segments in accordance with the Helmholtz laws of conservation of vorticity. The circulation strengths of different vortex segments along a vortex filament vary in accordance with the variation of the bound circulations with azimuth position.

3. Viscous dissipation effects on the wake circulation strengths are neglected in that the circulation of a given wake segment is constant with time. However, the number of wake revolutions retained in the analysis can be limited to evaluate an abrupt dissipation of the wake. Also, the viscous rollup of the tip vortex can be approximated by combining tip filaments beyond a prescribed rollup azimuth interval. In addition, a vortex core size is assigned to each vortex filament. Potential theory is assumed to apply outside the vortex core. Inside the vortex core zero flow is assumed or a linear variation can be used (zero at the core center to the potential value at the core radius).

6

4. It is assumed that the rotor is operating in steady-state flight. The inflow and wake from each blade is assumed to be periodic with each rotor revolution. That is, the inflow and wake geometry is the same for each blade when at a specific azimuth position.

5. Shed wake segments (segments normal to and connecting the end points of the trailing wake segments mentioned above) arising from the time rather than radial variations of bound vorticity, are not included in the wake model. Although this omission technically violates the Helmholtz law, it is believed that a more accurate representation of the shed wake effect is obtained through the use of experimental unsteady airfoil characteristics in the analysis. This implies that the primary effects of the shed vorticity are those associated with the shed wake immediately behind the blade, and thus can be approximated by those of a fixed-wing type of wake. Miller (Ref. 13) shows that this is a reasonable assumption for most steady flight conditions. This approach not only permits a factor-of-two reduction in computer time and prevents unrealistic results associated with a finite filament shed wake model, but more importantly, permits nonlinear unsteady stall effects to be included in a rational empirical manner in the blade response program.

6. The wake geometry is prescribed from analytical or experimental results. Various options are available for selecting the representation of the wake model. Options for acquiring wake coordinates internal or external to the inflow program are included. To date, the two primary models used are the internally calculated undistorted wake and the transferred distorted wake predicted by the UTRC Wake Geometry Analysis (Ref. 5).

7. The airfoil characteristics at the blade element section are assumed two-dimensional (radial velocity components are neglected). For the linearized circulation solutions and associated calculations, a set of lift curve slopes, stall angles of attack, and angles of zero lift are provided which vary with Mach number and blade thickness. These may be based on unsteady aerodynamic data, if available, and provided directly from a blade response program. The blade section circulation is limited to a constant value for angles of attack above stall.

8. In the blade-wake geometry calculations, the blades are assumed straight (rigid blades). However, flexibility effects may be included in the circulation solution by providing the necessary noninduced velocity at each segment associated with flexible blade motions, and rotor control inputs from a blade response program. Also, an anhedral or swept tip may be prescribed.

9. Blade induced velocities normal to the rotor disk (axial induced velocities) are calculated. Tangential and radial induced velocity components are neglected in the circulation solution procedure. Following the circulation solution, all three components of induced flow velocities can be calculated at selected field points.

7

10. Small angle assumptions are included in the circulation solution.

11. In this version, the aerodynamic interference effects of the rotor hub, fuselage, etc. can be included by the use of an external velocity field input option. The rotor is assumed to be operating out of ground effect.

Wake Geometry

The rotor inflow program requires that the rotor wake geometry be specified for the circulations and induced velocities to be determined. Generality regarding the specification of the wake geometry is provided in the computer program. This is accomplished, if desired, by allowing the wake coordinates for the wake segment end points to be input and read from tape for computational purposes in a prescribed format. This facilitates the adaption of improved wake models to the program as they become available. For this program at the present time there are two options available for wake geometry: (1) a classical undistorted wake model option is available internally, which computes wake coordinates and (2) an analytical wake model (undistorted or distorted wake) can be input from a separate wake geometry program or experimental data as mentioned above.

It has been established that the requirement for wake distortions from the classical type of wake geometry is dependent on the rotorcraft configuration, flight condition, parameters of interest, and accuracy required. For example, an undistorted wake geometry is generally sufficiently accurate for integrated performance calculations (thrust, torque, etc.) for conventional rotorcraft operating in high speed flight. However, for low speed flight conditions, wake distortions are very significant. Also, wake distortions are very significant for conditions with low rotor disk attitudes which result in close proximity of the wake and blades. The calculation of realistic distorted wake geometries requires complex and operationally expensive computer analyses (e.g., Ref. 5). Appropriate experimental wake data is certainly desirable, but is not currently available for most rotor configurations and flight conditions. UTRC is currently developing a forward flight generalized wake model which can be used in this analysis. However, the use of this model should be restricted to applications under the author's direction.

Generally the classical undistorted wake model is used. It has the least complex wake geometry and is simply a function of the flight condition and momentum inflow velocity. The coordinates of this helical wake, which is skewed in forward flight, are generated internally in the program, given the governing parameters. Brief descriptions of the undistorted and distorted wake geometries are provided below.

Undistorted Wake Model

In its simplest form, the wake can be assumed to be a classical undistorted skewed helical sheet of vorticity defined from momentum considerations (hereafter referred to as the undistorted wake). A sample undistorted wake representation is shown in Fig. 5. The coordinates of the undistorted wake representation are obtained from the rotor advance ratio, μ, thrust coefficient, C_T, and rotor angle of attack relative to the tip path plane, α_{TPP}. For example, the top view of the tip vortex filaments is obtained directly from the helicoidal path of the blade tip as it translates at the velocity $V\cos\alpha_{TPP}$ and rotates at the velocity ΩR. The side view image is dependent on the wake skew angle, χ (angle between normal to the rotor disc and wake boundary, Fig. 5), which is normally defined from momentum considerations as follows.

$$\chi = \tan^{-1}\left(\frac{\lambda}{\mu}\right) = \frac{V\sin\alpha_{TPP} - V_{MOM}}{V\cos\alpha_{TPP}}$$

Distorted Wake Model

A distorted wake model may be incorporated in the analysis via tape input. A rotor wake geometry analysis, such as that developed at UTRC and described in Refs. 3 and 4, can be used to generate wake coordinates on tape for use in the rotor inflow analysis. A sample calculated distorted wake geometry (tip vortices) is shown in Fig. 6.

Generalized Wake Model

A program module for the UTRC Generalized Wake Model for forward flight can be included in the future as a subroutine in this version. A first level version of this subroutine module has recently been developed under a separate government contract (NASA Langley Research Center, Contract NAS1-14568, currently in the final stage of completion). Delivery of the subroutine module and documentation is planned for late 1982. Application of this first level wake generalization is limited to specific flight conditions and rotor designs. The mathematical formulation for this model is based on careful study of analytical calculations of the tip vortex geometry for a large number of flight conditions for two rotor systems using the analysis of Ref. 5. The generalization is for the tip vortex only. This generalization is based on perturbations from the undistorted wake model (based on momentum considerations) as functions of the primary rotor wake definition parameters. The axial distortions from the undistorted wake are expressed in terms of an envelope function (E_f) and shape function (S_f).

$$\Delta Z = E_f \cdot S_f$$

These functions have been developed in a form such that the envelope function determines the general amplitude of the distortions, and the shape function defines the general characteristics of the distortions.

$$E_f = A_o \Psi_{age} e^{A_1 \Psi_{age}}, \quad \Psi_{age} \leq 4\pi$$

$$= m \Psi_{age} + B \quad , \quad \Psi_{age} > 4\pi$$

$$S_f = a_{o_i} + \sum_{n=1}^{N} (a_{n_i} \cos n(\Psi_b - \Psi_{age}) + b_{n_i} \sin n(\Psi_b - \Psi_{age})), \begin{array}{l} i=1 \ \ \Psi_{age} \leq 2\pi \\ i=2 \ \ \Psi_{age} > 2\pi \end{array}$$

The wake coefficients have been determined to be functions of the primary distorted wake definition parameters.

Wake Coordinate System

A Cartesian coordinate system is used in the program to define the wake geometry. The coordinate system is illustrated in Fig. 7 and is defined as follows;

1) Coordinates are nondimensionalized by the rotor radius.
2) The origin is fixed at the center of the rotor hub.
3) The x-y plane is parallel to the rotor tip path plane.
4) The coordinate system is a right handed system with z positive upwards.
5) The coordinate system is aligned such that positive x is always in the downstream direction and zero azimuth lies on the positive x axis as illustrated in Fig. 7.

Velocity Coordinate System

The definition of positive velocities used internally in the program is consistent with the wake coordinate system in the cartesian coordinate system. In the blade element system used to define the blade aerodynamics the sign conventions are defined as depicted in Fig. 8.

Circulation and Induced Velocity Solution

Following the formulation of the wake geometry representation in the program, the geometric relations between the wake and the blades are calculated, and the equations for the blade circulations and induced velocities are applied.

10

The Kutta-Joukowski law and the Biot-Savart law are the basic relations used to obtain a closed form solution for these parameters. The Kutta-Joukowski law is used to relate the blade bound circulation and induced velocity distributions. The Biot-Savart law relates the blade induced velocity distribution to the blade-wake circulations and blade-wake geometry. The term "geometric coefficients" will be used to describe the influence coefficients in the Biot-Savart Law. These are functions of the wake geometry as related to the blade positions. Several assumptions, mentioned above, for the airfoil lift characteristics, velocity components, and inflow angles are included in the establishment of the circulation matrix. The solution procedure consists mainly of (1) determination of the geometric coefficients, (2) establishment of the circulation matrix, (3) solution of the circulation matrix by a modified Gauss-Seidel iteration technique, (4) calculation of the induced velocity distribution (inflow) using the results of the circulation solution, and (5) harmonic analysis of the inflow solution and calculation of associated parameters.

In order to summarize the technical approach and indicate where assumptions are made, the primary equations contained in the circulation and inflow solution and a brief derivation thereof are presented below.

The circulation, Γ, of any blade element of chord length, c, at any radial position, r, and azimuthal location, ψ, can be related to the local velocity, U, and the local lift coefficient, C_ℓ, through manipulation of the Kutta-Joukowski relationship between lift and circulation.

$$\Gamma = 1/2 \ cUC_\ell \tag{1}$$

The local velocity U, in terms of induced (v) and non-induced (u) portions is just

$$U^2 = (u_T + v_T)^2 + (u_R + v_R)^2 + (u_Z + v_Z)^2 \tag{2}$$

where the subscripts, T, R, and Z denote tangential, radial, and axial components, respectively.

Linearized Solution

For helicopter applications the induced velocity terms are assumed small compared with U, and are neglected except to define the angle of attack, α. The principles of skewed flow aerodynamics (flow normal to the leading edge) are applied with linearized airfoil data to obtain the following relationship for circulation;

$$\Gamma = 1/2 \ cu_T \ a \ \alpha \qquad (3)$$

where a is the linearized lift curve slope (see section entitled "Linearized Sectional Airfoil Characteristics Input") which is a function of local Mach number and airfoil type.

On replacing the local angle of attack, defined in the plane normal to the leading edge, by its geometric (θ) and aerodynamic (ϕ) components, the relationship becomes:

$$\Gamma = 1/2 \ cu_T a \ (\theta + \phi) \qquad (4)$$

With the assumption of small angles, and on neglecting the tangential induced velocity, the inflow angle, ϕ, is

$$\phi = \frac{u_Z + v_Z}{u_T} \qquad (5)$$

Rearranging the equation for circulation in terms of induced and noninduced contributions yields,

$$\Gamma = 1/2 \ caW_1 + 1/2 \ cav_Z \qquad (6)$$

where $W_1 = u_T\theta + u_Z$ is the noninduced inflow contribution.

In this version of the analysis, the noninduced velocity terms u_z can contain influences other than those due to conventional considerations such as blade motions. The program allows for the input of additional velocities in the u_z, u_T, and u_R terms. These velocities may be due to any source such as fuselage induced flow fields, gust velocities, or other aircraft influences which alter the aerodynamic environment that the rotor blade senses.

The axial induced velocity, v_Z, is expressed using the Biot-Savart law as a function of the trailing wake filaments and circulation strengths, Γ_{T_r}, from all the blades. The axial induced velocity, w, at a given blade segment at any azimuth position, induced by a given wake segment, m, of a trailing wake filament, k, from a particular blade, B, is expressed in terms of the Biot-Savart law as

$$w \ (B,k,m) = \frac{\Gamma_{T_r} \ (B,k,m) \ GC \ (B,k,m)}{4\pi R} \qquad (7)$$

where: R = rotor radius
$GC(B, k, m)$ = axial geometric influence coefficient from the Biot-Savart law for a straight line segment.

12

The geometric coefficient, GC, is a function only of the prescribed wake geometry of the particular segment (B,k,m). The lengthy expressions for GC may be extracted from the program listing. Since the circulation distribution is assumed to be periodic in time (for each rotor revolution) for the forward flight condition, each wake filament has a periodic circulation strength distribution along its length. The total axial induced velocity due to contributions of all trailing circulations for all of the filaments at the blade element of interest is the sum of Eq. (7) evaluated for all segments in the wake for all blades at one instant in time. The resultant expression for the axial induced velocity due to the trailing wake geometries ($v_{Z_{Tr}}$) becomes:

$$v_{Z_{Tr}} = \frac{1}{4\pi R} \sum_B \sum_k \sum_m \Gamma_{T_r} (B,k,m) \; GC \; (B,k,m) \tag{8}$$

However, since the circulations are periodic in m and hence blade dependancy is related to azimuth position, this relationship can be expressed by combining elements with like circulations in terms of new geometric influence coefficients, $\overline{GC}$, which are now functions only of the trailing filament, k, and the blade azimuth position, L, from which the original elements were shed, thus reducing the relationship to:

$$v_{Z_{Tr}} = \frac{1}{4\pi R} \sum_k \sum_L \Gamma_{T_r} (k,L) \; \overline{GC} \; (k,L) \tag{9}$$

In a similar manner, the total axial induced velocity (v_{Z_B}) at a particular blade element due to the bound circulations, Γ_B, of all of the blade elements is,

$$v_{Z_B} = \frac{1}{4\pi R} \sum_N \sum_L \Gamma_B (N,L) \; GCB \; (N,L) \tag{10}$$

where GCB (N, L) is the bound geometric influence coefficient for a particular blade element (N) at an azimuthal position, L.

