NASA TM-87104

NASA Technical Memorandum 87104

NASA-TM-87104 19860003827

A FORTRAN Computer Code for Calculating Flows in Multiple-Blade-Element Cascades

Eric R. McFarland
Lewis Research Center
Cleveland, Ohio

November 1985

LIBRARY COPY
DEC 6 1985
LANGLEY RESEARCH CENTER
LIBRARY, NASA
HAMPTON, VIRGINIA
NF01115

NASA

A FORTRAN COMPUTER CODE FOR CALCULATING FLOWS IN MULTIPLE-BLADE-ELEMENT CASCADES

by

Eric R. McFarland
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135

SUMMARY

A solution technique has been developed for solving the multiple-blade-element, surface-of-revolution, blade-to-blade flow problem in turbomachinery. The calculation solves approximate flow equations which include the effects of compressibility, radius change, blade-row rotation, and variable stream sheet thickness. An integral equation solution (i.e., panel method) is used to solve the equations. A description of the computer code and computer code input is given in this report.

INTRODUCTION

A computer code has been written to analyze flow on turbomachinery blade-to-blade surfaces. The code has the capability of analyzing blade rows which are made up of several different blade shapes and/or spacings. This capability makes the code useful for calculating flows in centrifugal machinery with splitter blades or in mis-tuned blade rows, where the blade spacing varies. The number of varying blade shapes that can be used in a blade-row design is not limited by the code. However, the amount of computer storage available does impose a practical limit on any problem. The code being described here is dimensioned to handle from one to four bodies in cascade. After redimensioning, this same code has been used to calculate a flow problem with as many as 15 bodies.

The basic method used to solve the flow field was described in reference 1. However, the method of reference 1 was extended in the code reported here to include multiple-body cascades. Briefly, the solution method solves approximate governing equations for the blade-to-blade, steady-state flow problem in turbomachinery. The effects of compressibility, radius change, blade-row rotation, and variable stream sheet thickness are included. The working fluid is assumed to be inviscid, irrotational, and a perfect gas. Comparisons with other solution techniques and experimental data is given in reference 1. Overall, the solution produces good results for subsonic flow, but the accuracy decreases for high subsonic and supersonic flows.

CODE DESCRIPTION

General

The multiple-body-panel code is written primarily in FORTRAN IV, and uses double precision arithmetic. Storage for the four-body version of the code,

N86-13295

which accompanies this report, is approximately 650 000 words when run in double precision. Typical run times for the code on the IBM 370/3033 computer at NASA Lewis is 25 CPU sec. A slightly different version of the code which runs on the CRAY 1S computer at NASA Lewis requires less than 5 CPU sec/run.

The computer code accompanying this report is essentially the version of the code that runs on the CRAY computer. However, coding that is CRAY computer dependent and allows vectorization of the calculation has been replaced with scalar coding which is machine independent. Also, calls to the Society of Industrial and Applied Mathematics (SIAM) matrix solvers contained in their LINPACK software package have been replaced by calls to an in-house generated solver which is included with the code. The result of these code changes is a code which is slower running than the version of the code used at NASA Lewis, but is more machine and system independent. If the user has access to SIAM's LINPACK software, instructions on how to impliment it into the code are given in the comment statements of subroutine PLMSOL.

Some machine dependency was left in the code. The code is set up with the coding for variable storage arrays and dimensions separated from the main body of the code. On both the IBM and CRAY computer systems at NASA Lewis, this coding is inserted into the main code by the compiler. This compiling option allows the user to adjust the storage requirements of the code and to increase (or decrease) the number of bodies which can be considered. The option is available on many computing systems, and so the code's structure was left in this form. If the user's system does not have this capability, he can physically dimension the arrays and insert the coding into the program before compiling.

Code Structure

The code on the tape as given to COSMIC consists of several short pieces of coding followed by the main code and its accompanying subroutines. The short pieces of coding contain the variable array and dimensioning information. These segments are inserted into the main coding during compilation. This coding appears first on the tape. Each segment is preceded by a line that reads "*CDK" followed by the name of the segment. Five of these segments are used in the code. Each segment is inserted into the program wherever the coding "*CALL" followed by the name of the segment appears. The main program and subroutines are given next on the tape. A line reading "*DK" followed by the name of the routine precedes each subroutine and the main program. The multiple body panel code consists of the main program plus 24 subroutines.

The multiple body panel code is written in a modular style. No calculations or input/output takes place in the main program. The main program only serves as a calling routine. Each subroutine then performs a particular function in the overall solution. The various parts of the code are listed below along with a brief description of the coding's function. Items are listed in the order that they should appear on the tape.

Variable Arrays and Dimension Information Coding (Code to be Inserted into the Main Program Before Compiling)

BASBLK - Contains all geometry and flow solution variable common blocks. Also contains parameter statements which establishes the size of the arrays.

DIMEN - Sets values of specific array dimensions which are used in read/write statements and are passed to variable dimension subroutines.

REG2 - Contains arrays to be used in the calculation of velocities in the flow field (off-body velocity calculations).

REG3 - Contains bulk storage arrays used to restore some of the two-dimensional arrays of BASBLK as one-dimensional arrays for calculations. Also contains arrays used for temporary storage of intermediate results.

REG5 - Contains temporary storage arrays for use in subroutine PLMSOL.

Main Program and Subroutine Coding

MLTPNL - Main program acts as a calling routine for subroutines.

PLMIPT - Reads basic input data file and writes labeled echo check. Called by MLTPNL.

BLADE - Generates blade shapes by patching together spline curves (input option IBLD = 1). Called by PLMIPT.

PLMTRF - Performs the geometric transformation from a surface of revolution to a plane cascade. Called by MLTPNL.

ALPOCL - Calculates the trailing-edge-station flow angle from the downstream flow determined during the Kutta condition iteration. Called by PLMSOL.

PLMFRT - Partially transforms the solution velocity back to physical velocity. Called by PLMSOL.

GEOOPT - Outputs the transformed and physical geometry of the bodies to both the output listing file and the plot file. Called by MLTPNL.

CMPOPT - Completes transformation of the solution velocity back to physical velocity. Makes additional compressibilty corrections (input option ISL = 1). Outputs final blade-surface flow solution to both output listing and plot files. Called by PLMSOL.

RBINT - Calculates the one-dimensional, compressible flow used in approximate governing equations. Also performs integrations to get contributions of the compressibility, blade-row rotation, and stream sheet thickness variation to the flow solution. Called by PLMSOL.

PLMSOL - Sets up the system of equations to be solved. Acts as a secondary calling routine which brings together all the subroutines and information used to solve the matrix of equations for the flow. Called by MLTPNL.

PLMGEO - Calculates geometric parameters and variables used in the solution. Called by MLTPNL.

KCGEOM - Calculates the points downstream of the trailing edges and the flow angles to be used in the Kutta condition (input option ICSCD = 0, 2, or 3). Called by PLMGEO.

VORTEX - Calculates influence of a linear vortex singularity distribution. Called by ABCOEF and OFFBOD.

ABCOEF - Acts as a secondary calling routine to call the subroutines which calculate the influence coefficients of the integral-equation solution. Called by MLTPNL.

JUMP - Calculates the influence coefficients for a discontinuous change in a vortex singularity distribution which is due to discontinuous changes in the slope of the surface of a body. Called by ABCOEF and OFFBOD.

SOURCE - Calculates influence coefficients of a linear-source singularity distribution. Called by ABCOEF and OFFBOD.

