MULTISTAGE TURBOMACHINERY FLOW CALCULATION PROGRAM-- MULTALL

MULTALL-OPEN-17.1 OLD_READIN DATA INPUT
January 2017

WARNING: The data input for the OLD_READIN option in MULTALL-OPEN-17.1 is closely, but not exactly, the same as used in previous versions of MULTALL, most recently in version 15.2. The main differences are the use of, and calls to, subroutines IF_DESIGN, IF_RESTAGGER and IF_LEAN, and extra input needed for the option to perform a throughflow calculation. Also, the separate plotting and restart files used in previous versions are combined to a single unformatted file, "flow_out", in this version.

SOLUTION ALGORITHM

All recent versions use the SCREE algorithm instead of the previous "Opposed Difference Scheme" which was used up to MULTIP75. The SCREE scheme is an extremely simple method that is completely second order accurate in space. The primary flow variables F (where $F = \rho$, ρE, ρV_x, ρV_r, or $\rho r V_t$) are updated on every timestep using

$$\Delta F = \ \left(2\frac{\partial F}{\partial T}\right)^{n} \ - \ \frac{\partial F}{\partial T})^{n-1}\right)\Delta t$$

where n is the time step level. This involves only a single flux evaluation per time step and so is much faster than multi-step schemes. The scheme is only first order accurate in time, but this is not important for steady calculations. The SCREE scheme can be used with much lower values of artificial viscosity (smoothing) than early versions of the code, which used the opposed-difference algorithm, and does not need any special treatment or loss of accuracy to handle reversed axial velocities. The scheme is also better at very low Mach numbers and solutions can be obtained for effectively incompressible flow at Mach numbers of order 0.1.

The maximum stable timestep with the SCREE scheme is slightly larger than with the opposed difference scheme (typically the maximum stable value of the input variable CFL is 0.5, but as a safety factor it is more usual to set CFL = 0.4). The CPU time per point per timestep is only slightly reduced but the

number of timesteps required for convergence is generally significantly reduced compared to all previous versions of this code. Convergence is generally much more continuous than with the opposed difference scheme and once over the initial transients the graph of log(residual) vs time step number soon becomes a straight line.

The "super_scree" algorithm available in TBLOCK never worked well on real problems, although it works well on a uniform grid. It was found to be sensitive to the multigrid on non-uniform grids. It was then realised that the "super_scree" algorithm could be approximated without using the additional storage for the derivatives at the N-2 timestep by setting

$$\Delta = F_1 * R_n + F_2 R_{n-1}$$
$$R_{n-1} = R_n + F_3 * R_{n-1}$$

Where Δ is the change applied and R_n is the residual at step n . F_1, F_2 and F_3 are constants.

This makes the second term into a geometric series of past residuals and to make to sum of all coefficients = 1.0, so that its time steps are comparable to the "scree" scheme, we must set

$$F_1 + F_2/(1-F_3) = 1.0 .$$

 It is found that the combination

$$F_1 = 2.0, \ F_2 = -1.65, \ F_3 = -0.65$$

is close to optimum and allows CFL numbers up to about 0.7 compared to 0.4 for the "scree" scheme.   The combination $F_2/(1-F_3)$ is referred to as the effective value of F_2, F_{2eff} , and it is this value that must be input in the data file. i.e. the typical input value of F_{2eff} = -1.0 .