To define the total axial induced velocity due to all of the trailing circulations in terms of the bound circulation, the circulation strength of each trailing filament segment may be expressed as the difference between the bound circulation of the two blade segments adjacent to its point of origin at the blade at the time of generation. Thus,

13

$$\Gamma_{T_r} (N,L) = \Gamma_B (N,L) - \Gamma_B (N-1,L) \tag{11}$$

Combining both the trailing wake and bound vortex contributions for the axial induced velocity at a blade element segment and with the use of Eq. 11, one obtains the following relationship for the total axial induced velocity,

$$v_Z = v_{Z_{T_r}} + v_{Z_B} = \frac{1}{4\pi R} \sum_N \sum_L \Gamma_B (N,L) \; GCD \; (N,L) \tag{12}$$

where

$$GCD \; (N,L) = \overline{GC} \; (N-1,L) + GCB \; (N,L)$$

Combining Eq. 12 with Eq. 6 yields a linear equation for the circulation of a blade element segment, I, at an azimuth location, J.

$$\Gamma \; (I,J) = 1/2 \; caW_1 + \frac{ca}{8\pi R} \sum_N \sum_L GCD \; (N,L) \; \Gamma \; (N,L) \tag{13}$$

This relationship can be rewritten as

$$\Gamma \; (I,J) \{ D(I,J) - GCD \; (I,J) \} - \sum_N \sum_L GCD \; (N,L) \; \Gamma(N,L) = 4\pi R \; W_1 \; (I,J) \tag{14}$$
$$N{\neq}I, \; L{\neq}J$$

where

$$D(I,J) = \frac{8\pi R}{ca}$$

This equation is used to construct a system of M linear equations with the same number of unknowns of the matrix form

$$Ax = B \tag{15}$$

where

$$M = I \times J$$

Because of the diagonally dominant character of this system of equations, a modified Gauss-Seidal iteration technique is used to solved for the unknown bound circulation strengths. The resulting induced velocities are then calculated from Eq. (12).

Nonlinear Solution

This version of the analysis provides, as an optional feature, the ability to calculate a nonlinear solution which includes all three components of induced velocity if so desired. The nonlinear solution removes the small angle assumptions and the blade element velocity assumptions noted in the description of the linearized solution. In addition, the use of tabulated airfoil data replaces the linearized lift curve slope and stall model. It should be noted that generally this feature is not exercised because the limited increase in accuracy for conventional operating conditions does not justify the increase in computer cost to obtain the solution.

The nonlinear solution is based on the linearization of the relationships given above to form a system of linear equations. These equations can be corrected for the actual nonlinearities of the problem by a lagged iteration procedure described below. Equation 16 can be written as,

$$\Gamma^{(n)} = \frac{1}{2} a c \left(W_1 + v_z^{(n)} + C_f^{(n-1)} \right) \tag{16}$$

where C_f represents the correction for the nonlinearities of the equation lagged in iteration time (n). Note that $v_z^{(n)}$ and $C_f^{(n-1)}$ are functions of the unknown and known circulation ($\Gamma^{(n)}$ and $\Gamma^{(n-1)}$) respectively. The relationship for $C_f^{(n)}$ is

$$C_f^{(n)} = U^{(n)} \frac{C_\ell(\alpha^{(n)})}{a} - U_T \quad \theta + \frac{(u_z + v_z)}{u_T} \tag{17}$$

when $U^{(n)}$ and $\alpha^{(n)}$ include the influence of the appropriate induced velocity components and are not linearized. Thus in the same manner as Eq. 13 was obtained, the relationship for the circulation now becomes;

$$\Gamma^{(n)}(I,J) = \frac{1}{2} caW_1 + \frac{ca}{8\pi R} \underset{N}{\Sigma} \underset{L}{\Sigma} GCD(N,L) \, \Gamma^{(n)}(N,L) + \frac{1}{2} caC_f^{(n-1)} \tag{18}$$

or (c.f. Eq. (14)) as:

$$\Gamma^{(n)}(I,J)\{D(I,J)-GCD(I,J)\} - \underset{N \neq I}{\Sigma} \underset{L \neq J}{\Sigma} GCD(N,L)\Gamma^{(n)}(N,L) = 4\pi R(W_1 + C_f^{(n-1)}) \tag{19}$$

This formulation results in a system of M simultaneous linear equations with the same number of unknowns expressed in matrix form as

$$Ax^{(n)} = B + C^{(n-1)}(x^{(n-1)})$$
(20)

This system is solved for each iteration (n), starting with the linear solution ($C_f = 0$), to obtain a converged nonlinear circulation solution. The matrix solution technique for any iteration step is again a modified Gauss-Seidel iteration procedure with a self adjusting relaxation algorithm to help stablize and speed up the nonlinear iteration solution procedures. The resulting induced velocities are again computed from Eq. 12.

Flow Field Induced Velocity Calculation

For the calculation of the induced velocity at user selected field points other than the rotor blade quarter chordline, the analysis calculates the geometric influence coefficients of Eq. 12 for the selected field point locations. The blade bound circulations ($\Gamma(I,J)$) are input to the analysis (calculated from an earlier run or obtained from alternate sources) and all three components of induced velocity are calculated using Eq. 12 for the axial component and similar equations for the radial and tangential components.

In order to provide the user with the ability to define flow field induced velocities which are compatible with the UTRC RIEVA program which internally calculates the near field rotor induced velocities for a tail surface, the user is given the option of modifying the field point induced velocity predictions. One method is by zeroing out the bound circulation at selected points on the rotor disk. This allows the user to isolate near and far wake influence as required for a particular application by zeroing the appropriate bound circulations which result in the trailing circulations of trailing filaments which are near a particular field point. This method results in regions of the wake (a strip) in the direction of the flight velocity whose influences are nulled. An alternate method can be used which allows the user to specify physically unrealistic vortex filament core radii on the inboard vortex filament sheet model and/or the tip vortex filaments. Within these vortex filament core radii the induced influence is zeroed. This method results in a sphere of influence about the field point for which filaments inside this sphere have no influence. Both methods can be employed simultaneously to define near and far wake influence models.

COMPUTER PROGRAM STRUCTURE

This version of the Prescribed Wake Rotor Inflow Analysis Program was originally programmed in FORTRAN IV computer language for use on the UNIVAC 1110 high-speed digital computer. The program presented has been converted for use on IBM 370 and 360 computers and the CDC 6600 and Cyber series computers.

Program Structure

The structure of the program consists of an initial routine (MAIN) and a series of subroutines. The symbolic names for the program subroutines are listed below. A brief description of each of the subroutines is presented in the following section. A diagram of the subroutine calling sequence is presented in Fig. 9. Utility subroutines have not been included in the diagram. Utility subroutines are those routines whose tasks are routine, used in different areas of the program, or not considered significant to the understanding of the program structure and concepts in which it is based.

Main Routine

MAIN

Subroutines

AERPAR	AFDATA	ANALYS	BCIN	BLDXYZ
CALATT	CALBET	CFUNCT	CIRPCH	CIRPRT
CLASWK	DATAIN	DISWAK	FPVICL	GCCAL
GCJACK	GWAKE2	GWCOEF	INTMAT	INTVEL
LAERO	LINKUP	PCOORD	PRINIT	PRINT2
READFP	READIJ	READWI	SIEDEL	SOLVEM
TAILLC	TIPWAK	UAERO	VICAL	VTHETA

Utility Subroutines

AFCIP	BLIN4	CLCHEK	CLDUMP	CLOAD
CONPLT	HARCAL	HARM	INOUT	INTERP
INTPO	LOADER	PCHHAR	PRTGC	RDHARM
SETTAB	UNINT	WRHARM	WRTGC	

Brief Description of Program Subroutines

AERPAR

This subroutine computes blade element aerodynamic quantities required for the circulation matrix.

AFCIP

This subroutine is used in the utility airfoil data package.

AFDATA

This subroutine controls the input of the real airfoil data tables as required in the analysis.

ANALYS

This subroutine controls the harmonic analysis of the inflow solution and controls associated output options for the results of the harmonic analysis.

BCIN

This subroutine reads in additional noninduced axial velocities at the rotor blades due to an external influence such as a fuselage.

BLDXYZ

This subroutine computes blade coordinates, shaft angle quantities, and other wake related parameters.

BLIN4

This utility subroutine performs bi-variant interpolation on an input vector of a particular form.

CALATT

This subroutine calculates the rotor attitude and transformation quantities based on input data.

CALBET

This subroutine calculates the rotor blades position based on input flapping response data.

18

CFUNCT

This subroutine calculates the nonlinear correction terms for the nonlinear solution.

CIRPCH

This subroutine controls the output of the blade bound circulation distribution.

CIRPRT

This subroutine controls inflow solution output and computes the following quantities for output purposes: blade element angle of attack, stall indicator table, circulation solutions, and the inflow solution.

CLASWK

This subroutine computes the wake coordinates of a classifical undistorted wake. The wake is hub referenced and parallel to the tip path plane. After the wake has been generated, it is written on disk.

CLCHEK

This utility subroutine checks the input airfoil data for the correct formatting.

CLDUMP

This utility subroutine outputs the input airfoil data if it is incorrect in format.

CLOAD

This utility subroutine reads in the real airfoil data if requested.

CONPLT

This utility subroutine plots symbol contour region maps of a specified quantity on the rotor disk.

DATAIN

This subroutine controls input of basic variable data and computes program control associated parameters and constants. These parameters and constants are stored in labeled common blocks used throughout the whole program.

DISWAK

This subroutine controls the input of wake geometry from tape or disk.

FPVICL

This subroutine controls the calculation of the field point induced velocities.

GCCAL

The subroutine calculates the geometric influence coefficients due to the rotor wake and rotor blade for a specified point in space, either at the blade or a field point location.

GCJACK

This subroutine is the major computational subroutine. It computes the geometric influence coefficients of each wake element at each blade radial station and azimuth position. Geometric coefficients for both bound and trailing vortex elements are computed in this subroutine.

GWAKE2 (Not functional in this version)

This subroutine calculates the generalized tip vortex distortions from the generalized wake coefficients.

GWCOEF (Not available in this version)

This subroutine calculates the generalized wake coefficients as functions of the primary wake definition parameters.

HARCAL

This subroutine calculates the value of a quantity at a selected azimuth position from the harmonics of the quantity.

HARM

This utility subroutine will harmonically analyze an input data string using a positive Fourier series.

INOUT

This is a utility subroutine which will perform various input-output operations depending on the entry point which is called.

INTERP

This is a utility subroutine which will linearly interpolate on an input vector of a particular form.

INTMAT

This subroutine reads and sets up matrix coefficients in the required form for the matrix solution technique.

INTPO

This is a utility subroutine which prints two dimensional data arrays in a format which is a function of blade radial location and azimuth position.

INTVEL

This subroutine is used to control the interpolation of inflow velocities and the output of these velocities.

LAERO

This subroutine reads input linearized airfoil data.

LINKUP

This subroutine controls the transfer of rotor response and blade motion data from a blade response analysis.

LOADER

This utility subroutine reads variable input data from cards with a specified format.

MAIN

This is the main program and controls the flow of the program.

PCHHAR

This subroutine outputs the harmonics of the rotor blade induced inflow in punched cards according to a defined format compatible with UTRC blade responses program.

PCOORD

This subroutine is used to print wake geometry coordinates.

PRINIT

This subroutine prints out the basic variable input data in a special output format which is compatible with the loader input requirements. It also prints the variable's location in the array, name description, units and value.

PRINT2

This subroutine prints out most of the blade element aerodynamic quantities associated with the program operation.

PRTGC

This subroutine prints the geometric influence coefficients calculated in the program if requested by a user option.

RDHARM

This subroutine reads in data in harmonic form and converts it to blade element format.

READFP

This subroutine controls the input of field point positions for the calculation of flow field induced velocities.

READIJ

This subroutine controls the zeroing of the bound circulation on the rotor disk as specified by user input.

READWI

This subroutine reads the W1 vector from an external source if so desired.

SETTAB

This subroutine sets up the independent variable vector for an interpolation procedure.

SIEDEL

This subroutine solves the circulation matrix using a Gauss Seidel itera-
tion technique. Incorporated into the iteration technique is a check on
maximum circulation as determined from stall considerations. If any solution
value in the iteration exceeds the maximum circulation, this solution value is
reset to the maximum circulation and iteration continues.

SOLVEM

This subroutine controls the circulation solution procedure for the
analysis.

UAERO

This subroutine reads linearized airfoil data from a blade response
analysis.

UNINT

This subroutine performs linear interpolation.

VICAL

This subroutine uses the circulation solution and the wake influence
coefficients to calculate the induced velocities based on the Biot-Savart
relationship.

VTHETA

This subroutine reads the input for the calculation of blade element
pitch distributions from a blade airloads and response analysis.

WRHARM

This subroutine writes the harmonic analysis results in a compact
output format.

WRTGC

This subroutine outputs geometric influence coefficients to mass
storage.

MODES OF OPERATION

This version of the inflow program can be run as either an independent analysis or coupled to the aeroelastic blade response program G400 (Ref. 2). For the coupled mode of operation, the inflow and blade response programs are run sequentially and the required transferal of output from each program is done internally on mass storage devices. In this mode of operation, sequential executions of either the inflow or blade response program do not require reloading input data. The input data is also saved on mass storage devices. The use of this mode of operation should be left to the user who is familar with both of the analyses mentioned above in their independent modes of operation. Generally, several cycles of iterations between inflow and blade response analysis are required for a converged solution. With this fact in mind, the user is cautioned to be very careful in the setup of the required input data of both analyses. This option is controlled by a single input to the inflow analysis, location 89, in the loader input data. All input parameters which will be transferred from G400 to the inflow analysis in the coupled mode of operation are so noted in the section entitled: "Description of Program Input."

In addition this program has the capability of predicting both the induced inflow at the rotor blades or at selected field point locations. For the induced inflow at the rotor blades, the circulations are calculated internally from prescribed input. For the field point calculations, the circulations are read in from an external source or from an earlier run in the rotor blade induced inflow mode. The control of this mode of operation is input in location 224.

DESCRIPTION OF PROGRAM INPUT

Summary of Input Parameters

The input required for the rotor inflow program are summarized below by grouping the input parameters into descriptive categories.

Flight Condition Parameters:	Flight velocity, rotor tip speed, rotor shaft angle, speed of sound
Rotor Design Parameters:	Number of blades, flapping hinge offset ratio, rotor radius, blade chord, blade twist, blade precone (for hingeless rotor), tip anhedral, tip sweep
Rotor Pitch Control Parameters:	Collective pitch, cyclic pitch
Blade Response Parameters:	Coning (articulated rotor), flapping angles, steady lag angle, blade bending and torsional flexibility (optional)
Rotor Model Parameters:	Radial stations, azimuth increment
Wake Model Parameters:	Azimuth increment, number of wake revolutions, wake transport velocity, vortex core radius, number of vortex filaments to rollup in tip vortex, tip vortex rollup azimuth, inboard wake truncation azimuth
Airfoil Data:	Linearized airfoil data in the form of lift curve slopes, stall angles of attack, angles of attack for zero lift (all vs. Mach number and thickness ratio), and real airfoil data in tabular form if required
Option Controls:	Print, punch, coupling, and input/output controls
Field Point Locations:	Blade bound circulation data and field point positions

Although the number of input parameters appears large, provisions have been incorporated in the program to facilitate the input setup by:

(1) omitting input for unnecessary parameters for specific cases,

(2) avoiding input repetition for multiple cases.

Sequence of Input

The input sequence for the computer program data input is:

1. Basic Variable Input Data (LOADER FORMAT)

2. Multiple Case Control (1 Card)

3. Linearized Sectional Airfoil Characteristics Input

4. Tabulated Airfoil Data (Optional)

5. Blade Element Linearized Airfoil Data (Optional)

6. External Velocity Field Input (Optional)

7. Blade Element Variable Blade Pitch Angle (Optional)

8. Field Point Location Data (Optional)

9. W1 Input (Optional)

10. Tip Vortex Coordinate Over Write Input (Optional)

11. Generalized Wake Coefficient Data (Optional)

12. Blade Bound Circulation Data (Optional)

    For runs with multiple cases, the input sequence for cases following the first case is identical except that repetition of data in the Basic Variable Data is not required.[*] The use of multiple cases is not possible in a coupled mode of operation with a blade airloads and response analysis (e.g., G400).