VELCAL - Calculates the velocity influence coefficient due to source and vortex distributions for isolated bodies. Called by ABCOEF and OFFBOD.

CASCAD - Calculates the velocity influence coefficients due to source and vortex distributions for a cascade of bodies. Called by ABCOEF and OFFBOD.

BNDCND - Calculates the far-stream boundary conditions from the input flow conditions at the blade leading- and trailing-edge stations. Called by PLMSOL.

POTOPT - Outputs the transformed velocities and singularity strength distribution. Used primarily for solution debugging. Called by PLMSOL.

SPLINT - Calculates a spline curve fit and interpolation from a given array of data points. Assumes the second derivative of the curve goes to zero at the end points. Also has an entry point SPLENT for multiple interpolations along the same curve. Called by BLADE, PLMGEO, PLMTRF, PLMSOL, BNDCND, RBINT, and OFFBOD.

SPINSL - Calculates a spline curve fit and interpolation from a given array of data points. Uses the given slope at the curve ends to calculate the spline curve fit. Called by BLADE.

OFFBOD - Reads coordinates of points that the user has specified. Calculates the flow velocity and direction at those points. Writes out body velocity information to both the output listing and plot files. Called by CMPOPT.

CHLSKY - Solves a set of linear equations by upper and lower triangular matrix decomposition. Has an entry point called CHGCNT for repeat solutions of the same system of equations with a different right hand side. Called by PLMSOL.

CFCALC - Calculates the tangential and normal force coefficients for each body. Also calculates lift, drag, and moment coefficients. Outputs results to the output listing file. Called by CMPOPT.

Input, Output, and Storage Files

The multiple-body-panel code is reasonably straightforward to set up and run. The input data file is column formated. All numerical data is either 8F10.0 for floating point or 16I5 for fixed point data. Floating and fixed point data are not mixed on any input line. Details of the input data file are given in a separate section of this report. Input is read in from unit 05, and printed output is sent to unit 06. In addition, an unformated binary output data set can be sent to unit 60. This binary data set contains geometric and flow solution information. It is intended for use in any graphics code that the user may develop. A description of the information in this "graphics" file will be given later. In order to reduce the in-core storage needed to run the code, extensive use is made of mass storage devices. This requires that units 10 to 20 be assigned when running the code. These units are used strictly as scratch storage and should be released upon completion of the calculation.

The use of mass storage (units 10 to 20) is a trade-off of run time for computer storage. If the user wants a faster running code and has a large-memory computer system available, he may want to eliminate the out-of-core reads and writes. The experience of the author is that this increases the required in-core storage by about a factor of five for the four-body problem (from 0.65 to 4 M words).

The conversion of the code to run using only in-core storage is not a simple task. First, the user must locate all the initial writes to units 10 to 20. These are located in subroutines ABCOEF, OFFBOD, and PLMSOL. Second, the user must define two-dimensional arrays to store the values that had previously been written out and read in from out-of-core storage. Third, the user must substitute the new variables into the solution equations with the proper indices. Lastly, the user must remove all read and write statements to units 10 to 20.

Code-Generated Diagnostic Messages

Several messages can appear in the printed output from the multiple-body code. The name of the subroutine generating the message is appended to the end of the message. The messages are listed together along with a discussion of what occurs in the calculation when the message appears.

1. THE AVERAGE PANEL IS SMALLER THAN THE ADJACENT L. E. OR T. E. PANELS - NS OR NE IS LESS THAN ZERO. THE PERCENTAGE OF POINTS IN THE LEADING EDGE HAS BEEN INCREASED. PLE IS NOW - BLADE

The blade shape generator was set up assuming that the elements used to describe the leading and trailing edges would be smaller than the elements used to describe the blade surface. If the user specifies input parameters that violate this condition, the code will increase the number of elements used to describe the leading edge until the leading-edge elements are smaller than the surface elements. Everytime the message is printed, 5 percent of the total number of elements to be used in describing the blade are added to the leading-edge elements. Increasing the number of leading-edge points does not always allow the solution to continue. If the trailing-edge radius is large compared to the leading edge and has panels that are larger than the average blade surface panel, then the blade shape generator will fail to give a distribution of panel elements that will allow the calculation to proceed. In these cases, the user will have to provide the code with a descrete point description of the input body.

2. THE DOWNSTREAM FLOW CALCULATION IN SUBROUTINE ALPOCL IS IN ERROR - PLMSOL

This error message can appear in solutions using the Kutta condition option. It indicates that the downstream flow angle, calculated using the blade-row circulation, exceeds the maximum possible turning angle for the flow. The downstream angle is set to the maximum value so that the calculation may proceed.

3. KUTTA CONDITION HAS FAILED TO FIND A SOLUTION IN 100 ITERATIONS. ERROR IN THE INLET FLOW ANGLE IS _ RADIANS - PLMSOL

The Kutta condition iteration (ICSCD = 0, 2, or 3 in the input) has failed to converge to the given input flow. The calculation will continue using the flow field at the 100th iteration. If the error is large, the calculated results will be invalid.

4. ITERATION FOR (UPSTREAM/OUTLET/DOWNSTREAM) FLOW CONDITIONS HAS FAILED TO CONVERGE. ERROR IS - BNDCND

Using the given inlet flow conditions, the solution calculates a mass flow for the blade passage. Flow conditions at the upstream, downstream, and outlet blade-row stations are then calculated. If the iteration procedure involved in solving the equations does not converge in 100 iterations, the above message is printed with the appropriate station noted, and the calculation is allowed to continue.

5. SPECIFIED INLET FLOW EXCEEDS CHOKING MASS FLOW - BNDCND

If the given inlet flow conditions define a flow that exceeds the choking mass flow, the above message is printed. The calculation continues but the solution is invalid.

6. SPECIFIED OUTLET FLOW EXCEEDS CHOKING FLOW - BNDCND

7. GIVEN OUTLET FLOW ANGLE EXCEEDS MAXIMUM ALLOWABLE - IT HAS BEEN LOWERED TO - BNDCND

The above two messages are printed when the outlet flow angle produces a flow that is greater then the maximum flow possible. The flow angle is adjusted to produce the maximum flow at the exit, and the calculation continues. The messages can be disregarded for Kutta condition calculations, since the final outlet flow conditions are determined later in the calculation.

8. MAXIMUM OR NEGATIVE VELOCITY ENCOUNTERED AT (UPSTREAM/DOWNSTREAM) BOUNDARY - (UPSTREAM/DOWNSTREAM) RADIUS TOO SMALL ? - BNDCND

This message indicates that the critical velocity ratio has exceeded the maximum value at either of the far-stream stations as noted. This can occur in radial machinery if the far-stream boundary condition is taken at too small a radius. For rotors, the error can also occur if the radius is too large. The solution continues, but it is invalid. The user can avoid the error by moving the boundary station (MRSP(1) or MRSP(NMN) in the input) in closer to the blade row.

9. AVERAGE MASS FLOW CALCULATION EXCEEDS CHOKING FLOW AT M = . CALCULATED AVERAGE MASS FLOW IS TIMES THE CHOKING MASS FLOW. THE FLOW TURNING ANGLE HAS BEEN REDUCED TO ALLOW THE CHOKING MASS FLOW TO PASS - RBINT

This message can occur for calculations using the blade-passage geometry and one-dimensional continuity to calculate the mean flow estimates. The calculation is allowed to continue, but the mean-flow turning angle is set to the maximum turning value instead of using the average blade angle.