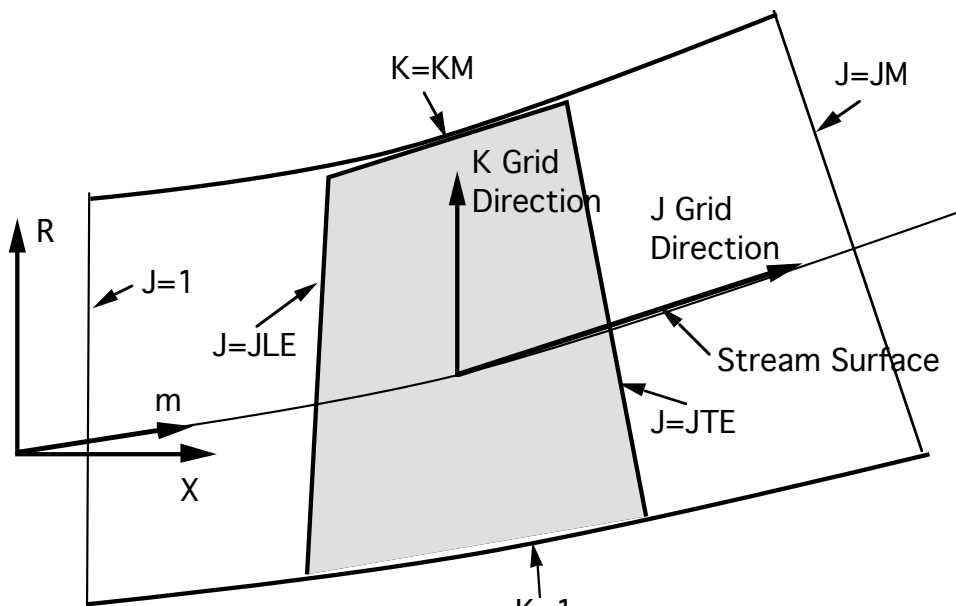This is called the "SSS" (Simple Super Scree) scheme.