    A detailed description of the input items follows below. A summary listing of input options is provided at the end of this section, and the options normally used are indicated in the listing.

---

[*] A computer case is referred to as the inflow solution for one combination of rotor design, operating condition, and wake geometry. A computer run is referred to as the inflow solution for one case or multiple cases submitted to the computer at one time.

Basic Variable Input

The basic input format is referred to as LOADER format (as specified by subroutine LOADER). The input items are assigned location numbers within a storage array. The use of this format and subroutine facilitates the changing of input. An input item may be changed simply by specifying the location number and input item value on a card following the card on which the original value for the item was located. In addition, items which are unchanged between consecutive cases need not be repeated in the input for the successive cases. Locations for which no input is included during a computer run will be automatically set to 0.0 unless stated otherwise in the input instructions. The LOADER format is as follows:

LL   L   DATA(L)   DATA(L+1).........DATA(L+4)

where: LL is the word count, i.e., the number of data items on a card (Column 2, fixed point, right adjusted).

L is the location number of the first data item on the card (Column 3-6, fixed point, right adjusted).

DATA represents the various input item values on the card (Columns 7-18, 19-30, 31-42, 43-54, 55-66; floating point numbers).

The location numbers, program symbols, and description of each input item are presented below.

LOCATIONS	SYMBOLS	DESCRIPTION OF INPUT ITEM
1*	V	Flight velocity, kt. This item is used in computations involving various blade element aerodynamic quantities.
2*	OMGR	Rotor tip speed, fps. Product of the rotor rotational velocity (rad/sec) and the rotor radius (ft).
3*	SOUND	Speed of sound, fps. This item is used in calculation of local blade element Mach number.
4	DPSI	Azimuth increment, deg. This item defines the computational azimuth increment of a blade over one revolution. It also is used to divide the trailing wake filaments into elements and to define the azimuth interval of the circulation and inflow solution. 360/(B·DPSI) must be an integer. DPSI must be equal to or greater than 10 deg. 15 deg is normally used for 2, 3, 4, 6, or 8 blades, although 30 deg is used when computer cost is more of a concern than increased accuracy.
5	R	Rotor radius, ft.
6	B	Number of blades
7-21	C	Blade chord for each segment in the inflow solution, ft. Maximum of 15. Ordered from the innermost to the outermost segment.
22	E	Blade flapping hinge offset ratio. Offset non-dimensionalized by the rotor radius, R. Used to obtain blade segment orientation.
23	STNS	Number of blade segments used for the inflow solution. A maximum of 15 is allowed. Normally, 9 segments are used.

---

* These items are transferred from G400 when running in a coupled mode.

28

LOCATIONS	SYMBOLS	DESCRIPTION OF INPUT ITEM
24-38	RS	Radial coordinates of blade segment centers (blade stations) for the inflow solution nondimensionalized by the rotor radius, R. These coordinates are used to define the representative locations of the blade segments for which the aerodynamic parameters are calculated and the inflow solutions are obtained. In addition, the boundaries of the blade segments, which designate the origins of the trailing vortex filaments, are calculated from these segment center locations. The innermost segment center is normally selected so as to position the corresponding innermost segment boundary at the blade root cutout position (inner aerodynamic boundary of blade). The maximum number of blade segments is 15. Careful selection of the RS values must be made to insure that the radial position of each segment boundary between adjacent segments is identical (i.e., segments do not overlap). The innermost value must be the first value of RS. Careful selection of the segment center locations is necessary to avoid introducing numerical inaccuracies associated with the use of finite vortex filaments. Smaller segments in regions of large bound circulation gradients (e.g., at the blade tip) are required. Segment lengths less than 0.02R should normally be avoided. See Fig. 3 and the section, Special Input and Operating Instructions for a typical segment distribution.
39*	STNEW	Number of blade segments for the interpolation of the inflow solution and harmonic analysis of the inflow. If this item is zero, the inflow at the solution segment centers (RS) are used for the harmonic analysis. A maximum of 25 is allowed. In the coupled mode of operation these values must be input and be the same as those used by the blade response analysis.

* These items are transferred from G400 when running in a coupled mode.

29

LOCATIONS	SYMBOLS	DESCRIPTION OF INPUT ITEM
40-64*	RSNEW	Radial coordinates of blade segment centers of interpolation calculations (nondimensionalized by R). These segments are not used in the circulation and inflow calculations. They are used only to interpolate the inflow solutions at the original segments (RS) to these new radial locations. Normally, RSNEW values are not input, and the original RS segments are used throughout the computation. A limit of 25 RSNEW values exists in the program. The innermost value must be the first value in RSNEW.
65*	VIMOM	Wake transport velocity, fps. Positive upflow. This velocity is only used if the undistorted wake model is to be internally generated in the program. It is used to define the transport velocity, normal to the rotor disc. Normally, the momentum inflow velocity is used as defined by the following equation:

$$VIMOM = -\frac{1}{2} \frac{C_T \Omega R}{(\mu^2 + \lambda^2)^{1/2}}$$

where ΩR, C_T, μ, and λ are the tip speed, thrust coefficient, advance ratio, and inflow ratio as conventionally defined in Ref. 12, in the tip path plane coordinate system.

66*	ALPHAS	Rotor shaft angle, deg. Positive shaft angle corresponds to a nose-up tilt of the rotor shaft with respect to the forward flight velocity of the rotor hub.
67*	THET75	Blade collective pitch at the 0.75R stations, deg. This item is used in conjuction with THETA1 and DTHETA to calculate the blade pitch distribution.

---

* These items are transferred from G400 when running in a coupled mode.

LOCATIONS	SYMBOLS	DESCRIPTION OF INPUT ITEM
68	THETA1	Blade linear twist rate, deg. This input item is used to calculate the blade built-in pitch distribution for linearly twisted blades, and is normally negative to indicate decreased built-in pitch angles at the blade tip. The blade pitch, THETA, at each station, I, is calculated from the following equation: THETA(I)=THET75 + THETA1 (RS(I)-0.75) + DTHETA (I)
69-83	DTHETA	Increment of blade pitch angle at each inflow solution blade station (RS), deg. For a non-linear twisted blade, this item is used in place of or in conjunction with THETA1 to define the built-in pitch angle at each station, I, due to blade twist (see above equation). The value for the innermost station must be the first value of DTHETA.
84*	A1SP	The first cosine harmonic coefficient of cyclic pitch, deg. Shaft axis system. Based on negative Fourier series.
85*	B1SP	The first sine harmonic coefficient of cyclic pitch, deg. Shaft axis system. Based on negative Fourier series.
86	DELPSI	Blade phase angle, deg. This item can be used to position the rotor in any desired starting location relative to the zero azimuth position. It is positive in the direction of rotation. Normally not input and blade 1 of the rotor is assumed to lie at zero azimuth position for rotor position 1.
87	DELTA	Blade lag-angle, deg. Positive opposite to the direction of rotation. A constant lag angle is assumed.

* These items are transferred from G400 when running in a coupled mode.

LOCATIONS	SYMBOLS	DESCRIPTION OF INPUT ITEM
88	XLINK	Option to couple with G400. A value of 1.0 is used for the first cycle coupling, a value of 2.0 is used for all subsequent cycles. When running in a coupled mode, the data and JCL is set up for the program as if it were in an independent mode with the exception that the data so marked (*) is not required as loader input. For the subsequent cycles, only the loader input for this location (88) is required; thus, no linearized airfoil data or other loader data is input.
89	GWKOP	Option control for the use of the UTRC generalized tip vortex distortion model. Nonzero input value of 2.0 or 3.0 requests the use of this model. A value of 2.0 specifies internal calculation of the generalized wake distortion coefficients, a value of 3.0 requires card input of these coefficients. See section entitled: Generalized Wake Coefficient Data (not functional in this version).
90	CTCOEF	Input C_T for definition of the internally calculated generalized wake coefficients. A zero input requests the value corresponding to the input VIMOM value and input rotor attitude be used (not functional in this version).
91		Not used.
92	YDIST	Option to use lateral tip vortex distortion in the generalized wake model, input a 1.0 to use (not functional in this version).
93-98		Not used.
99		Multiple case control card, see section entitled, Multiple Case Control.
100	PLTCON	Option to generate symbol contour region plots, input 1.0 to use this option.
101-162		Not used.

32

LOCATIONS	SYMBOLS	DESCRIPTION OF INPUT ITEM
163	AFMAX	Number of blade flapping harmonics. Maximum of 10.
164*	AO	Blade coning angle, deg. This input item is used in conjuction with the offset ratio, E, to orient the blade in the program coordinate system. For a hingeless blade, an effective value of coning angle may be used to approximate the steady bending deflection.
165-174*	AFL	The cosine harmonic coefficients of flapping, deg, (negative Fourier series). A maximum of 10 in sequential order. If W1OPT (Loc. 206) is nonzero, only the first harmonic value is used to define the rotor tip path plane angle.
175-184*	BFL	The sine harmonic coefficient of flapping, deg, (negative Fourier series). A maximum of 10 in sequential order. If W1OPT (Loc. 206) is nonzero, this item can be ignored.
185	XNH	The number of harmonics of induced velocity desired for printout. Must not be greater than 180/DPSI. If left blank or 0.0, 180/DPSI will be used.
186	TYWAKE	Wake model option control. Must be a nonzero value between 1.0 and 4.0. An input value of 1.0 results in a classical undistorted wake. A value of 2.0 requests the generalized wake model (loc. 89 nonzero) be used. A value of 3.0 allows for a classical undistorted wake model with an input prescribed tip vortex overwrite (see section entitled: Input Tip Vortex Overwrite Data. A value of 4.0 requests that the complete wake be read from a tape which contains the geometry in a specified format. This last model is infact the distorted wake output of Ref. 3.

* These items are transferred from G400 when running in a coupled mode.

LOCATIONS	SYMBOLS	DESCRIPTION OF INPUT ITEM
187	RUN	Run number for wake geometry location on tape. A 1.0 is used for TYWAKE = 1 through 3.
188	TSTEP	Time step in tape file to start using wake geometry. Used only when TYWAKE = 4.
189	REV	Number of wake revolutions. Selection of this item is a compromise between accuracy and computer cost. The following values are typical: for hover (advance ratio, $\mu = 0$) use at least 8 revs; for 0 - 0.05 use 6 revs; for 0.05 - 0.15 use 4 revs; for 0.15 - 0.2 use 3 revs; for $\mu \geq 0.25$ use 2 revs.
190	ROLLUP	The number of filaments to rollup in the tip vortex (at rollup angle TRUNC). If rollup is not desired, such as for a classical undistorted wake, use 0.0 or 1.0. If rollup is desired, this value should be the number of tip vortex filaments outboard of the radial station at which the peak circulation occurs. This value is assumed constant (not varying with azimuth position), and thus a representative value must be selected. Since this value is not known a priori, it must be estimated, and if necessary iterated on. In hover the peak circulation normally occurs between 0.9 and 0.96R. In high speed forward flight it normally occurs between 0.7 and 0.85R.
191	ANDTIP	Option indicator for anhedral tip design. A 0.0 indicates no droop, a 1.0 indicates a linear droop computed internally using DROOP, and a 2.0 indicates input droop coordinates (see Loc. 195).
192	DROOP	Linear droop angle for anhedral tip, positive down, deg.
193	RBEND	Radial location for the start of the droop section, nondimensionalized by R. It must be positioned such that there are no more than 5 inflow stations outboard of this location.

LOCATIONS	SYMBOLS	DESCRIPTION OF INPUT ITEM
194	RATIOV	Ratio of the tip filament displacement velocity to VIMOM of the wake. If not input or a 0.0 is input, the value is automatically set equal to 1.0.
195-199	ZANDTP	Axial coordinates of the drooped stations nondimensionalized by R. Referenced to unconed, undrooped blade. Dihedral: positive; Anhedral: negative. Maximum of 5 are allowed. This is used only if ANDTIP is set to 2.0.
200	TRUNC	Angle of rollup of the tip vortex filaments with respect to the blade, deg. Tip vortex filaments beyond this wake azimuth angle are all assigned coordinates equivalent to the outermost tip filament, thereby simulating the rollup of the tip velocity into a concentrated tip vortex. TRUNC/DPSI must be an integer. A typical value of TRUNC is 15. If no rollup is desired, set TRUNC = 0.0.
201	TRUNCI	Truncation angle of vortex sheet filaments with respect to the blade, deg. Normally, TRUNCI = 0.0 which suppresses the truncation feature. If a truncation angle is desired, TRUNCI = wake azimuth angle for truncation.
202	RCORE	Core radius of the tip vortex filament nondimensionalized by the rotor radius. Generally, a value representing approximately 0.1 of the blade chord is used. This value is based on limited experimental observations. Caution – RCORE must not exceed one-half of the length of the outer blade segment for the circulation solution application of this analysis.
203	RCOREI	Vortex filament core radius for filaments representing the inboard vortex sheet, non-dimensionalized by the motor radius. Caution – RCOREI must not exceed one-half the length of the smallest inboard blade segment from which inboard vortex sheet is emitted for the circulation solution application of this analysis.

LOCATIONS	SYMBOLS	DESCRIPTION OF INPUT ITEM
204	DEL3	Pitch flap coupling, tangent of the angle. Sign convention consistent with G400; normally positive.
205	TOL	Tolerance control for the matrix solution of circulation. Normally, set TOL = 0.0 and a tolerance of 0.0005 will be used (tolerance = allowable difference in summation of circulation solutions between last and previous iteration divided by summation for last iteration). Thus, 0.0005 represents a 0.05% accuracy of convergence of the solution summation. If a tolerance value other than 0.0005 is desired, set TOL equal to the desired value.
206	W1OPT	Option indicator for W1 input. If a value of 0.0 is input, W1 is computed from rigid blade equations using flapping harmonics and blade control parameters. A value of 1.0 indicates that W1's will be read in via cards. See section entitled, Noninduced Axial Velocity Component Input, W1.
207	WOPT	Blade element linearized aderodynamics input option. Input a 1.0 to use (see section entitled: Blade Element Linearized Airfoil Data).
208	OPTPO	Geometric coefficients print option. A 0.0 indicates no print; a 1.0 calls for the print option. 0.0 normally used.
209	PRINT	The matrix solution print option. A 0.0 indicates no printout; a 1.0 calls for printout of the matrix coefficients and solution at each iteration, and a 2.0 prints only the solutions at each iteration. 0.0 normally used.
210	PUNCH	Card punch option for induced velocities. A 0.0 indicates no punch cards, a 1.0 requests harmonics of induced velocity, a 2.0 requests induced velocities at each point in the rotor disc, and a 3.0 requests both harmonics and point values. A minus value (i.e., -1.0, -2.0, -3.0) indicates punch or print of only the axial component.