10. AVERAGE MASS FLOW HAS FAILED TO CONVERGE. THE REMAINING ERROR IS . - RBINT

This message can occur for calculations of the mean flow estimate from the one-dimensional continuity and the blade-passage geometry. It indicates that the iteration procedure used to calculate the velocity and density has failed to converge in 100 iterations. The calculation is allowed to continue without further iteration.

11. (UPSTREAM/INLET/OUTLET/DOWNSTREAM) VALUE OF THE W/WCR ARRAY IS INCONSISTENT WITH VELOCITY DIAGRAM - RBINT

This message can appear in calculations when the user has given the critical velocity ratio distribution through the blade row. It warns the user that at the location noted in the error message the given mean-flow critical velocity distribution differs from the critical-velocity-ratio values calculated by the code at the boundary-condition stations by more than 0.01. The

calculation continues. The given critical-velocity-ratio distribution is used to estimate the flow compressibility, and the calculated values are used as the boundary conditions for the solution.

12. MASS FLOW CALCULATION AT THE THROAT EXCEEDS THE CHOKING VALUE - RBINT

This message only appears for a single body in cascade problem when the user has set IMN = 2 or 3. The mean flow is set equal to the choking value at the code-calculated throat and the calculation continues.

INPUT FILE DESCRIPTION

The input data file for the multiple-body-panel code consists of two parts. The basic input data describes the problem geometry and flow conditions. If only the basic input data is given, the flow velocity is calculated only on the body surfaces. The second part of the input file is blocks of data which contain coordinates of points in the flow field where the user wants the flow velocity and direction to be calculated. This off-body velocity-calculation input-data file is added to the basic input-data file when velocities in the flow field are needed.

Basic Input File

The basic input-data file is described in this section. The actual form of the input data varies according to the input options selected by the user.

Item 1 - Data set title - up to 80 characters. (Format 20A4)

Item 2 - Calculation control parameters. (Format 16I5)

NBODY NCS ICSCD IMN NMN ISL

NBODY - Number of separate bodies or blade shapes which make up a cascade group. Minimum number 1, maximum number 4. Code can be redimensioned to allow more or fewer bodies.

NCS - Number of flow variation cases to be run. Maximum number 8.

ICSCD - Type of calculation to be made.

ICSCD = 0, Isolated multi-element airfoil. Kutta condition used to set circulation for each element.

ICSCD = 1, Multi-element cascade solution outlet flow angle given. Circulation split for each body element is specified in Item 7.

ICSCD = 2, Multi-element cascade solution Kutta condition used to split the circulation between the cascade body elements. Trailing-edge flow angle taken to be the trailing-edge angle bisector.

ICSCD = 3, Same as ICSCD = 2, except the user specifies in Item 7 the trailing-edge flow angle to be used in the Kutta condition.

IMN - Type of flow path description and mean flow calculation.

IMN = 0, Radius and stream sheet thickness are given as functions of the meridional coordinate. Mean flow is calculated using given blade geometry. Only subsonic flows (with local supersonic pockets) can be calculated.

IMN = 1, In addition to radius and stream sheet thickness, the mean critical velocity ratio is also given as a function of meridional coordinates. All flow ranges, including supersonic flows, can be calculated.

IMN = 2, For single body in cascade only (NBODY = 1, ICSCD = 1). The radii and stream sheet thicknesses are given. Program finds the blade passage throat and calculates mean velocity at the throat. If the inlet critical velocity ratio is greater than the outlet critical velocity ratio, a linear distribution is calculated from the inlet to the throat, and the blade geometry is used to calculate the mean flow from the throat to the outlet. If the inlet critical velocity ratio is less than the outlet critical velocity ratio, the blade geometry is used to calculate the mean flow from the inlet to the throat, and a linear mean flow distribution is used from the throat to the outlet. If inlet and outlet critical velocity ratios are approximately equal, a linear mean flow distribution is assumed from the inlet to the throat and from the throat to the exit.

IMN = 3, Does the same thing as IMN = 2, except the outlet flow is assumed to be supersonic, and a sonic flow is assumed at the throat.

NMN - Number of values used to describe the flow path. Minimum number 2, maximum number 50.

ISL - Compressibility correction.

ISL = 0, Physical velocity is calculated assuming no density variations in the tangential direction.

ISL = 1, A correction is made in calculating the physical velocity from the transformed velocity. This correction is used to account for large variations in density in the tangential direction and gives better results for some problems where supercritical flow is encountered.

Item 3 - Absolute flow conditions at the cascade inlet station, which is taken to be the leading edge of the farthest upstream body. Any consistent set of units may be used. (Format 8F10.0)

GAMMA AR PTAI TTAI

GAMMA - Ratio of specific heats.

AR - Gas constant.

PTAI - Total pressure at the inlet station, measured in the absolute reference frame.

TTAI - Total temperature at the inlet station, measured in the absolute reference frame.

Item 4 - Inlet relative velocity array. Units should be consistent with contents of Item 3. NCS values are to be given. Values are to be taken at the blade-row-inlet (leading-edge) station. (Format 8F10.0)

WIN(1) . . . WIN(NCS)

WIN - Inlet relative velocity array.

Item 5 - Inlet relative flow angle array. Given in degrees. NCS values are to be given at the blade-row-inlet (leading-edge) station. (Format 8F10.0)

ALPIN(1) . . . ALPIN(NCS)

ALPIN - Inlet flow angle array.

Item 6 - Outlet relative flow angle array. Given in degrees. NCS values are to be given at the blade-row-outlet (trailing-edge) station if ICSCD = 1. If ICSCD = 0, 2, or 3, a dummy line or card must be included in the data set. (Format 8F10.0)

ALPOUT(1) . . . ALPOUT(NCS)

ALPOUT - Outlet flow angle array.

Item 7 - Circulation control array. NBODY values to be given. If ICSCD = 1, the fraction of total blade-row circulation to be applied to each element is given. If ICSCD = 3, the trailing-edge flow angle to be used in the Kutta condition is given in degrees. If ICSCD = 0 or 2, a dummy or blank line is given. (Format 8F10.0)

CRCLTN(1) . . . CRCLTN(NBODY)

CRCLTN - Circulation control values for each blade element.

Item 8 - Blade-row rotation-rate array. Given in revolutions per minute (rpm). NCS values to be given. Must be given even if the rotation rate is zero. (Format 8F10.0)

OMEGA(1) . . . OMEGA(NCS)

OMEGA - Angular velocity array.

Item 9 - Flow path description. Radius and stream sheet thickness arrays as a function of the meridional coordinate for IMN = 0, 2, and 3. Radius stream sheet thickness and mean-flow critical-velocity-ratio arrays as functions of the meridional coordinate for IMN = 1. Minimum number 2, maximum number 50. (Format 8F10.0)

9a. MRSP(1) . . . MRSP(NMN)

MRSP - Array of meridional coordinates where flow path description values are to be given. NMN values are to be given. The far-stream boundary-condition stations are taken at MRSP(1) and MRSP(NMN).

9b. RMSP(1) . . . RMSP(NMN)

RMSP - Array of flow path transverse radii measured between the turbomachine centerline and the blade-to-blade surface of revolution at the corresponding MRSP array locations. NMN values to be given.