It was also found that the CFL number could be increased slightly further by the use of residual averaging, i.e. smoothing the values of R_n . A new smoothing subroutine SMOOTH_RESID is added to perform this, although its use is optional. A single smoothing with smoothing factor 0.4 allows a small
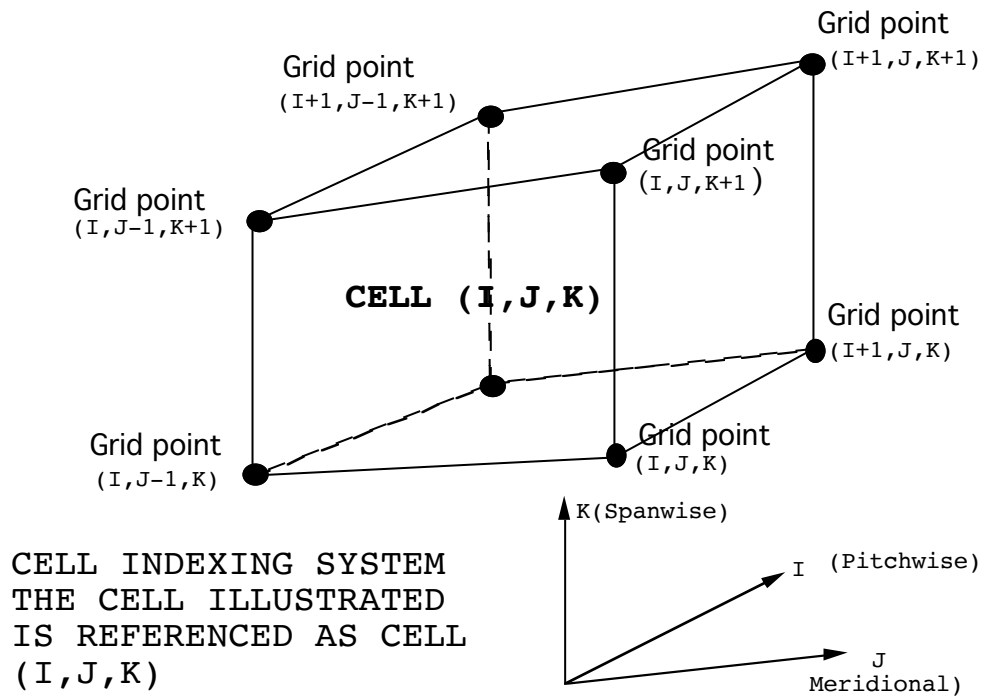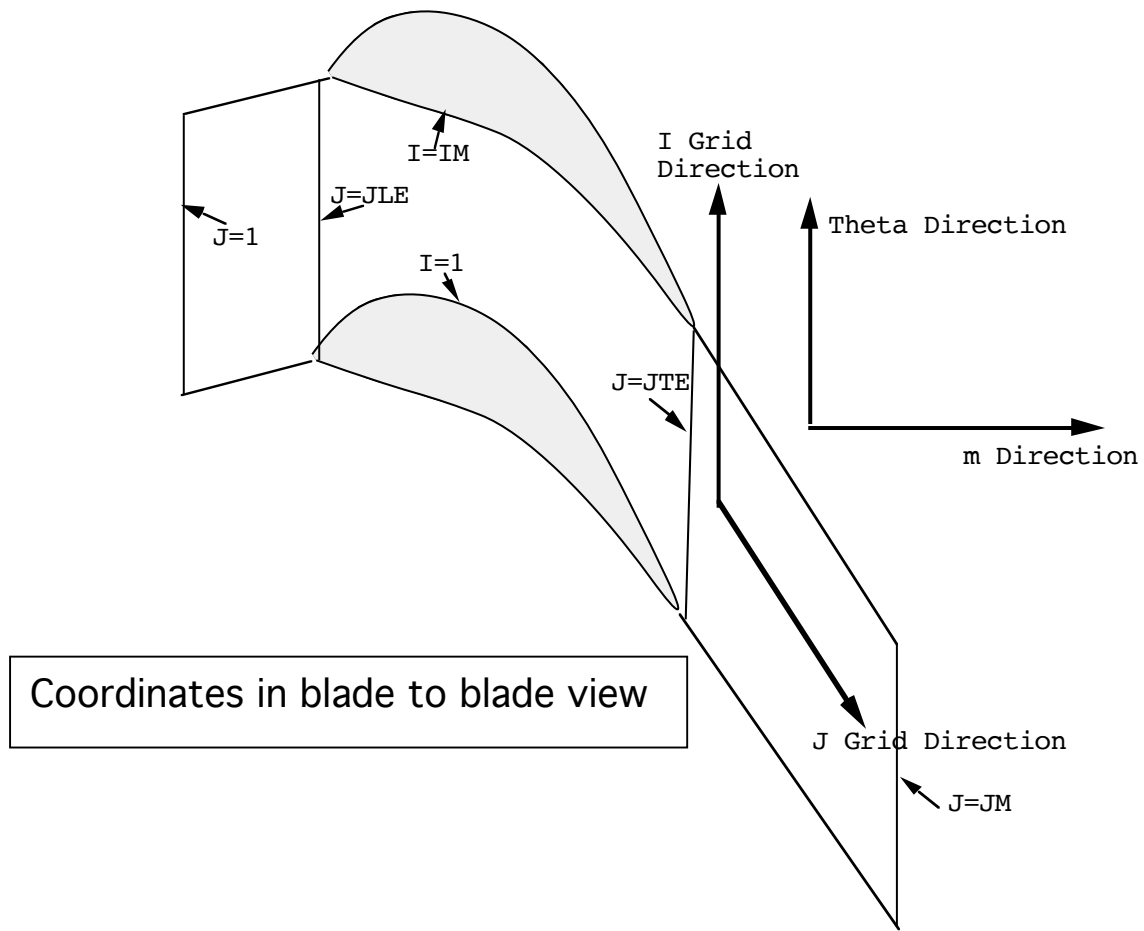
increase in CFL at the expense of about 7.5% increase in run time per step. However, a major advantage is an increase in robustness. The program no longer fails when the stable CFL number is exceeded. Instead the residuals at the unstable points oscillate at high frequency about a steady average. Although the residuals may not decrease to low levels the resulting solutions appear identical to fully converged ones. CFL values up to 0.9 can sometimes be used in such cases but it is safer to choose 0.7 as a standard value.
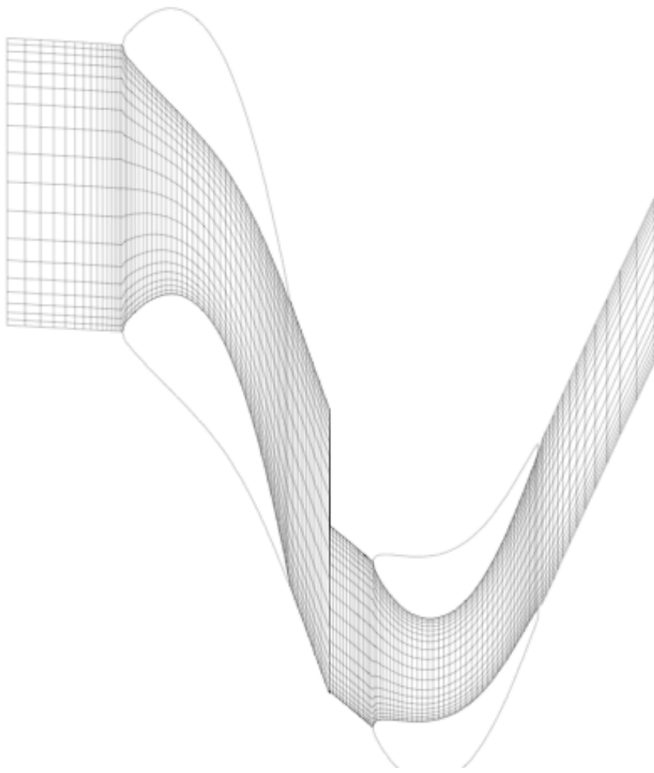
To use the SSS scheme set the value of ITIMST to a negative value, i.e. -3, -4, -5 or -6, the options are then the same as with the positive value of ITIMST but using the SSS scheme instead of the SCREE scheme.