36

LOCATIONS	SYMBOLS	DESCRIPTION OF INPUT ITEM
211	CPOPT	Wake coordinate print option. A 0.0 indicates no print, a 1.0 calls for a printout of all of the wake coordinates, and a 2.0 calls for the printout of only the tip filament wake coordinates.
212	CROPT	Print option for wake-element point combinations within the core radius. A 0.0 input indicates no printout, while a 1.0 calls for the printout.
213	CPUNCH	Option for punched cards of the rotor circulation solutions. A 0.0 indicates no punch, while a 1.0 calls for punched card output.
214-216		Not used.
217	OPVIBC	Option to use input axial velocities at the rotor blades due to an external influence such as flow interference velocities due to a fuselage. Input a 1.0 or 2.0 to request this option. The value of 1.0 requests three components, 2.0 only the axial one. See section entitled: External Velocity Field Input.
218	SKEWFO	Option to use an alternate skewed flow formulation of Sikorsky's Generalized Rotor Performance Analysis (GRP). Input a 1.0 to request this model without stall effects or a 2.0 to include stall effects. (This option should always be used in this version when coupled to GRP, generally with the stall effects.) Use 0.0 for G400 coupling.
219	VIROTR	Option to include the tangential and radial induced velocities in the circulation solution procedure (loc. 293 must be set to 1.0). Normally not used.
220		Not used.

LOCATIONS	SYMBOLS	DESCRIPTION OF INPUT ITEM
221	DEBUGP	Option for intermediate printout, a 0.0 indicates no printout, a 1.0 requests printout. This print option should generally be used.
222	OPCORE	Vortex core option, a 0.0 requests a solid body rotation within the specified core radius, while a value of 1.0 requests no velocity within the core radius.
223	GRPLNK	Option to link to GRP. A value of 1.0 requests this option. This option is not used outside of UTC.
224	FROPT	Option switch to control circulation solution or field point induced velocity calculation mode of operation, 0.0 for circulation solution mode, 1.0 for field point induced velocity mode. See section entitled: Field Point Location Data, and the section entitled: Blade Bound Circulation Data.
225	TAILVI	Option switch for induced velocity calculations at tail surface locations. A 1.0 request that a field point grid compatible with the Rotor Induced Empennage Vibration Analysis, RIEVA (Ref. 15) be calculated internally based on input data. See section entitled: Field Point Location Data. This option is used in conjunction with loc. 224 = 1.0.
226	FPHARM	Option switch to request harmonic analysis of field point induced velocities.
227	HARMOP	If nonzero (1.0), the induced velocities at the rotor blades will be linearly interpolated to an azimuth increment other than DPSI (Loc. 4), and harmonically analyzed for that distribution.
228	HPSI	Azimuth increment to be used when HARMOP is nonzero (5.0 minimum).
229-292		Not used.

LOCATIONS	SYMBOLS	DESCRIPTION OF INPUT ITEM
293	SOLVE	Option switch to request nonlinear solution procedure. A value of 1.0 will request this option. See section entitled: Tabulated Airfoil Data. Normally not used (input 0.0). May be necessary for significantly stalled conditions.
294	GCSAVE	Option to save and use geometric influence coefficients from previous iterations when run in a coupled mode. A value of 0.0 indicates no saving of the geometric influence coefficients, a value of 1.0 requests that they be saved from this iteration, and a value of 2.0 requests that they by read from the previous iteration. This option should only be used when the rotor tip path plane and blade response is not expected to change significantly between iterations.
295	RELAXF	Value of the relaxation factor used in the nonlinear solution procedure. The default value is 0.7 (Generally not used).
296		Not used.
297	AUTORF	Option switch for the use of an automatic internally self-adjusting relaxation factor for the nonlinear solution. A value of 1.0 will request that this option not be used. A value of 0.0 will request that the relaxation factor from location 295 be used.
298-362		Not used.
363	RSWEPT	Blade radial position where sweep begins. This input will determine the location of the blade lifting line, thus it is most important to be sure that the location of the blade trailing filaments and this value are consistent. If not, the radial inflow locations near the value RSWEPT will not lie on the lifting line. To be consistent it is necessary that RSWEPT and a trailing filament location be coincident at the blade.

LOCATIONS	SYMBOLS	DESCRIPTION OF INPUT ITEM
364	ASWEPT	The blade sweep angle, positive in the lag direction.
365	OPTHET	Option switch to read in blade element pitch angle distributions, a 1.0 will request this feature. See section entitled: Blade Element Variable Pitch Angle.
366-380	THICK	Blade thickness ratio distribution, inboard to outboard corresponding to the radial inflow stations (RS). Used for linearized airfoil characteristic lookup.
381		Not used.
382	SAVGAM	Option to save blade circulation solution for the next iteration of a coupled mode of operation execution. Nonzero (1.0 or 2.0) to use this option, a value of 1.0 requests that the solution be saved, a value of 2.0 requests that the previous solution be read to start the matrix solution iteration, and also to save the new solution.
383-384		Not used.
385	COURSE	Option to use a course far wake model. A non-zero value specifies the number of DPSI in wake age from the blade at which the program will use a wake model based on twice the input value of DPSI (loc. 4).
386-399		Not used.
400	CASENM	Dummy input for identification purposes only.

Multiple Case Control

Location 99 is an option control for running multiple cases. If additional cases follow, a value of 1.0 is input. If it is the final case, a -1.0 is input. A special input card format is required for this item which is consistent with the requirements of subroutine LOADER. A -1 is placed in columns 1 and 2, a 99 in columns 5 and 6 and the control value ($\pm$ 1.0) in columns 7 through 18. Following the basic variable input data (loader format), this separate card containing the multiple case control information is inserted. When running in a coupled mode with G400, multiple cases are not allowed.

Linearized Airfoil Characteristics Input

The use of this program requires the input of linearized airfoil characteristics. The more accurate the linearization, the more realistic the linearized solution will be and the faster the nonlinear solution will be obtained. The linearization is made by assuming that at any Mach number the lift coefficient (C_ℓ) can be modeled by assuming that the variation of C_ℓ with angle of attack (α) can be broken into two distinct regions, an unstalled region and a stalled region. In the unstalled region, the C_ℓ distribution is assumed to vary linearly with angle of attack. In the stalled region, there is no variation of lift with angle of attack. Figure 9 is an example of this modeling process for two Mach number conditions for a nonsymmetrical airfoil section. The input parameters required to define this linearization consist of the angle of zero lift (α_o), the linear lift curve slope (C_{ℓ_α}) and the angle of maximum lift (α_{max}). These quantities as applied to the linearization, are noted in Fig. 9.

The input method consists of inputting the linearized quantities as a function of Mach number and thickness distribution. Up to five thickness ratios (t/c) with a total of 14 Mach number varations per thickness are allowed for each linearized quantity (C_{ℓ_α}, α_{max}, α_o). The ordering of the linearized quantities is C_{ℓ_α} distributions first, followed by α_{max} distribution, and finally the α_o distributions. Each of these quantities has the following input ordering:

1. Title card containing the number of thicknesses and a title. (FORTRAN Format: I2, 16A4).

2. The data set for the smallest thickness

 .

 . increasing thickness[*]

n. The data set for the largest thickness

Each data set consists of variations of the particular linearized quantity versus Mach number in the following format:

J, DATA(1) DATA(2) ...., DATA(J)

where J = 2n + 2 when n is the number of Mach number variations.

DATA(1) = n

DATA(2) = The thickness of the particular airfoil section[**]

DATA(3) = First Mach number

DATA(4) = First linearized quantity

.

. | increasing Mach number

.   ↓

DATA(2n + 1) = Last Mach number

DATA(2n + 2) = Last linearized quantity

[FORTRAN format: I2, 10F7.0/(F9.0,9F7.0)]

---

* If only one thickness is required, it is necessary to input a minimum of two thickness data sets of linearized airfoil characteristics which bound the thickness required because of interpolation requirements of the computer program.

**It should be noted that it is not necessary to use the actual thickness values. The thickness can be used as an artificial parameter to obtain variations of airfoil characteristics over the rotor blades.

42

Tabulated Airfoil Data (Optional)

This program will accept two formats (CLOAD and BERQUIST) of tabulated airfoil data (C_ℓ and C_d) as functions of angle of attack, Mach number, and thickness ratio if the Nonlinear Solution Model is requested. The two formats represent various levels of complexity. The Berquist format is restricted to angle of attack and Mach number variations while the CLOAD format is trivariant with thickness ratio included as an additional parameter. Because the program does not explicitly use the Mach number or blade thickness ratio for any use other than the airfoil data lookup, these interpolation parameters can be used for other variations. For example, under certain low speed applications, the rotor may not be sensitive to Mach number variations, and thus the Mach number interpolation parameter can be used to incorporate Reynolds number effects by the appropriate equating of the local blade inflow station Mach number values to the tabulated data of available Reynolds number dependent data. In the same manner, the thickness interpolation parameter (CLOAD format only) can be used if spanwise varying airfoil data is used which varys in some manner other than thickness. The input of an appropriate "thickness" distribution (loc. 366) on the blade and the appropriate values of "thickness" in the tabulated airfoil data will make this possible.

The formats which can be used for the input of lift and drag coefficients are described below. The use of the tabled data is controlled by the nonlinear solution input item, loc. 293. A nonzero value will use the Berquist format if in conjunction with the nonlinear solution, a -1.0 is placed in loc. 366. The program will use the CLOAD format if the first thickness distribution is positive.

CLOAD Formatted Data

This airfoil data format requires a header card with the alphanumeric label CLOAD in card columns 1 through 5. The lift and drag coefficients are tabulated separately. Each table requires a header card with the alphanumeric label CLDATA or CDDATA for lift and drag respectively (card columns 1-6). A title card following this header card is required which may contain alphanumeric information for the users convenience. The complete lift data is always followed by the drag data. The format for the actual lift and drag table consists of an additional header card with the alphanumeric label ALPHA in card columns 3 through 7 and the label CL or CD respectively in card columns 15 and 16. The tabulated data follows in order by increasing thickness and for each thickness by increasing Mach number as functions of angle of attack in degrees. Each set of lift or drag data as a function of angle of attack for a given Mach number and thickness ratio value has the following format. The first card of each set has the information specifying the number of data points, the Mach number value and the thickness ratio value. The label NPTS is in card columns 1 through 4 with the number of data points (lift or drag versus angle of attack pairs) specified as a floating point value

in card columns 8 through 10. The Mach number label MACH occurs in card columns 14 through 17 with the floating point value in card columns 21 through 30. The thickness ratio label THICK follows in card columns 34 through 38 with its floating point value in card columns 41 through 50. The actual data, lift or drag coefficient versus angle of attack are input as paired points (angle of attack and then the data value) in order of increasing angle of attack using an 8F10.x format. Each set of data for a given Mach number and thickness ratio is input in the order noted above, increasing Mach number for each thickness ratio, with each set of thickness data in order of increasing thickness. The lift tables are input first and the drag tables follow. The total set of data is completed by a final card containing alphanumeric label END in card columns 1 to 3. The total amount of data must not exceed 4000 angle of attack values for both lift and drag tables.

Berquist Formatted Data

This format is restricted to only Mach number and angle of attack variations for lift and drag data. The format uses an identical format for both lift and drag information. This lift data set is input first and the drag data set follows. The first card of each data set specifies the integer number of Mach number variations, in card columns 1 and 2, and title information in columns 3 through 80. A maximum of 15 Mach number variations can be used. For each Mach number data set the following format is used. Card columns 1 through 2 of card one of each set contain an integer number whose value is equal to the total number of input items following for this set. The next items on the card is the floating point number of angle of attack, lift coefficient pairs followed by the particular floating point Mach number value. Then the floating point angle of attack and lift coefficients pairs follow. The angle of attack first, followed by the corresponding lift coefficient in order of increasing angle of attack.

The format for this data is I2, 10F7.x for the first card of each set and F9.x, 9F7.x for each subsequent card required to complete the set of data. A maximum of 17 pairs of data are allowed per Mach number. The sets of data are ordered in increasing Mach number.

Blade Element Linearized Airfoil Characteristics Input (Optional)

This input provides an option for using an externally calculated set of airfoil characteristics for each point in the rotor disc rather than by interpolation from the linearized airfoil data. The option is controlled by WOPT in the basic variable input (loc. 207). If the sectional airfoil characteristics option is requested, the angle of attack of the maximum section lift coefficient, the angle of zero lift, and the slope of the lift curve are read in via cards for each blade element station (each combination of azimuth position and interpolated radial blade station, RSNEW). Subroutine UAERO transforms this information to

the inflow stations, RS, for use in the program. This option is useful when the sectional airfoil characteristics are generated by another computer program. In particular, this option provides a means of incorporating unsteady airfoil characteristics generated by another computer program. The required input format of the above data for each azimuthal position is presented below. All input is in exponential format (E12.5) and right adjusted.

Card No.	Column	Description of Input Item
1	1-12	Azimuthal position on the rotor disk which corresponds to the starting azimuth of the rotor.
2	1-12 13-24, 25-26, 37-48, 49-60.	Angle of maximum section lift coefficient (deg), starting at the innermost station 5 to a card for as many stations as are required (RSNEW stations).
3 . . . N		
N+1	1-12 13-24, 25-36, 37-48, 49-60.	Angle of zero lift, (deg) starting at the innermost station, 5 to a card for as many stations as are required.
N+2 . . . N+M		
N+M+1	1-12 13-24, 25-36, 37-48, 49-60.	Slope of the lift curve (C_l/rad) starting at the innermost station, 5 to a card for as many stations as are required.
N+M+2 . . . N+J		

Repeat the sequence starting with card 1 for the next blade azimuth position, until data for all rotor azimuth positions have been input.

External Velocity Field Input (Optional)

As noted in the technical description of this analysis, the ability to include external velocity influences, such as fuselage interference velocities, has been included in the formulation. This external influence is superimposed on the blade segment flow velocities due to rotation, blade motion, and flight speed. The use of this feature is controlled by loc. 217 in the basic variable.

All three components or only the axial component may be used if desired. This option is used (a 1.0 or 2.0 respectively in loc. 218). Data is specified at each RSNEW station in terms of the harmonic coefficients for a positive harmonic series for each component. The data is specified in order of increasing RSNEW value, root to tip. The format for the harmonic coefficients for each station is specified as: The first card contains two input items in card columns 1-3 and 7-16. The first input is I3 format and contains the number of harmonics (maximum of 18 harmonics). The second item is a scaling factor. If it is 0.0 the program will scale the input velocities by the flight speed. Thus the normal method of running will assume that the input velocities are nondimensional if this item is set to zero. Card 2 contains the steady state value in an E15.6 format in card columns 1-15. The cosine components follow on the next card(s) using a 5E14.6 format (columns 1-70). Following the cosine components, the sine components are input on a new card with the same format (5E14.6). If the three components of velocity are requested, the program reads a set of harmonics for each component (axial, tangential, radial) at each station.