9c. BESP(1) . . . BESP(NMN)

BESP - Array of stream sheet thicknesses corresponding to MRSP array locations. Values may be normalized by any arbitrary value. NMN values to be given.

```
9d. WMSP(1,1)      . . .      WMSP(NMN,1)
       .           . . .         .
       .           . . .         .
       .           . . .         .
    WMSP(1,NCS)    . . .      WMSP(NMN,NCS)
```

WMSP - Arrays of mean critical velocity ratio corresponding to the MRSP locations and the the flow case being analyzed. Given when IMN = 1. NMN * NCS values to be given.

Body/Blade Shape Input - Items 10 to 12 are read for each body or blade element in the flow. The sequence is repeated NBODY times.

Item 10 - Body-shape input controls. (Format 16I5)

NELM IBLD ITE NBRK

NELM - Number of panel elements (number of body points minus one). If IBLD and ITE = 1, two panels are added to the value of NELM. Maximum number 98 or when IBLD and ITE = 1, maximum number 96.

IBLD - Type-of-blade description.

IBLD = 0, Descrete points are used to describe the body.

IBLD = 1, Blade shape generator used to develop blade coordinates.

ITE - Type of trailing edge.

ITE = 0, Sharp trailing edge.

ITE = 1, Rounded trailing edge.

NBRK - Number of surface slope discontinuities. Maximum number 2. A surface slope discontinuity occurs where the body shape makes an abrupt change in direction (e.g., the sharp trailing edge of the airfoil in fig. 1).

Item 11 - Index of body points where surface slope discontinuities occur. The index of a body point is the number it is given when counting the points, starting at the body trailing edge and moving around the body in a clockwise direction. In figure 1, IBRK(1) = 1, indicating that a surface slope discontinuity occurs at the first element end point (i.e., the trailing edge), and IBRK(2) = K, indicating that a surface slope discontinuity also occurs at the kth end point (i.e., the leading edge in the figure). If NBRK = 0, a dummy or blank line must be included. If IBLD = 1 (see Item 12, Option 2 below) and the blade has a sharp leading edge, the index value for the leading edge is calculated and need not be specified. For blade shapes, always run with NBRK = 1 and IBRK(1) = 1, even for rounded trailing edges. This seems to give the best results. (Format 16I5)

IBRK(1) . . . IBRK(NBRK)

IBRK - Array of body point indices where slope discontinuities occur.

Item 12 - Blade geometry input. Option 1 (IBLD = 0). Discrete blade surface coordinates are given by the user. The coordinates must be given starting at the lower-surface trailing edge and moving around the surface in a clockwise direction. See figure 1. NELM plus one points are to be given with the first and last points being coincident. (Format 8F10.0)

12a. M(1) . . . M(NELM + 1)

M - Array of blade-surface meridional coordinates.

12b. PHI(1) . . . PHI(NELM + 1)

PHI - Array of blade-surface tangential coordinates.

Item 12 - Blade geometry input. Option 2 (IBLD = 1). Blade shape generator will be used to develop blade surface coordinates. The blade generator uses spline curve fits of the user-supplied blade-surface coordinates to develop the upper and lower blade surfaces. These surfaces are then joined to form a sharp leading or trailing edge, or are patched together using either a circle or an ellipse to make a blunt leading or trailing edge. The blade surface is then divided into surface elements by the generator according to the user's specifications. See figures 2 and 3. (Format 8F10.0)

12a. MLE PHILE RA RB TAU

MLE,PHILE - Coordinates of the leading edge for a sharp leading edge. Coordinates of the center of the leading edge circle or ellipse for a rounded leading edge.

RA,RB - Semi-major and semi-minor axes for an elliptical leading edge. RA = RB for a circular leading edge. RA = RB = 0 for a sharp leading edge. RA and RB are determined from a drawing of the leading edge on an M, R * PHI plot. See figure 2.

TAU - Angle, in degrees, of the orientation of the major axis of the ellipse with respect to the m axis on an M, R * PHI plot. See figure 2. TAU = 0 for circular and sharp leading edges.

12b. MTE PHITE RTE

MTE,PHITE - Coordinates of the trailing edge for a sharp trailing edge. Coordinates of the center of the trailing edge circle for a rounded trailing edge.

RTE - Trailing edge circle radius as found on an M, R * PHI plot. See figure 2. RTE = 0 for a sharp trailing edge. For blunt trailing edges, two elements are located on the trailing edge circle. These elements are then added to the number of elements given in Item 10 to bring the total element count to NELM + 2. For sharp trailing edges, the size of the first trailing-edge element is set equal to the size of the leading-edge element.

12c. PLE PUPS FLE FTE

PLE - Fraction of total blade elements (NELM in Item 10) to be used in the leading-edge description for a rounded leading edge. Size of the leading-edge element, as fraction of chord, for a sharp leading edge. See figure 3.

PUPS - Fraction of total blade elements minus the leading-edge elements to be used in the upper surface description. See figure 3.

FLE - Element multiplication factor for transition of element sizes from leading edge to upper and lower surface elements. Value should be taken between 1 and 2. See figure 3.

FTE - Element multiplication factor for transition of element sizes from trailing edge to upper and lower surface elements. Value should be taken between 1 and 2. See figure 3.

12d. BTUPS BTUPE SPLUP

BTUPS - Upper-surface tangency angle with leading edge. Given in degrees. Angle to be taken from an M, R * PHI plot. See figure 2.

BTUPE - Upper-surface tangency angle with trailing edge. Given in degrees. Angle is to be taken from an M, R * PHI plot. See figure 2.

SPLUP - Floating point number of spline points to be given on the upper surface. Minimum number 2, maximum number 50.

12e. M(1) . . . M(SPLUP)

M - Array of upper-surface meridional coordinates to be used in spline curve fit.

12f. PHI(1) . . . PHI(SPLUP)

PHI - Array of upper-surface tangential coordinates to be used in spline curve fit.

Note 1: The first and last spline points are calculated by the code so that these values need not be supplied by the user. However, blank spaces for these values are required in the spline-point coordinate arrays.

12g. BTLWS BTLWE SPLLW

BTLWS - Lower-surface tangency angle with leading edge. Given in degrees. Angle to be taken from an M, R * PHI plot. See figure 2.

BTLWE - Lower-surface tangency angle with trailing edge. Given in degrees. Angle to be taken from an M, R * PHI plot. See figure 2.

SPLLW - Floating point number of spline points to be given on lower surface. Minimum number 2, maximum number 50.

12h. M(1) . . . M(SPLLW)

M - Array of lower-surface meridional coordinates to be used in spline curve fit.

12i. PHI(1) . . . PHI(SPLLW)

PHI - Array of lower-surface tangential coordinates to be used in spline curve fit.

Note 2: The first and last spline points are calculated by the code so that these values need not be supplied by the user. However, blank spaces for these values are required in the spline-point coordinate arrays.

Items 10 to 12 are repeated for each additional body or blade shape until NBODY shapes have been described.

Item 13 - Cascade configuration parameters. See figure 4. (Format 8F10.0)

REFLGT CHORD STGR S

REFLGT - Reference length is used to scale the meridional coordinate in the solution output listing (i.e., the meridional coordinate is divided by REFLGT in the output listing).

CHORD - Chord length.

STGR - Cascade stagger is measured as the difference between the PHI location of the farthest upstream leading edge and the farthest downstream trailing edge. Given in radians. See figure 4.

S - Cascade pitch. Given in radians. The tangential spacing between a common point on the input body geometry and the repeat of the input geometry in cascade. See figure 4.