GRID

The program uses a standard "H" grid composed of pitchwise (J and K constant, I varies), streamwise (I and K constant, J varies) and quasi-orthogonal (I and J = constant, K varies) grid lines. The simple H grid greatly simplifies grid generation, the application of the periodic boundary conditions and the modeling of the mixing planes. However, it inevitably leads to highly sheared cells for staggered blade rows but experience is that the numerical errors associated with this (as judged by the entropy change in inviscid flow) are negligible when "viscous" grids with more than about 30 points across the pitch and span are used. All variables are stored at cell corners, as illustrated above, the author finds this to be simpler and more accurate than cell centre storage. The cell indexing system is also illustrated in the Figure below. The cell numbered (I,J,K) has the grid point I,J,K at its corner with the largest J value, lowest I and lowest K values.

I=IM

J=JLE

J=1

I Grid
Direction

Theta Direction

I=1

J=JTE

m Direction

Coordinates in blade to blade view

J Grid Direction

J=JM

Grid point
(I+1,J,K+1)

Grid point
(I+1,J-1,K+1)

Grid point
(I,J,K+1)

Grid point
(I,J-1,K+1)

CELL (I,J,K)

Grid point
(I+1,J,K)

Grid point
(I,J-1,K)

Grid point
(I,J,K)

K(Spanwise)

CELL INDEXING SYSTEM
THE CELL ILLUSTRATED
IS REFERENCED AS CELL
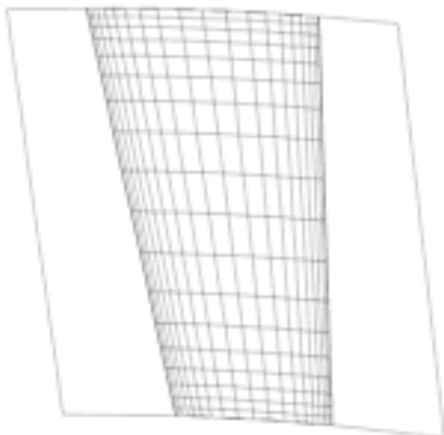(I,J,K)

I (Pitchwise)

J
Meridional)