The program will read this data and calculate the externally input velocities at the inflow (RS) stations. However, this data must be input at each RSNEW station. One of the resulting outputs of this program in the circulation solution mode is the induced velocity field at the rotor blades. Because one of the intentions of this analysis is to provide the induced velocities at the rotor blade for use in blade airload and response analyses, the externally input velocities are also included in the output induced velocities at the RSNEW stations on punched cards and in the harmonics of induced inflow. These combined velocities are directly transferred to the blade response analysis (G400). This avoids the necessity of reading two sets of velocities into the blade response analysis, but can confuse the uninformed user.

Blade Element Variable Blade Pitch Angle (Optional)

Currently, the analysis will internally calculate the blade element section pitch angles from specified (conventional) blade pitch control and twist distributions. For some applications, it may be desirable to use other formulations for the local blade pitch angles. The option to specify the blade pitch angle at each station has been included as an optional feature (loc. 365). The input is in the form of harmonic coefficients at each interpolation blade station (RSNEW) for a negative fourier series. The input format is identical to the format required for the External Velocity Field input and the description will not be repeated here. The scaling factor (second item) on card 1 should be set to 1.0 for this option. This option can be used to input higher harmonic control inputs to the analysis. It should be noted that this input will be used in addition to the normal input controls, not inplace of these inputs.

Field Point Coordinate Location Data (Optional)

When the field point flow field induced velocity calculation mode is used, the user must specify the coordinates of the desired field point locations. This may be done by two methods described below. All coordinate data is nondimensionized with respect to the rotor radius.

Arbitrary Field Point Input

The first method allows the user to specify via input, up to 250 arbitrary field points in the rotor tip path plane coordinate system. The format requires that the first card specify the mass storage device unit number that the field point coordinates will be read from. The integer unit number for the device is input in free field format (5 for card input). If a zero or negative number is used, the program will expect the input to follow the first card. If it is greater than 0 the program will attempt to read from the unit number specified by this input. The format for the actual data is as follows. The second card or mass storage device contains the number of field points in free field format. The following cards (or mass storage device) will then contain the x, y and z coordinates respectively for each point, three points to a card in an 9F8.0 format. This feature allows for completely arbitrary specification of the field points.

Tail Surface Field Point Input

In order to make the use of the field point induced flow field calculation compatible with the RIEVA program input requirements (Ref. 15). The program includes the capability to input selected data describing a planar tail surface with a minimum of effort. The program will automatically calculate the individual panel coordinates from this input. The first data is the x,y and z

position of a reference point on the tail with respect to the rotor hub, and the tail surface attitude (deg) with respect to the rotor shaft axis, input with a 4F12.6 format respectively. The tail surface data is divided into two sides and is input for the advancing side first, then the retreating side. Each side's input format requirements are the same. The number of panels in the chordwise and spanwise directions for a given side of this tail are input respectively on the next card using a 2I5 format. The cards following this card contain the x, y and z coordinates for each corner of the tail side referenced to the tail reference point, each corner starting a new card. The format is a 3F12.6 format for each corner. The corner sequence is defined as follows. The inboard leading edge corner is input first, followed by the outboard leading edge corner. The outboard trailing edge corner follows with the inboard trailing edge corner input last. The sequence is identical for each side of the tail.

Noninduced Axial Velocity Component Input, W1

The noninduced axial velocity component, W1, equation 6, may be either calculated internally in the program or read in via cards. The option control in the basic variable input is W1OPT. If calculated internally, blade flexibility is neglected and rigid, articulated blade equations are used to include the influence of rotor attitude, pitch angle and flapping motions. W1 values, for each point in the rotor disc, may be input on cards to include blade flexibility effects. This is particularly useful when the W1 values are calculated in another computer program. Appropriate values for articulated or nonarticulated rotors may be used. When including flexibility effects (using input mode), the total noninduced axial velocity component, W1, consists of axial velocity components due to both torsional and bending flexibility in addition to the rigid blade axial velocity. The W1 velocity is defined to be positive for upflow. The W1 values are input for the RSNEW stations and linearly interpolated in the program to the inflow stations, RS. The required input format is presented below.

The data is input six items to a card using a Fortran 6E12.5 format. There is a set of cards for each radial interpolated inflow solution station (RSNEW). The sets are ordered from the innermost to the outermost station. Each set of cards consists of the W1 values at all azimuthal positions for a given radial station. These values must start at the first actual blade azimuthal position (J = 1), and are ordered azimuthally in the direction of rotation; therefore, care must be taken when blade phasing is used to insure the correct order of input. Starting with the first set, a card is filled with 6 items, then a new card is started; if needed. This is continued until all azimuthal positions for a given set are complete. For the next station this process is repeated. There are no title cards or blank cards used. For example, a case with 15 azimuthal positions (DPSI = 24 deg) and 9 blade stations (STNEW = 9) would have a total of 27 cards input for that rotor (3 cards per set x 9 stations).

48

If the analysis is run in a coupled mode to G400, Wl's are transfered from G400 to this analysis. The program option to run the coupled analysis will automatically control this input option.

Forward Flight Generalized Wake Coefficient Data (Optional)

The ability to request a distorted wake geometry based on the UTRC Generalized Wake Model is not included in this version of the analysis. However, for compatibility with other versions of the analysis the description of the planned capability is provided.

The program will internally calculate the wake coefficients and associated tip vortex distortions. However, it is possible to input the coefficients if so desired. This option is requested by loc 89 being set to 1.0. Again, the user is warned of the preliminary and limited nature of this model. The first input card contains the number of harmonics used for the generalized wake shape function and the three coefficients describing the generalized wake envelope function (Ao, Al, SLOPE) in a I3, 3E10.4 format.

The harmonic coefficients for the generalized wake shape functions defining the first and all subsequent revolutions of wake geometry are input as follows. For each set (first revolution and subsequent revolutions) the cards(s) will contain the steady coefficient, followed immediately by the cosine harmonic coefficients, and then directly followed by the sine harmonic coefficients using a 8F10.4 format. The first set of cards will contain the coefficients for the first revolution of wake geometry, and the second set will contain the coefficients used to describe all subsequent revolutions of wake geometry.

Blade Bound Circulation Data (Optional)

When the program is run in the field point flow field mode (loc 224), the circulation distribution must be input to the analysis. The format to do so is similar to that required for the input of the noninduced velocity from Wl. For each inflow station (RS) the circulation distribution around the azimuth is input on cards using a 10F7.0 format. This input is repeated (starting a new card) for each inflow station moving outboard to the tip station. The input format is compatible with the punched output format for the circulation control by loc. 213, except that the peak circulations which are punched out are not used.

An additional feature which allows the user to isolate regions of wake influence is included in this version. This feature is implicity requested when the tail surface location input is requested. This feature allows the user to zero out selected bound circulations which in turn allows for the nulling of the trailing circulations of selected trailing wake filaments.

The input for this option follows the circulation input when the tail surface location option has been requested. Using a free field format, the integer index pair values (I,J) corresponding to the radial and azimuthal positions for the rotor blades on the rotor disk are input, one pair per card, to zero out the bound circulation of that segment. The termination of this input option is obtained by the input of a 0,0 index pair. Thus if this option is not desired, the first (and last) input for this option will be just the 0,0 pair.

Tip Vortex Coordinate Overwrite Input (Optional)

In order to include a more realistic tip vortex geometry than the classical undistorted geometry, a tip vortex coordinate overwrite option is available. This option allows the program operator to input, via cards, the tip filament geometry. This geometry can be obtained from experimental data or analytical results. With this option more realistic near field effects can be obtained in cases where the wake is known to distort significantly from the classical model, and the geometry of the tip vortex is critical to obtaining accurate inflow solutions. The required input format for each rotor is presented below.

The tip filament wake geometry is defined by inputting the vortex segment end point coordinates in the rotor wake reference system, (hub centered, parallel to the tip path plane and aligned with the free stream velocity). Since the wake geometry is assumed to be periodic with blade spacing, the number of rotor positions is just 360/(B·DPSI). In the program this quantity is called BJTOT. Thus, with the wake input for each blade at all of the rotor positions, the total wake geometry for all azimuthal positions is defined. The wake is input in sets of data where each set is associated with a rotor position (L). Each set is then composed of subsets of data corresponding to each blade at a given rotor position. The format is as follows:

For each subset the first card describes the blade number (IB) and the number of vortex segment endpoints to be input for that blade at that rotor position (NPM). These must be two integer, right adjusted numbers described by the Fortran format 2I5 (columns 1-5, 6-10). On the following cards the x, y and z coordinates, in that order, for each endpoint are punched.

The point input sequence is in the following order. The last point in the wake to be overwritten is input first, the next to last next, and so forth until the point corresponding to the blade segment boundary is input, this being the last point to be input for that filament. This allows the computer to retain all the classical wake points past the overwrite. the number of overwrite points for any filament must not exceed the total number associated with the corresponding classical wake. These items are input using a floating

point format, eight spaces to an item, nine items to a card (3 segment end points). When this is completed the following blade subsets (ordered first to last) are punched in the same manner.[*]

To complete a set of data (one rotor position), this grouping of cards is preceeded by a single card describing the particular rotor position this set is for. A fixed point, right adjusted format is used, Fortran format I10 (Columns 1-10). This format is repeated for each rotor position (ordered first to last) to complete the wake overwrite input.

Summary List of Available Options

In order to group together the available options for use in this program the following list is presented. All of the items listed are controlled by input items in the basic variable input. Listed are the available option controls and their locations. Details of their use were described in the input instructions.

---

[*] The rotor blades are ordered as follows. Blade one lies at the first azimuthal position (including phasing) and the following blades (2 through B) are defined to exist azimuthally behind the first blade. For example: with a 3-bladed rotor, if blade one is at zero azimuth, blade two is at 240 deg and blade three is at 120 deg.

OPTION	LOCATION	NAME OF OPTION CONTROL
Interpolated induced velocities	39	STNEW
Couple to G400	88	XLINK
Generalized wake option	89	GWKOP[+]
Lateral wake distortions	92	YDIST[+]
Contour plots	100	PLTCON
Rollup of filaments into tip filament	190	ROLLUP
Different types of wake models	186	TYWAKE
Anhedral tip	191	ANDTIP[*]
Wake truncation	200, 210	TRUNC, TRUNCI
Input of W1 (noninduced inflow)	206	W1OPT[*]
Input for blade element linearized airfoil characteristics	207	WOPT[*]
Geometric coefficient print option	208	OPTPO[*]
Matrix solution print option	209	PRINT[*]
Induced velocity punch option	210	PUNCH[*]
Wake coordinate print option	211	CPOPT
Print option for blade points within vortex core	212	CROPT[*]
Circulation punch option	213	CPUNCH[*]
Input of external axial velocities	217	OPVIBC[*]

[*] Generally not used or required
[+] Not functional in this version

OPTION	LOCATION	NAME OF OPTION CONTROL
Alternate skewed flow formulation	218	SKEWFO*
Include tangential and radial induced velocities in the inflow solution	219	VIROTR*
Intermediate print option	221	DEBUGP
Vortex core option	222	OPCORE
Field point mode option	224	FROPT
Tail surface field point option	225	TAILVI
Harmonic analysis option	272	HARMOP
Nonlinear circulation solution option	293	SOLVE*
Option for automatic relaxation feature	297	AUTORF*
Save GC option (coupled mode)	299	GCSAVE*
Swept blade	363	RSWEPT*
Option for variable blade pitch input	365	OPTHET*
Save circulation solution (coupled mode)	382	SAVGAM
Coarse wake model	385	COURSE*

* Generally not used or required

DESCRIPTION OF PROGRAM OUTPUT

Categories and Sequence of Program Output

The categories of program output which are printed in the following sequence are:

1. Print of Basic Variable Input Data

2. Intermediate Printout (optional)

3. Wake Coordinate Printout (optional)

4. Circulation Matrix Printout (optional)

5. Print of Circulation and Induced Velocity Solutions and Related Parameters

6. Print of Harmonic Coefficients of Induced Velocity at the Rotor Blades

7. Field Point Flow Field Output

In addition, punched card output of the induced velocity and circulation solutions is optional. A detailed description of the output items included in each of the above categories follows below.

Print of Basic Variable Input Data

This output is the same data which is input in the Basic Variable Input section via LOADER format. It is printed with a special output format to facilitate the program operator's ability to quickly change variable items without the necessity of using the program documentation once a basic understanding of the program is obtained. Descriptions of any of the items can be found in the input section of this report. The print output format for each particular item is as follows. The first output quantity is the location of the item in the loader array. The second quantity is the variable name of the item as used in the program. A brief description of the item and its units follows. The last quantity output is the actual value of the item.

Intermediate Printout (Optional)

When running this program the user may or may not be interested in certain quantities related to the solution of the particular case in hand. In order to keep printout to a minimum, the printout of this information has been made optional. The option control is DEBUGP in location 221 of the basic variable input. The first of these quantities are the rotor blade coordinates nondimensionalized by the rotor radius in the cartesian coordinate system and blade flapping angles. The coordinates are listed in tabular form, horizontally as a function of radial station (RS) and vertically as a function of aximuth position (PSI).

The following items are aerodynamic in nature and are local blade element quantities. They use the same output format as noted above, and are output in the following order.

1. In-plane tangential velocity component, UTT (fps).

2. In-plane radial velocity component, URR (fps). Not used internally in this version, set to zero.

3. In-plane skew plane velocity component, USS (fps).

4. Cosine of skew angle, COSGAM. Set to 1.0 in this version.

5. Non-induced axial inflow, UPP (fps).

6. Mach number table, MACH.

7. Airfoil lift curve slope-table AT (C_ℓ/rad).

8. Angle of attack for maximum lift table, ALMAX (deg).

9. Angle of attack for zero lift table, ALO (deg).

10. Section pitch angle, THET (deg).

11. Total axial non-induced velocity component table W1 (fps).

12. The product of the lift curve slope and the inplane velocity component table, ACUTAB (ft^2/sec).

13. The maximum circulation table, based on linearized stall considerations, CIRCM (ft^2/sec).

Wake Coordinate Printout (Optional)

If desired, the coordinates of the vortex filament segment endpoints
are printed for each filament string of each blade at each rotor position.
The option control is CPOPT in location 211 of the basic variable input. The
coordinates listed are nondimensionalized by the rotor radius. For any given
filament, the first point is the value of the blade (i.e., the newest segment
endpoint). The last value listed is the oldest filament segment endpoint in
the wake. All points in between are naturally sequenced in order (i.e., newest
to oldest). The coordinate printout for all filaments is frequently omitted
because of the large printout page requirements, and only the tip filament
coordinates are printed. See CPOPT input description.