Item 14 - Output control. (Format 16I5)

IOUT IPLOT

IOUT - Control for printed output.

IOUT = 0, Gives echo check, error messages, and compressible flow results.

IOUT = 1, Blade geometry is added to the printout.

IOUT = 2, Auxiliary flow output is added to the printout.

IOUT = 3, Transformed solution is added to the printout. This information is primarily used to debug a solution.

IPLOT - Control for unformated graphical file output.

IPLOT = 0, No graphics file is generated.

IPLOT = 1, Graphics file is generated and sent to unit 60.

Basic Input Notes

Items 1 to 14 given in the previous section make up the basic input file. This data set supplies the information which is needed to calculate the flow velocity on the surfaces of the bodies in the flow. It represents the minimum input required to run the code. The following notes are intended to give the user additional information on how to use the code input to set up special flow cases, and to indicate to the user some possible problems that may occur during a solution:

(1) When the Kutta condition option (ICSCD = 2 or 3) is used to determine the blade row circulation and downstream flow angle, an iteration procedure is employed. This iteration procedure has difficulty converging when the downstream relative conditions are near sonic, and will fail to converge when the downstream flow conditions are supersonic. If the iteration count exceeds 101, the code stops iterating and continues on with the calculation, using the values of the solution variables at that point. The user should, therefore, always check the iteration count to make sure he has a valid solution. The iteration count is printed in the compressible-flow output when the Kutta condition option is used.

(2) Plane cascade flows may be calculated by setting all the elements in the mean flow path radius array (RMSP) to 1.0, and then considering the M and PHI coordinates used to describe the blade shapes to be Cartesian coordinates.

(3) Incompressible flow calculations are set up using input flow conditions (Item 3). The parameters PTAI and TTAI are still set to the absolute inlet total pressure and temperature, but the ratio of specific heats is used as a calculation control switch and is set to a value greater than 1000. The remaining parameter in input Item 3 (the gas constant AR) is set so that the fluid density is equal to the expression PTAI/(AR * TTAI). All other input items remain unchanged. For incompressible calculations, IOUT in input Item 14

should be set equal to 2. This will cause the dimensional velocity and pressure to be printed out. These values are more useful than the standard compressible-flow output for incompressible solutions.

(4) The user may simulate viscous loss and blockage effects by reducing the stream sheet thickness array (BESP) in the mean flow path description. The BESP array must be smoothly reduced from the blade-row inlet to exit by a percentage equal to the amount of total pressure loss expected across the blade row. The flow velocities calculated will reflect the effect of the viscous blockage; however, the pressures listed by the solution will be in error.

Field-Point Input File

If the user wants to calculate flow velocities in the flow field (i.e., off the body surface), flow-field input data is added to the basic input file described previously. The coordinates of the points where the velocity is to be calculated are given along with reference flow angles at the points. The off-body input is grouped into blocks of data. The user may add as many of these off-body data blocks as needed. The format of the data blocks is given as follows:

Item 1ob - Number of off-body (flow-field) point coordinates to be given in data block. Maximum number 98. (Format 16I5)

Item 2ob to Nob - Off-body point data. Nob equals the number given in Item 1ob plus one. (Format 8F10.0)

MOB PHIOB ANG

MOB - Meridional coordinate of the off-body point where the flow-field velocity is to be calculated.

PHIOB - Tangential coordinate of the off-body point where the flow-field velocity is to be calculated. (Given in radians.)

ANG - Reference angle. Given in degrees. The velocity is calculated along and normal to a line through the off-body point. The reference angle orients this line with respect to the meridional direction. If the reference angle (ANG) was aligned with the flow, then the calculated normal velocity would be zero. If ANG = 0, the flow angle will be calculated with respect to the meridional direction.

Items 1ob to Nob comprise one block of off-body input data. The user may add as many of these blocks as he wants by simply repeating Items 1ob through Nob for each additional block. The number of off-body points can vary from block to block; however, the number of points given in any one block can not exceed 98. No separate echo check is made of the off-body input data, but the data is printed out with the output listing of the calculated off-body velocities.

Off-Body Input Notes

Repeated blocks of items lob to Nob make up the off-body calculation input file. Information given in this data set locates the points in the flow field where the user wishes to calculate the flow velocity. The following notes are intended to help the user in making these off-body velocity calculations:

(1) For multiple-flow-case runs (NCS > 1), only the last block of off-body points are stored for solutions 2 through NCS.

(2) A singularity in the off-body velocity calculations occurs when an off-body point is located at a distance of n*pitch above or below a control point located on an input body (n = +1,+2, ..., -1,-2, ...). As the off-body points approach this location, errors will develop in the velocity calculation. These errors can be avoided by restricting the off-body points to a region which is within a tangential location of plus or minus 0.9 * pitch of the input body location as shown in figure 5.

Sample Input

A sample input data set for a three-body problem (two cylinders in a channel) is shown in Table I. Labels for the various sections of the input have been inserted between the input data lines. The input lines that make up the actual data set have been appended with item numbers which correspond with the descriptions given previously.

OUTPUT FILE DESCRIPTION

The output that is generated by the multiple-body-panel code is controlled by Item 14 in the basic input file. Two separate files may be generated. These are the listing or printed output file of the tabulated flow solution, and an unformatted flow solution file, which is intended for use with a post-processing graphics program.

Output Listing File

The code output listing file is sent to unit 06, which is the line printer on most computer systems. It is a formatted data set. If the maximum amount of output is requested by setting IOUT = 3 in the basic input file (Item 14), the following information appears on the listing in the order given below:

(1) Labelled echo check of basic input file.

(2) Transformed and physical blade shape geometry plus any blade shape generator code diagnostics. Listed for each body.

(3) Code-generated solution diagnostics.

(4) Transformed velocity solution and singularity strengths for each body.

(5) Compressible-flow-solution listing and auxilary-flow-solution output for each body.

(6) Off-body (flow-field) velocity-calculation results.

For multiple-flow-case runs (NCS > 1, Item 1 in basic input file), Items 3 to 6 are repeated for each additional flow case. The solution generates the output listing at various stages in the calculation. This allows the user to receive echo check and blade geometry even though the flow solution fails.

Graphics File

Solution information, which can be used in a user-generated graphics code, is sent to unit 60 by the code. The graphics file is generated if IPLOT = 1 in Item 14 of the basic input file. It is an unformatted data set. The file contains the following information in the order given below:

(1) The number of bodies in the calculation (NBODY in Item 1 basic input file), the number of flow cases in the run (NCS in Item 1 basic input file), the number of the body with the largest meridional coordinate (bodies are numbered according to the order of their input in the input file), the number of the body with the smallest meridional coordinate, the mean-flow-calculation control parameter (IMN in Item 1 basic input file), the number of points used to describe the mean flow (NMN in Item 1 basic input file), cascade pitch (S in Item 13 basic input file), and the calculation reference length (REFLGT in Item 13 basic input file).

(2) The number of elements used to describe each body (NELM in Item 10 basic input file).

(3) Index of body coordinate with the minimum meridional value for each body. Index refers to the the location of the point in the coordinate array as sent to this file.

(4) Index of body coordinate with the maximum meridional value for each body. Index refers to the the location of the point in the coordinate array as sent to this file.

(5) The meridional coordinate array of the body control points where the surface velocities are calculated.

(6) The meridional coordinate array of the panel/element end points that make up the body.

(7) The tangential coordinate array in radians of the body control points where the surface velocities are calculated.