Blade-to-blade view of grid on a streamwise surface.
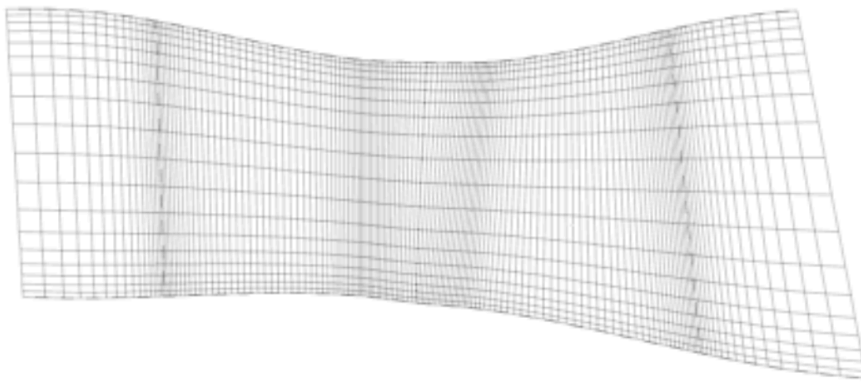
Meridional view of grid

Quasi-orthogonal view of grid

A typical number of grid points for an inviscid calculation would be IM = KM = 28, JM = 75, with fairly uniform spacing of the points in each direction, except for the J spacing being considerably reduced around the leading edge. For a viscous calculation a typical number would be IM = KM = 49, JM =120, with highly non-uniform spacing in the pitchwise and spanwise directions, a typical grid stretching factor in these directions would be 1:20.

The grid lines should be closely spaced around the leading edge where the cells are highly skewed and the flow properties change very rapidly, but experience shows that it is best to use a relatively coarse grid spacing with a cusp at the trailing edge. This is because a fine grid around the trailing edge usually gives unrealistic negative loading at the trailing edge. A cusp at the trailing edge is the standard option in this program but version 09 and beyond allows a body force field and a find grid to be used instead of a cusp to force the flow to separate at the trailing edge.

MULTIGRID

The most important factor in accelerating the convergence of the calculation is the use of multigrid. Three levels of multigrid are usually used and, unlike most other methods, the block sizes are not limited to simply doubling of the number of cells in each direction. In fact the size of the blocks may be chosen arbitrarily by the user but experience is that blocks of 3x3x3 cells for the lowest level and 9x9x9 cells for the middle level are often the optimum. It is best, but not essential, if there are a whole number of blocks across the span and pitch of the blade passage. The third level of multigrid is a one-dimensional calculation in which the blocks extend across the whole pitch and whole span with typically about 5 blocks per blade row in the streamwise direction. The gives very rapid transfer of information from inlet to outlet of the whole calculation and so greatly speeds up convergence.

The number of time steps for convergence depends greatly on the problem, particularly on the number of stages and on the uniformity of the grid, but it is typically in the range 2000 – 5000. The more the number of blade rows being calculated the larger is the number of steps required for convergence.

MIXING PLANE MODEL

Development of a satisfactory mixing plane model is an extremely difficult task. The objective is to allow the flow to mix out instantaneously at a plane,

rather than gradually within the downstream blade row, whilst maintaining the mixing loss at a similar value. The mixing plane must conserve the pitchwise averaged fluxes of mass, momentum and energy but it must not impose pitchwise uniform conditions on the flow. In general the static pressure will rise and the entropy increase across the mixing plane to simulate the real mixing process. The mixing plane treatment has evolved over many years and has been changed and improved compared to that in MULTIP75. This was necessary because the previous mixing plane treatment sometimes gave inaccurate results with the new SCREE scheme. The quasi-orthogonal grid lines at J = JMIX and J = JMIX +1 **must be** coincident in the meridional view, i.e. they have the same x and r coordinates. This is done automatically by subroutine GRIDUP if the options ISHIFT = 2 , 3 or 4 are used (CARD 2). **Use of this option is very strongly recommended.**

MULTALL-14.6 and later have a new mixing plane model which is described in detail in NOTE 22.