Circulation Matrix Printout (Optional)

The formulation of the circulation matrix coefficients was previously
presented in the description associated with Equations 1 through 15. In
summary, the geometric coefficients are defined as the geometric functions in
the Biot-Savart relation used to calculate the velocities induced by the wake
and bound vortex segments at the blade. Since the circulation strengths of all
wake segments within a given trailing vortex filament are assumed periodic with
azimuth position for the forward flight condition, the geometric coefficients
of a group of segments within a filament for the same azimuthal position at the
time of generation may be summed over all revolutions to establish a representa-
tive geometric coefficient for the group (GC). To relate these coefficients to
the respective bound circulations, a difference is computed between the wake
coefficients of the trailing filaments to obtain the geometric coefficients
corresponding to the azimuthally varying bound circulations as related to the
wake structure. To these values are added the effects of the bound circula-
tions (GCB). The resulting quantities, GCD are used to form the coefficients
of the circulation matrix as shown in Equation 14. Within the matrix solving
subroutine, the matrix is normalized by dividing each of these values by the
diagonal element. The option control for printing these circulation matrix
coefficients is PRINT (location 209) in the basic variable input. An option is
also provided by PRINT to output the intermediate circulation solutions at each
iteration. These two options are generally not used because of the lengthy
print requirements. If the circulation matrix coefficients are output, the
matrix row and column is printed for each line of data for reference. The
intermediate circulation solutions are output with its position in the column
vector. If desired, the GCD quantities may be printed. The option control in
the basic variable input is OPTPO (location 208). When this option is used
the output is in the following form. At each inflow point on the rotor disk
the geometric coefficients are listed (10 to a line). They are ordered accord-
ing to radial station for each azimuth position with no indication where the
break from one azimuth position to another is made. This printout is not in a
convenient output form as is the matrix coefficients and is generally not used.

56

Print of Circulation and Induced Velocity
Solutions and Related Parameters

The blade bound circulation and induced velocity solutions are printed for each radial station and azimuth position. These output items are not optional since they are the desired results of the program operation. Following the circulation and induced velocity printout, the blade angle of attack distribution is printed. A stall indicator table is printed next. In this table, a 0.0 or a 99.99 is printed for each radial station and azimuth position. A 0.0 indicates no stall. A 99.99 indicates that the stall limit (ACURVE, see input description) was exceeded and that the circulation at that station was limited to a maximum value in the circulation solution. If used, the interpolated induced velocity distribution is then printed.

Print of Harmonic Coefficients of
Induced Velocity

The induced velocity solutions are harmonically analyzed and the harmonic coefficients, based on a positive Fourier series, are printed. The harmonics are printed for the inflow stations (RS) unless interpolated radial stations (RSNEW) have been requested. This harmonic analysis may be done for blade azimuth increments as small as 5.0 degrees (see loader location 227), even though the actual blade azimuth increment was larger.

Punched Cards of Induced Velocity
and Circulation Solutions (Optional)

An option for punched cards of the three components of induced velocity at each blade radial station and azimuth position is available and controlled by PUNCH (location 210) in the basic variable input. The output is compatible with the input format for induced velocities required by Sikorsky and UTRC blade airload and response analyses (e.g., G400). An option for punched cards of the harmonic coefficients of the three components of induced velocity at each blade radial station is also controlled by PUNCH. An option for punched cards of the circulation solutions is controlled by CPUNCH (location 213) in a format compatible with input requirements of the UTRC Distorted Wake Analysis program.

Contour Plots (Optional)

An option for obtaining contour plots of selected solution qualities is available in the version of the analysis (PLTCON, loc. 100). If this option is requested, contour plots of the blade bound circulation, axial induced velocity and effective angle of attack are included in the printout. These contour plots consist of regions defined by different symbols for different levels of the given quality.

Field Point Flow Field Output

If the analysis is run in the flow field induced velocity mode (FPOPT) the output is reduced to include only information which is pertinent to this mode of operation. The input data from the basic variable data input is output. If the intermediate ouptput option is requested, only the blade segment center data is output. The tail geometry input qualities associated with the internal tail surface location calculations are included if this option is requested (TAILVI). In addition, the actual field point coordinates (three components) are listed in a summary list at this point on the program printout. The wake geometry printout can be requested in this mode of operation. The input blade bound circulations are included in the printout, but include the bound segment zeroing effect when used. The final output for this mode includes the actual time history of the three components of induced velocity at each field point location in the rotor tip path plane coordinate system. If requested, these time histories can be harmonically analyzed (FPHARM) based on the rotor blade spacing azimuthal interval as the reference time period.

SPECIAL INPUT AND OPERATING INSTRUCTIONS

This program will allow the use of a wide range of rotor blade geometries. Variable twist blades (both linear and nonlinear) can be used. Variable chord distributions can be used. No restriction on the number of blades is made. The program assumes solution periodicity with blade spacing. The choice of DPSI and the number of blades is interdependent since DPSI must be such that the number of rotor positions between blades and the number of blades are compatible (360/B/DPSI = integer). The flapping hinge offset must be inboard of the innermost vortex sheet filament.

Concerning the choice of radial stations for the inflow solutions, careful selection of stations is necessary. Since the inflow stations also define the trailing vortex filament spacing, attention must be given during their selection to the wake model that results. Generally the tip region should contain a larger number of filaments per unit length than the inboard region. This allows for a better modeling of the concentration of trailing vorticity in the region of the tip vortex and also gives better resolution of the bound circulation near the tip (this is needed because of the severe loading gradients observed in this region). Standard values which are generally used at UTRC for a 9 station case are .25, .375, .525, .65, .75, .85, .925, .965, and .99 with appropriate changes for the root cutout at the inboard locations for different blade designs. If an anhedral tip is being used, a maximum of five inflow solution stations is allowed outboard of the start of the anhedral tip section.

When modeling wake rollup in forward flight, careful selection of the number of filaments that rollup into a concentrated tip vortex is also necessary. The selection is based on taking the azimuthal average of the radial position of peak circulation on the disk and choosing the number of filaments for rollup to be the total of those which are outboard of this average radial position. Since the locations of the peak circulations are not known beforehand an iteration procedure may be necessary. Generally for the stations mentioned above in forward flight, the 5 outboard filaments are "on the average" outboard of the average radial location of peak circulation.

The blade motion and control parameters should be chosen to be compatible with the wake model used, since these parameters determine to a large degree the rotor thrust, blade loading and other parameters which are directly related to the wake geometry. Generally a separate program is used to compute these parameters for a desired operating condition. With these parameters, other appropriate input, and a wake model, the resulting inflow distribution is computed with this program. This inflow is then used in the program which generated the control parameters values and rerun. If the new control parameters are reasonably close to the initial values, then the wake model and control parameters are compatible. If not, more iterations may be necessary.

Operation of this program is generally straightforward once a basic understanding of the program input is obtained. However, there is one area where problems may occur. For cases in which vortex elements come within extreme close proximity to the blades, large unrealistic geometric coefficients can result. These large values can lead to large off-diagonal terms in the circulation matrix which results in convergence problems in the matrix solution or locally unrealistic solutions. This may generally be avoided by proper selection of the core radius assigned to the vortex filaments. (The velocity induced by the vortex element can be set to zero for blade points within the vortex core radius, loc. 222.) Experimental data indicate tip vortex core radii of approximately one-tenth of the blade chord for helicopter rotors. However, the proper core radius for the inboard filaments representing the vortex sheets is not well defined at this time. Values of one-tenth to three-tenths of the blade chord have generally been used. However, some unrealistic solutions associated with close proximity have resulted in certain instances with the former value.

REFERENCES

1. Landgrebe, A. J., and T.A. Egolf: Rotorcraft Wake Analysis for the Prediction of Induced Velocities. USAAMRDL Technical Report 75-45, Eustis Directorate, USAAMRDL, Fort Eustis, Va., Jan 1976.

2. Bielawa, R. L.: Aeroelastic Analysis for Helicopter Rotors with Blade Appended Pendulum Vibration Absorbers-Mathematical Derivation and Program Users Manual, NASA CR-165886, June 1982.

3. Sopher, R., R. E Studwell, S. Cassarino and S. B. R. Kottapalli: Coupled Rotor/Airframe Vibration Analysis. NASA CR-3582. Contract NAS1-160;58, June 1982

4. Landgrebe, A. J., and T. A. Egolf: Prediction of Helicopter Induced Flow Velocities Using the Rotorcraft Wake Analysis. Presented at the 32nd Annual National V/STOL Forum of the American Helicopter Society, Washington D.C., May 1976, Preprint Number 1002.

5. Landgrebe, A. J.: An Analytical Method for Predicting Rotor Wake Geometry, Jl. American Helicopter Society, Vol 14, No. 4, October 1969.

6. Landgrebe, A.J., and E. D. Bellinger: An Investigation of the Quantitative Applicability of Model Helicopter Rotor Wake Patterns Obtained from a Water Tunnel, USSAMRDL Technical Report 71-69, December 1971.

7. Landgrebe, A. J., and M C. Cheney, Jr.: Rotor Wakes - Key to Performance Prediction, Presented at the Symposium on Status of Testing and Modeling Techniques for V/STOL Aircraft, Mideast Region of the American Helicopter Society, October 1972; also, AGARD Conference Proceedings No. 111, Aerodynamics of Rotary Wings, February 1973.

8. Arcidiacono, P. J., and R. G. Carlson, Helicopter Rotor Loads Prediction. Paper presented at the AGARD Specialists Meeting on Prediction and Verification of Loads on Helicopters, Milan, Italy, April 1973.

9. Landgrebe, A. J., and T. A. Egolf: Prediction of Rotor Wake Induced Flow Velocities Along the Rocket Trajectories of an Army AH-1G Helicopter. Picatinny Arsenal Technical Report 4797, U.S. Army Picatinny Arsenal, Dover, N.J., March 1975.
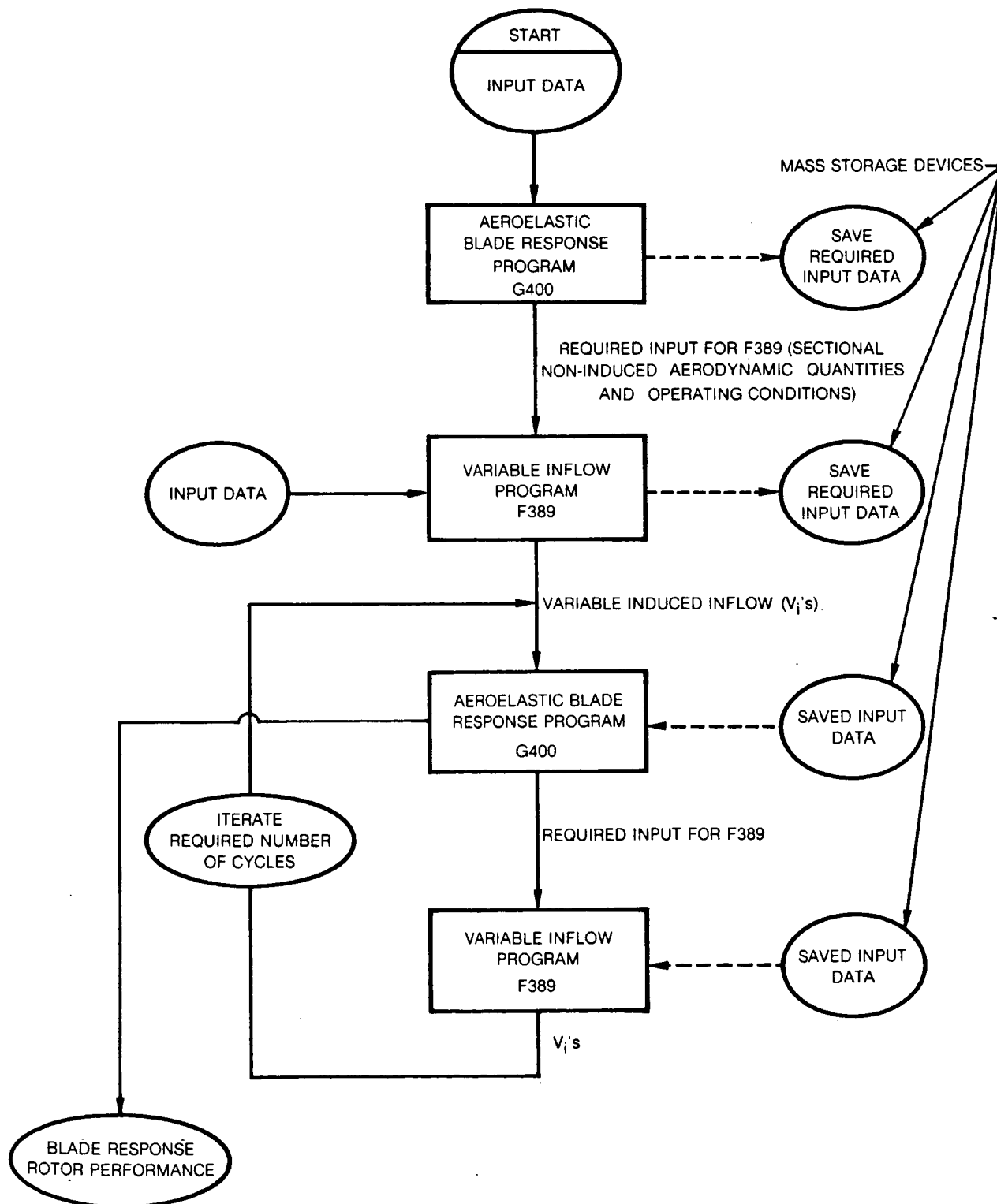
REFERENCES (Cont'd)

10. Landgrebe, A. J., R. B. Taylor, T. A. Egolf, and J. C. Bennett: Helicopter
    Airflow and Wake Characteristics for Low Speed and Hovering Flight from Rocket
    Interference Investigations, Proceedings of the 37th Annual Forum of the
    American Helicopter Society, May 1981.

11. Landgrebe, A. J., R. Moffitt, and O. R. Clark: Aerodynamic Technology for
    Advanced Rotorcraft, Journal of the American Helicopter Society, Part I:
    Vol. 22, No. 2, April 1977; Part II: Vol. 22, No. 3, July 1977.

12. Pizali, R. A., and F. A. Duwaldt: A Method for Computing Rotary Wing Airload
    Distributions in Forward Flight, TREC Technical Report 62-44 (U.S. Army
    TECOM), July 1962.

13. Miller, R. H.: On the Computation of Airloads Acting on Rotor Blades in
    Forward Flight, Journal of the American Helicopter Society, Vol. 7, No. 2,
    April 1962.

14. Gessow, A., and G. C. Myers, Jr.: Aerodynamics of the Helicopter, Macmillan
    Company, New York, 1952.

15. Gangwani, S. T.: A Doublet Lattice Method for the Determination of Rotor
    Induced Empennage Vibration Airloads-Analysis Description and Program
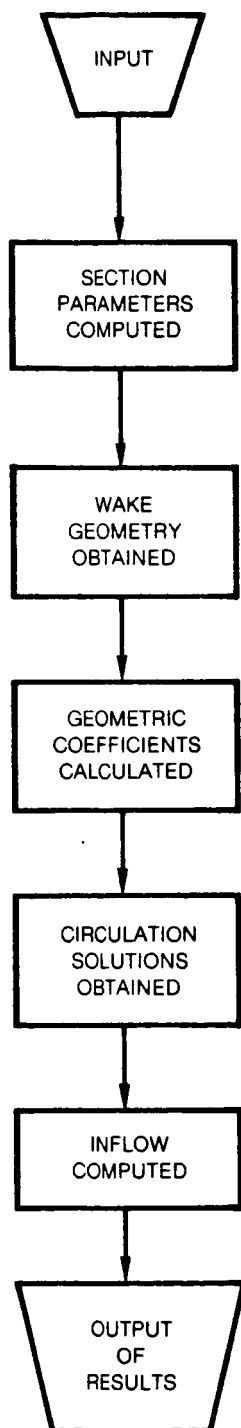   Description, NASA CR-165893, June 1982.