(8) The tangential coordinate array in radians of the panel/element end points that make up the body.

(9) Local transverse radius array at the panel end points.

(10) The panel end point transformed meridional coordinate array.

Items (5) through (10) are repeated for each body as they are ordered in the input.

(11) Meridional coordinate array for mean flow data (MRSP in Item 9 basic input file).

(12) Transverse radius array at mean-flow meridional coordinates (RMSP in Item 9 basic input file).

(13) Stream sheet thickness array at mean-flow meridional coordinates (BESP in Item 9 basic input file).

(14) Mean-flow critical velocity ratio array at the mean flow meridional coordinates (Either calculated or given in Item 9 basic input file).

(15) Index of the body control-point coordinate which marks the stagnation point on the body. Index refers to the the location of the point in the body coordinate array as sent to this file. If this index is ILU, then control points ILU to NELM lie on one side of the stagnation point while control points ILU - 1 to 1 lie on the other side.

(16) Distance along the body surface of the control points from the leading-edge stagnation point.

(17) Local relative critical velocity ratio at the body control points.

(18) Ratio of local static pressure to local relative total pressure at the body control points.

(19) Pressure coefficient at the body control points. Reference dynamic head is based on the upstream-station relative velocity and density.

(20) Local relative Mach number at the body control points.

(21) Relative static pressure at body control points.

(22) Relative velocity at body control points.

Items (15) through (22) are repeated for each body.

(23) Upstream and downstream relative velocity of the mean flow.

(24) Number of off-body points where the flow field velocity is to be calculated (Item 1ob in field point input file).

(25) The transformed meridional coordinate of the off-body point, the tangential coordinate for the off-body point, the relative velocity at the off-body point, and the flow angle in radians with respect to the input reference angle.

For the first flow case, Items (24) and (25) are repeated for all the blocks of off-body points given in the field-point input file. For all remaining flow cases, only the last block of off-body points are written out. (See field-point input file description for more information.)

(26) The integer value "9999" is written to mark the end of one flow case and the start of data for the next.

Items (14) to (26) are repeated for each flow case.

The graphics file is generated by write statements in three subroutines. Items (1) to (13) are written out in the subroutine GEOOPT. Items (14) to (23) are written out in subroutine CMPOPT. Items (24) to (26) are written out in subroutine OFFBOD. If the user plans to use this file, he should look at these write statements in order to get a better idea of how to go about reading in the information in the file.

TEST CASES

Three sample test-case input data sets accompany the code. The first case is an isolated, two-element airfoil. The calculated surface-velocity distribution and geometry are given in figure 6, along with an exact solution by Williams (ref. 2). The second case is a centrifugal compressor impeller with splitter blade. Figure 4 shows the problem geometry, and figure 7 shows the calculated surface velocity. A flow solution from the finite difference code TANSON (ref. 3) is also given on the figure. The final test case is a three-body problem of two cylinders in a channel. The off-body-velocity calculation capability of the code was used to generate the vector plot of the solution flow field shown in figure 8.

REFERENCES

1. McFarland, E.R.: A Rapid Blade-to-blade Solution for Use in Turbomachinery Design. J. Eng. Gas Turbines Power, vol. 106, no. 2, Apr. 1984, pp. 376-382.

2. Williams, B.R.. An Exact Test Case for Plane Potentential Flow About Two Adjacent Lifting Areofoils. R. & M. No. 3717, British A.R.C., 1973.

3. Katsanis, T.; and McNally, W.D.: FORTRAN Program for Calculating Velocities and Streamlines on a Blade-to-blade Stream Surface of a Tandem Blade Turbomachine. NASA TN D-5044, 1969.

TABLE I. - SAMPLE INPUT FILE. (ONLY LINES APPENDED WITH AN ITEM NUMBER ARE INCLUDED IN THE INPUT.)

```
BASIC INPUT FILE

INPUT DATA SET TITLE

      INCOMPRESSIBLE FLOW THROUGH CYLINDERS - DENSITY = 2                    ITEM 1

CALCULATION CONTROL PARAMETERS

        3    1    1    0    2    0                                          ITEM 2

FLOW CONDITIONS

     1000.     2.        2000.     500.                                     ITEM 3
     10.                                                                     ITEM 4
     0.                                                                      ITEM 5
     0.                                                                      ITEM 6
     0.        0.5       0.5                                                ITEM 7
     0.                                                                      ITEM 8

MEAN FLOW PATH DESCRIPTION

     -5.       7.                                                            ITEM 9A
     1.        1.                                                                 9B
     1.        1.                                                                 9C

FIRST BODY GEOMETRY DESCRIPTION (OPTION 2) - CHANNEL WALL (FLAT PLATE)

        60    1    1    1                                                    ITEM 10
         1                                                                   ITEM 11
     -3.       -1.       0.01      0.01      0.                             ITEM 12A
     5.        -1.       0.01                                                     12B
     .05       .5        1.75      1.5                                            12C
     0.        0.        2.                                                       12D
     0.        0.                                                                 12E
     0.        0.                                                                 12F
     0.        0.        2.                                                       12G
     0.        0.                                                                 12H
     0.        0.                                                                 12I

SECOND BODY GEOMETRY DESCRIPTION (OPTION 1) - CYLINDER

        40    0    0    0                                                    ITEM 10
     (BLANK/DUMMY LINE)                                                      ITEM 11
       0.500000  0.493840  0.475530  0.445500  0.404510  0.353550  0.293890  0.227000   ITEM 12A
       0.154510  0.078220  0.000000 -0.078210 -0.154500 -0.226990 -0.293890 -0.353550        12A
      -0.404500 -0.445500 -0.475520 -0.493840 -0.500000 -0.493850 -0.475530 -0.445510        12A
      -0.404520 -0.353570 -0.293910 -0.227010 -0.154530 -0.078240 -0.000020  0.078190        12A
       0.154490  0.226970  0.293870  0.353530  0.404490  0.445490  0.475520  0.493840        12A
       0.500000                                                                              12A
       0.000000 -0.078220 -0.154510 -0.227000 -0.293890 -0.353550 -0.404510 -0.445500   ITEM 12B
      -0.475530 -0.493840 -0.500000 -0.493840 -0.475530 -0.445510 -0.404510 -0.353560        12B
```

```
-0.293900 -0.227000 -0.154520 -0.078230 -0.000010  0.078200  0.154490  0.226980        12B
 0.293880  0.353540  0.404500  0.445490  0.475520  0.493840  0.500000  0.493850        12B
 0.475540  0.445510  0.404520  0.353570  0.293910  0.227020  0.154540  0.078250        12B
 0.000000                                                                              12B
```

THIRD BODY GEOMETRY DESCRIPTION (OPTION 1) - CYLINDER

```
   40    0    0    0                                                                  ITEM 10
(BLANK/DUMMY LINE)                                                                    ITEM 11
 2.500000  2.493843  2.475528  2.445502  2.404509  2.353553  2.293893  2.226995     ITEM 12A
 2.154510  2.078219  2.000004  1.921786  1.845496  1.773009  1.706113  1.646452          12A
 1.595496  1.554501  1.524474  1.506158  1.500000  1.506153  1.524467  1.554489          12A
 1.595482  1.646434  1.706093  1.772987  1.845471  1.921762  1.999977  2.078195          12A
 2.154486  2.226973  2.293872  2.353535  2.404491  2.445490  2.475518  2.493838          12A
 2.500000                                                                                12A
 1.000000  0.921783  0.845492  0.773005  0.706107  0.646447  0.595492  0.554497     ITEM 12B
 0.524472  0.506156  0.500000  0.506155  0.524470  0.554494  0.595487  0.646441          12B
 0.706100  0.772996  0.845481  0.921771  0.999987  1.078203  1.154494  1.226980          12B
 1.293879  1.353540  1.404497  1.445495  1.475521  1.493840  1.500000  1.493848          12B
 1.475535  1.445514  1.404524  1.353572  1.293915  1.227020  1.154535  1.078246          12B
 1.000000                                                                                12B
```