ROTORCRAFT WAKE ANALYSIS (1981)

FIG. 1

63

FIG. 2
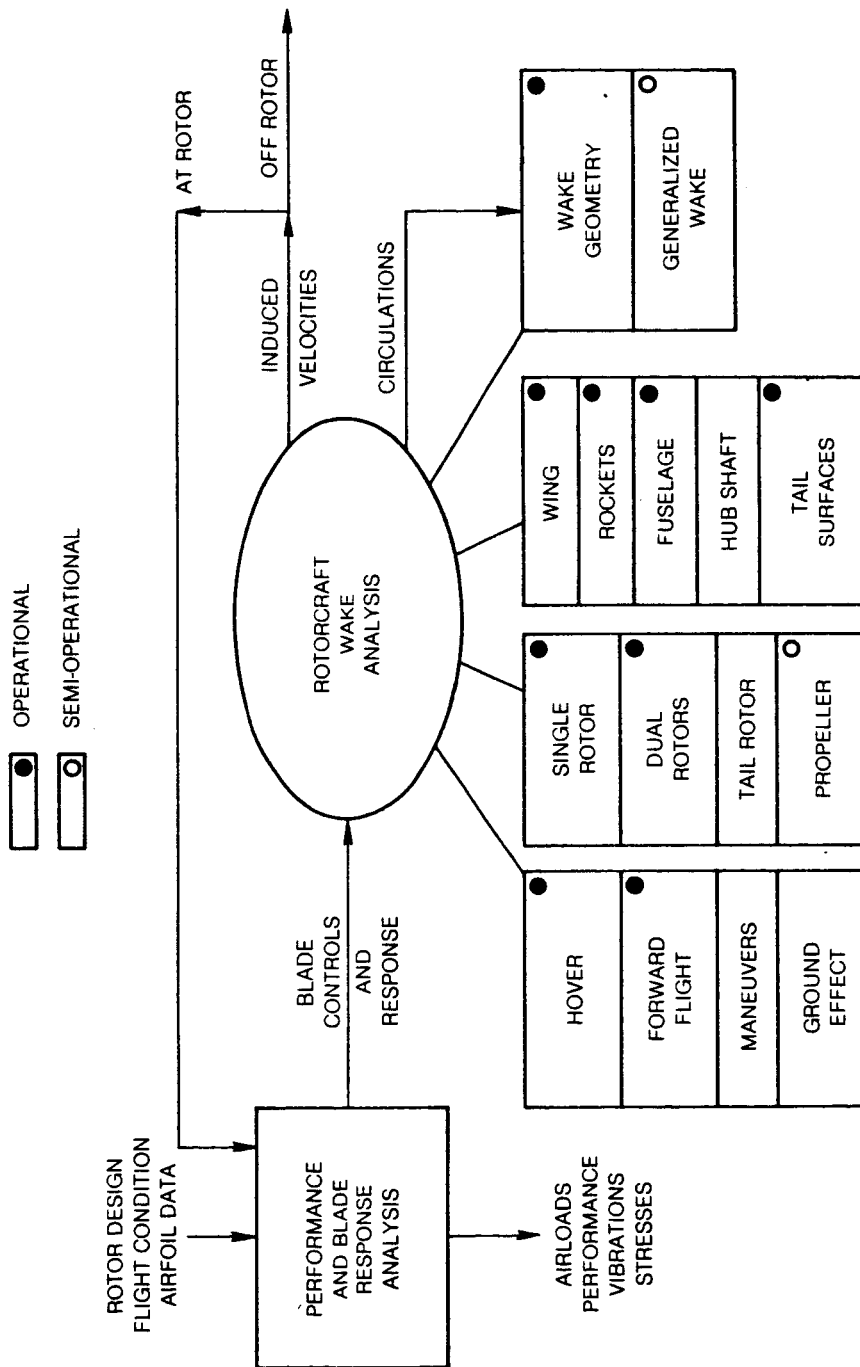
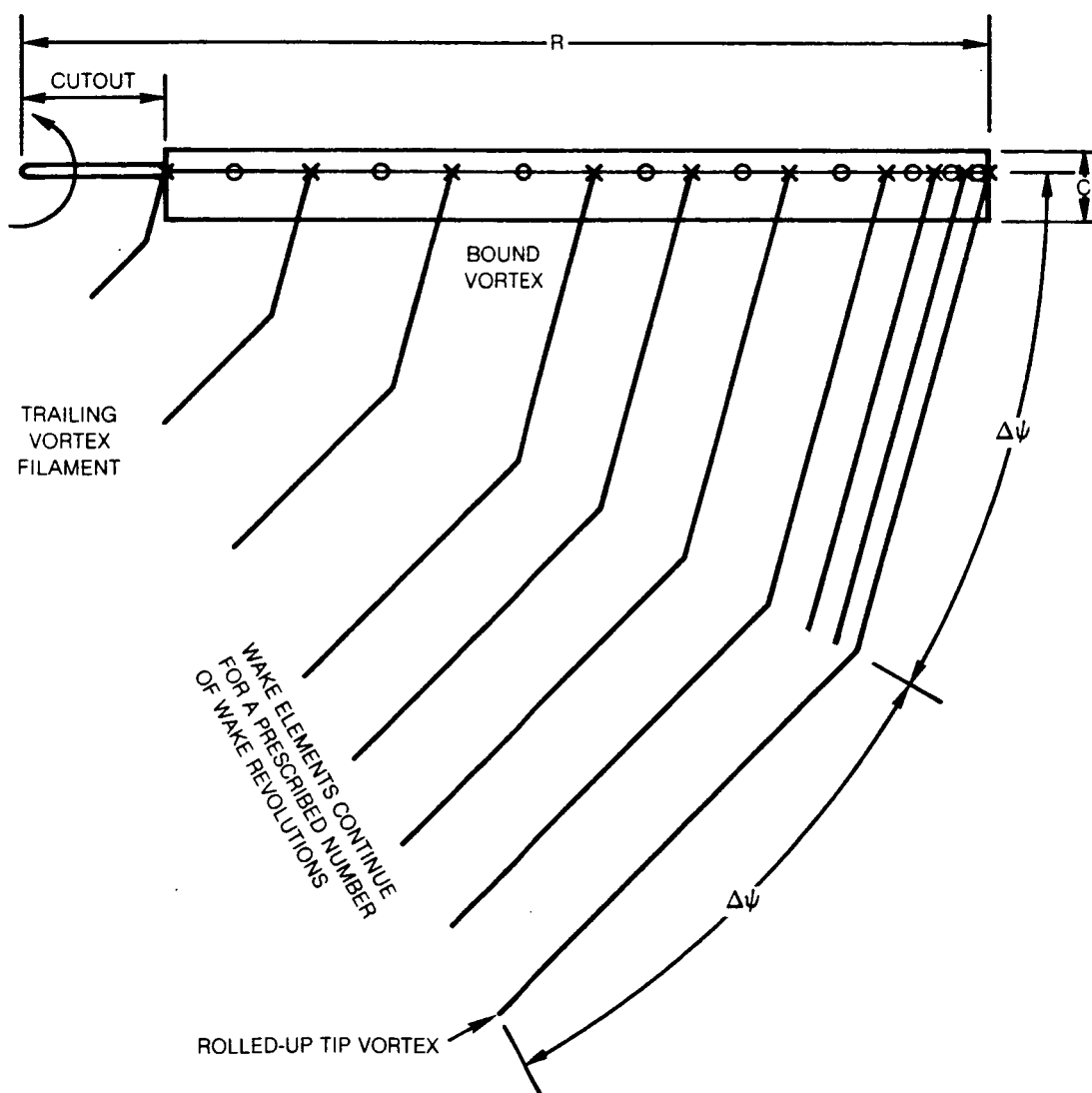**SIMPLIFIED FLOW CHART OF ROTOR INFLOW PROGRAM COUPLED WITH THE BLADE RESPONSE PROGRAM**

FIG. 3

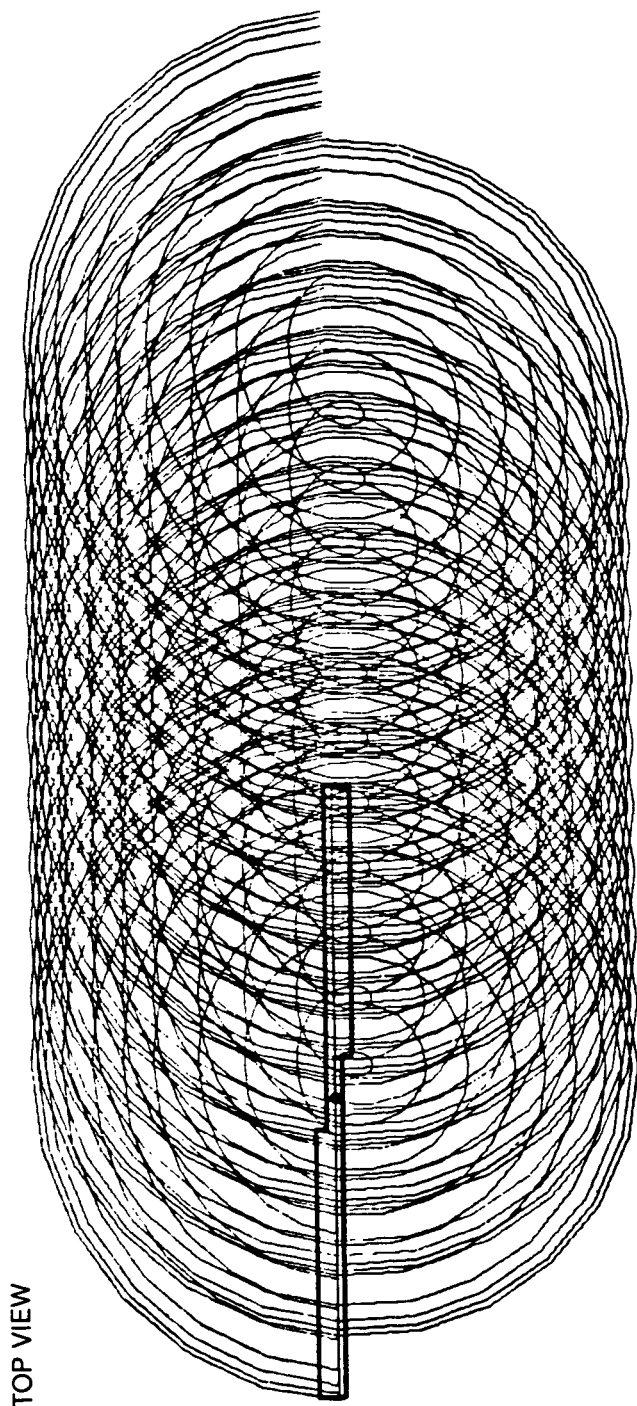BASIC SEQUENCE OF PROGRAM OPERATIONS FOR VARIABLE INFLOW PROGRAM

INPUT

SECTION
PARAMETERS
COMPUTED

WAKE
GEOMETRY
OBTAINED

GEOMETRIC
COEFFICIENTS
CALCULATED

CIRCULATION
SOLUTIONS
OBTAINED

INFLOW
COMPUTED

OUTPUT
OF
RESULTS

65

FIG. 4

REPRESENTATION OF BLADE AND WAKE BY BOUND AND TRAILING VORTEX SEGMENTS

O BLADE SEGMENT CENTERS (INFLOW STATIONS)
X BLADE SEGMENT BOUNDARIES

R

CUTOUT

C

BOUND
VORTEX

TRAILING
VORTEX
FILAMENT

$\Delta \psi$

WAKE ELEMENTS CONTINUE
FOR A PRESCRIBED NUMBER
OF WAKE REVOLUTIONS

$\Delta \psi$

ROLLED-UP TIP VORTEX

FIG. 5

COMPUTER WAKE REPRESENTATION — UNDISTORTED WAKE

TOP VIEW

SIDE VIEW

FIG. 6

PREDICTED TIP VORTEX GEOMETRY USING UTRC WAKE GEOMETRY ANALYSIS
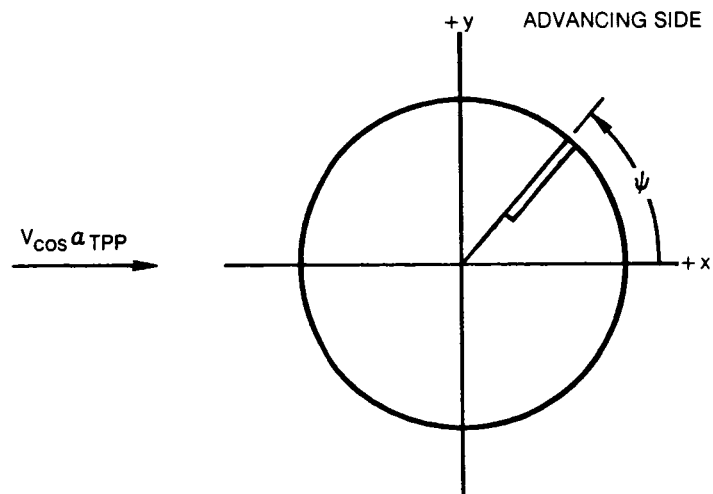
V = 30 knots

TOP

SIDE

FIG. **7**
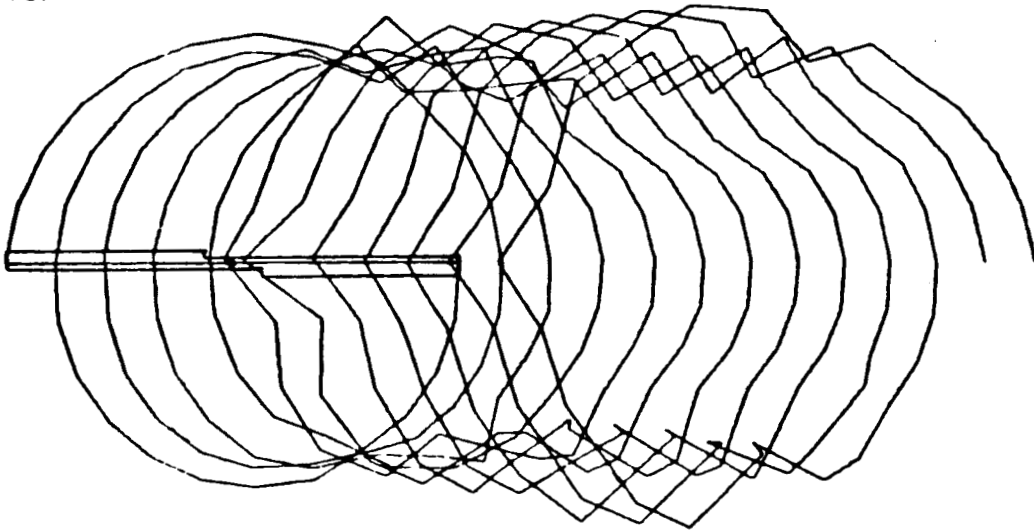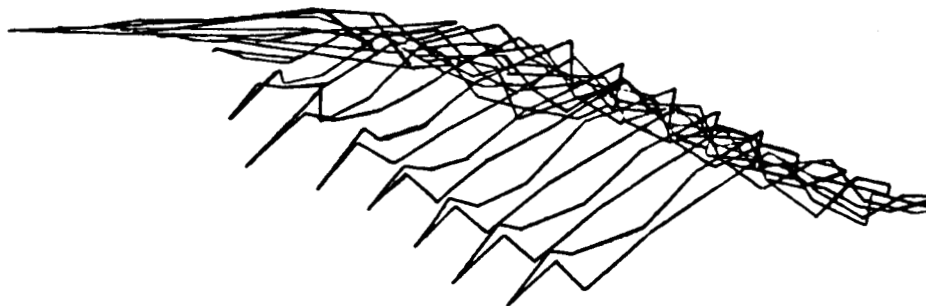
WAKE GEOMETRY COORDINATE SYSTEM