CASCADE CONFIGURATION

```
1.        1.        0.        3.                                                      ITEM 13
```

OUTPUT CONTROL

```
   2    1                                                                             ITEM 14
```

FIELD POINT INPUT FILE - BLOCK ONE

```
   5                                                                                  ITEM 1 OB
-2.       -.5       0.                                                                ITEM 2 OB
-2.       0.        0.                                                                ITEM 3 OB
-2.       .5        0.                                                                ITEM 4 OB
-2.       1.        0.                                                                ITEM 5 OB
-2.       1.5       0.                                                                ITEM 6 OB
```

BLOCK TWO

```
   3                                                                                  ITEM 1 OB
-1.       -.75      0.                                                                ITEM 2 OB
-1.       -.5       0.                                                                ITEM 3 OB
-1.       -.25      0.                                                                ITEM 4 OB
```

END OF INPUT DATA FILE

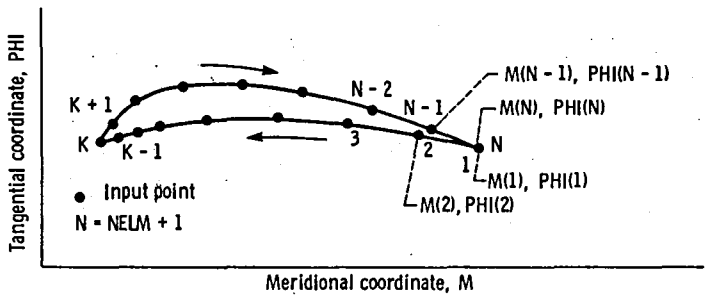

Figure 1. - Blade shape input, option 1, IBLD = 0.

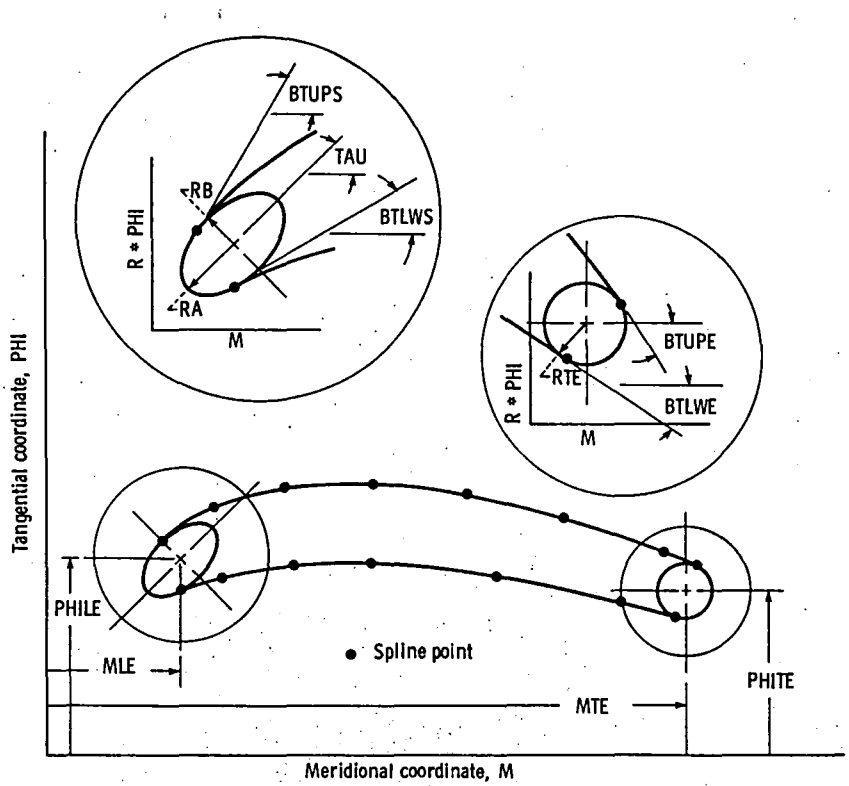

Figure 2. - Blade shape input, option 2, IBLD = 1.

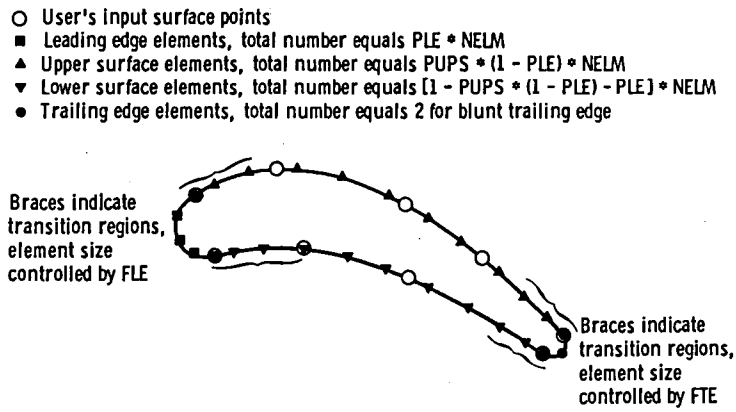

Figure 3. - Output from blade generator, option 2, IBLD = 1.

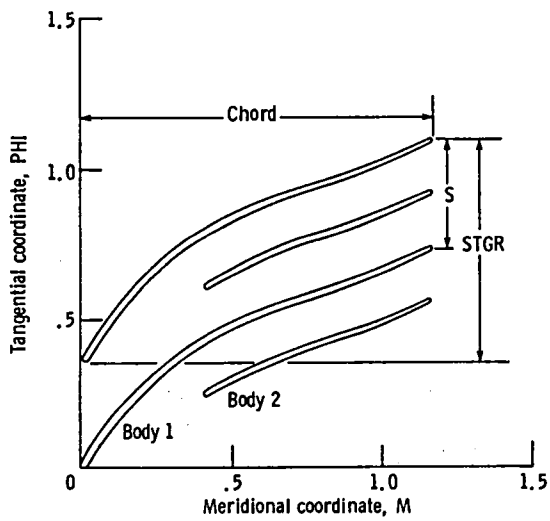

Figure 4. - Blade-row geometry input parameters.