a) SIDE VIEW

b) TOP VIEW
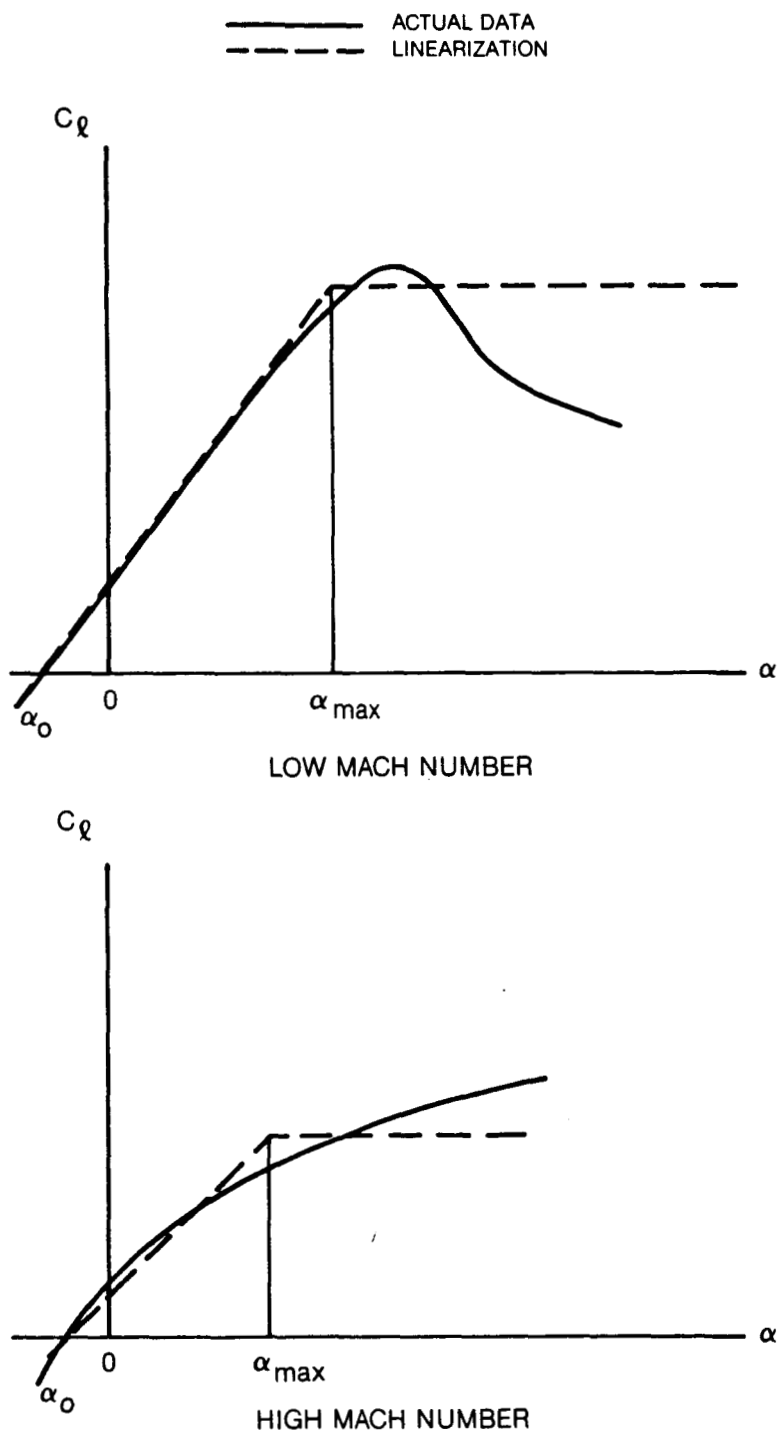
FIG. **8**
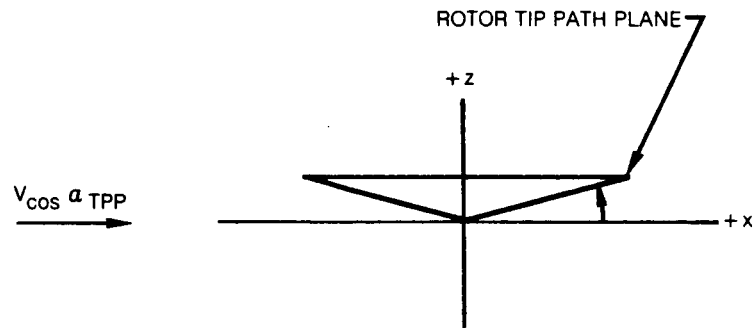
DEFINITION OF BLADE ELEMENT VELOCITIES

SECTION AA

FIG. 9
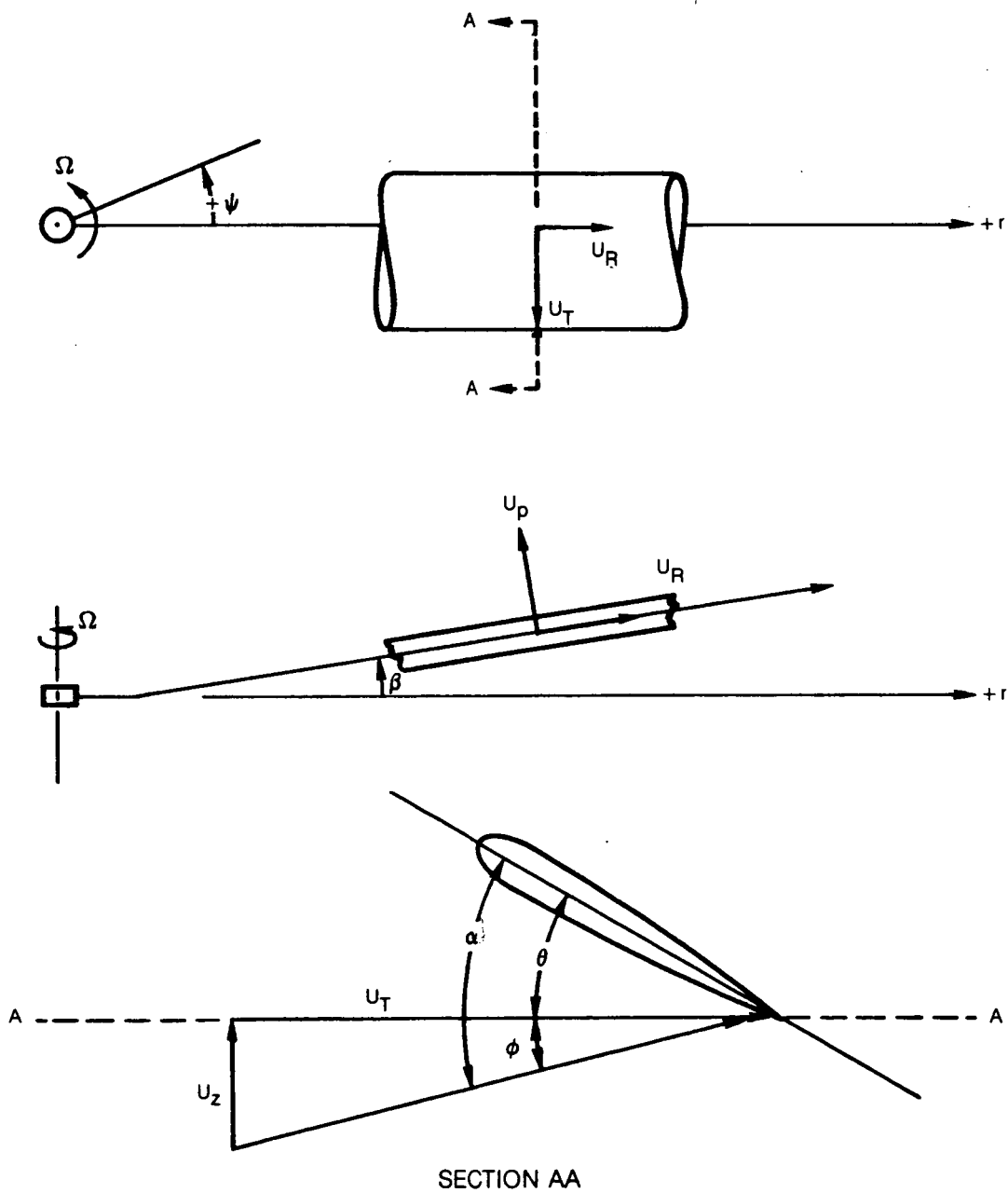
SUBROUTINE CALLING SEQUENCE DIAGRAM

FIG. **10**
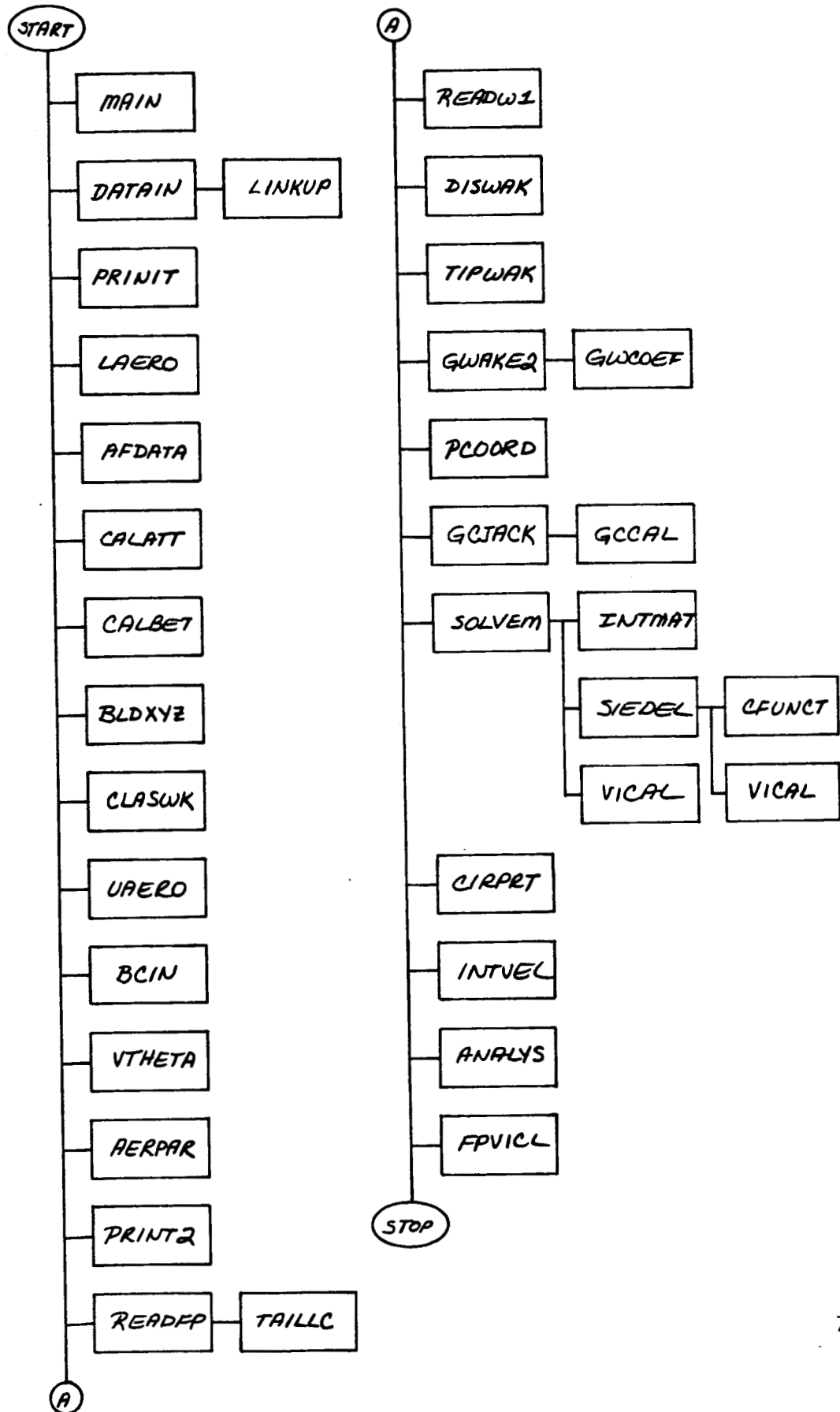
LINEARIZATION OF THE LIFT COEFFICIENT OF A NONSYMMETRIC AIRFOIL SECTION AT TWO MACH NUMBER CONDITIONS

LOW MACH NUMBER

HIGH MACH NUMBER

1. Report No. NASA CR-165894	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle Users Manual for a Prescribed Wake Rotor Inflow and Flow Field Prediction Analysis - Users Manual and Technical Approach		5. Report Date June 1982
		6. Performing Organization Code
7. Author(s) T. A. Egolf and A. J. Landgrebe		8. Performing Organization Report No. UTRC81-2
		10. Work Unit No.
9. Performing Organization Name and Address United Technologies Research Center Silver Lane East Hartford, CT 06108		11. Contract or Grant No. NAS1-16058
		13. Type of Report and Period Covered Contractor Report
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, DC 20546		14. Sponsoring Agency Code

15. Supplementary Notes

16. Abstract

A user's manual is provided which includes the technical approach for the Prescribed Wake Rotor Inflow and Flow Field Prediction Analysis. The analysis is used to provide the rotor wake induced velocities at the rotor blades for use in blade airloads and response analyses and to provide induced velocities at arbitrary field points such as at a tail surface.

This analysis calculates the distribution of rotor wake induced velocities based on a prescribed wake model. Section operating conditions are prescribed from blade motion and controls determined by a separate blade response analysis. The analysis represents each blade by a segmented lifting line, and the rotor wake by discrete segmented trailing vortex filaments. Blade loading and circulation distributions are calculated based on blade element strip theory including the local induced velocity predicted by the numerical integration of the Biot-Savart Law applied to the vortex wake model.

17. Key Words (Suggested by Author(s)) Rotor Wake, Prescribed Wake, Inflow, Induced Flow Field	18. Distribution Statement Unclassified-Unlimited

19. Security Classif. (of this report) Unclassified	20. Security Classif. (of this page) Unclassified	21. No. of Pages 80	22. Price*