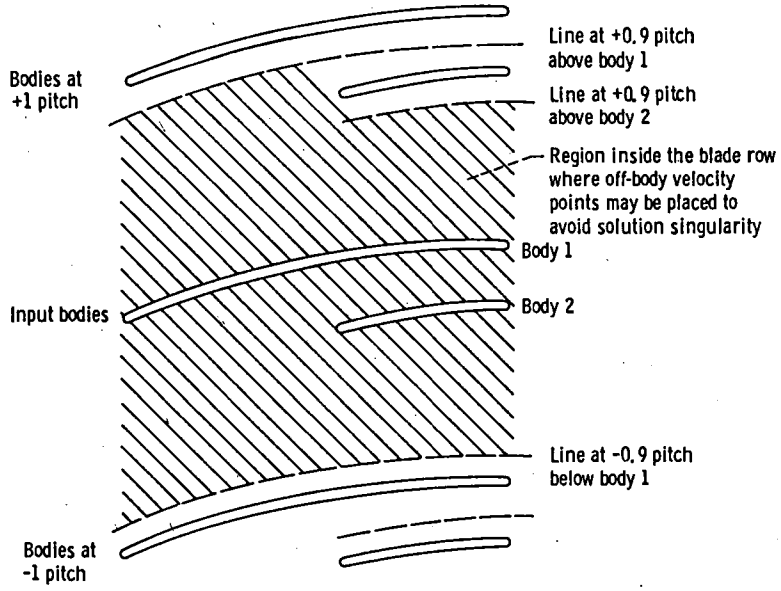

Figure 5. - Off-body velocity point placement.

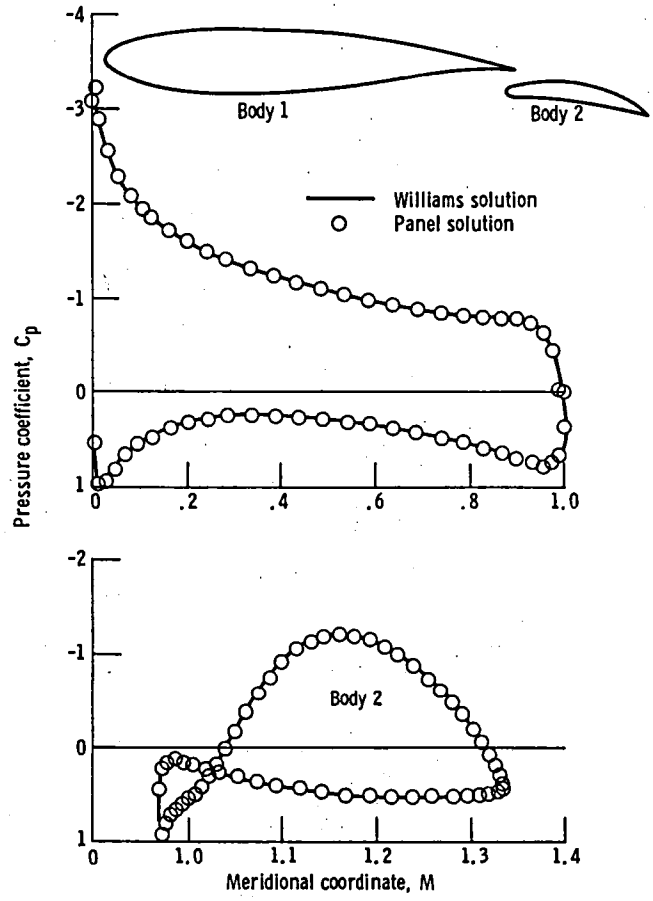

Figure 6. - Solution for isolated, two-element airfoil.

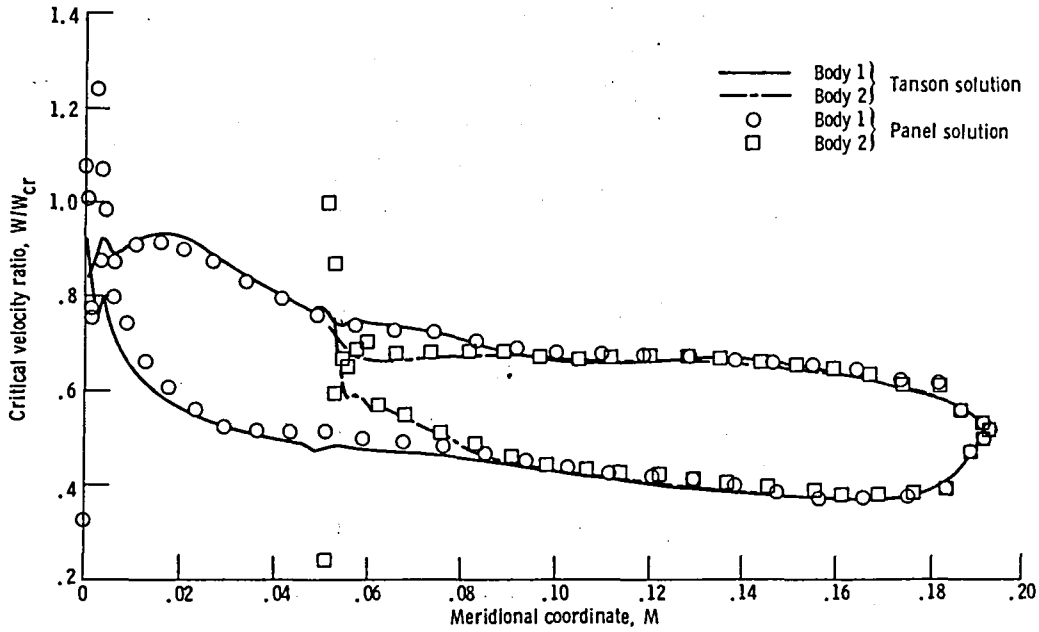

Figure 7. - Solution for centrifugal compressor with splitter.

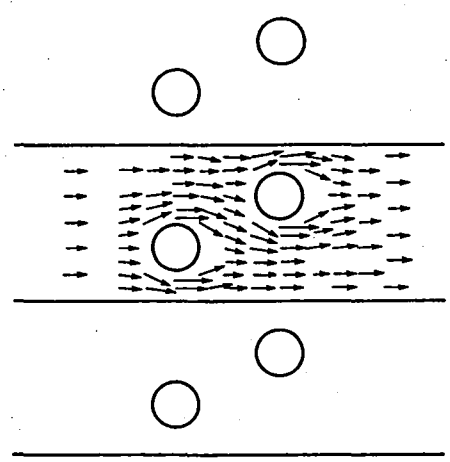

Figure 8. - Solution for two cylinders in a channel.

1. Report No.	2. Government Accession No.	3. Recipient's Catalog No.
NASA TM-87104		

4. Title and Subtitle	5. Report Date
A FORTRAN Computer Code for Calculating Flows in Multiple-Blade-Element Cascades	November 1985
	6. Performing Organization Code: 505-31-04
7. Author(s): Eric R. McFarland	8. Performing Organization Report No.: E-2701
	10. Work Unit No.
9. Performing Organization Name and Address: National Aeronautics and Space Administration, Lewis Research Center, Cleveland, Ohio 44135	11. Contract or Grant No.
	13. Type of Report and Period Covered: Technical Memorandum
12. Sponsoring Agency Name and Address: National Aeronautics and Space Administration, Washington, D.C. 20546	14. Sponsoring Agency Code

15. Supplementary Notes

Computer code available from COSMIC, Atlanta, Georgia.

16. Abstract

A solution technique has been developed for solving the multiple-blade-element, surface-of-revolution, blade-to-blade flow problem in turbomachinery. The calculation solves approximate flow equations which include the effects of compressibility, radius change, blade-row rotation, and variable stream sheet thickness. An integral equation solution (i.e., panel method) is used to solve the equations. A description of the computer code and computer code input is given in this report.

17. Key Words (Suggested by Author(s))	18. Distribution Statement
Turbomachinery; Multiple body; Cascade flow; Blade-to-blade flow	Unclassified - unlimited STAR Category 02

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

*For sale by the National Technical Information Service, Springfield, Virginia 22161

End of Document