VISCOUS MODELLING

As in all previous versions viscous terms are included via body forces or source terms. The use of body forces to solve the Navier-Stokes equations is not an approximation and can be made as exact as any other method. The advantage of using the body force model is that the viscous terms need not be evaluated every time step, typically they are evaluated only every 5 steps leading to a very significant saving in CPU time.

 Three different viscous models are included.

In the earliest model, subroutine LOSS, the body forces are obtained from a thin shear layer approximation to the Navier-Stokes equations. The "thin shear layer" model is an approximation which assumes that viscous normal stresses and viscous stresses on the quasi-orthogonal faces of the elements can be neglected. Experience is that this is not a severe limitation for turbomachinery flows and a great deal of user experience confirms that this model gives reasonable results.

An improved mixing length model in available in subroutine NEW_LOSS. This is full Navier–Stokes with the turbulent viscosity being proportional to the (mixing length squared) x the vorticity. In NEW_LOSS the local mixing length limit is calculated from the distance of the mid-span and mid-pitch point to the nearest solid surface. It is therefore roughly proportional to the local

blade passage width. This length is then multiplied by a scaling factor which in input as data. The scaling factor is input for each blade row at blade row inlet (mixing plane), leading edge, trailing edge and exit (mixing plane) and is taken to vary linearly with meridional distance between these points. The value of the scaling factor should be similar to the mixing length limit used in the original LOSS routine but experience is that is needs to be slightly larger, say 0.04 instead of 0.03. The mixing length is taken as the lower of this value and the local perpendicular distance to the nearest wall. Note that, unlike in LOSS, the mixing length limit cannot be varied between the blade surfaces or the endwall surfaces, it can only be varied with meridional distance. Free stream turbulence may be included by inputting the ratio of the free stream turbulent viscosity to laminar viscosity in every blade row.

SPAL_LOSS is the Spalart-Almaras turbulence model. In this an additional convection equation is solved for the transformed turbulent viscosity. This uses about 20% more CPU time than the original LOSS method. However, the very complex source term used in the SA model is only calculated every 5-10 time steps and so does not use too much CPU time. In addition to the 4 source terms used in the basic SA model an additional source term, ST_MIXL, has been added. This forces the turbulent viscosity towards the value obtained from NEW_LOSS model. Each of the source terms is multiplied by a scaling factor, which is input as data, and so the latter term would usually be turned off (scaled by zero) to obtain the basic SA model. By including the additional source term either hybrid mixing length-SA models or a modified mixing length model with convection of the turbulent viscosity, may be obtained. Experience is that better results are sometimes obtained by increasing the main turbulent viscosity source term, ST0, by 50%. A question arises from the transfer the turbulent viscosity across a mixing plane. This is modeled by pitchwise averaging the values of turbulent viscosity upstream of the mixing plane and passing a fraction of this average across the mixing plane as a pitchwise uniform variation to the next blade row. The fraction transferred in input as data for each blade row. It is difficult to know what is the correct fraction to transfer but a value around 0.5 seems appropriate.

Version 17.5 includes two new terms in the SA model. These are taken from the paper by Vahdati et al, ASME paper GT2017-63245. They increase the main source term in the SA model, ST_0, when there is streamwise vorticity or an adverse pressure gradient and this is certainly the correct physical trend. The magnitude of the increase is determined by the input values of FAC_VORT and FAC_PGRAD. Vahdati et al suggest values of

0.9191 for FAC_VORT and 0.6565 for FAC_PGRAD but these are very likely to be case dependent and more experience is needed before they can be used with confidence in MULTALL.

The wall shear stresses are obtained from wall functions. The original model assumes that the second grid point from the wall is either in the log-law region of a turbulent boundary layer, or in a laminar boundary layer. In this model the wall shear stress is obtained from a curve fit to the log law for equilibrium boundary layers. This model is used if the variable YPLUSWALL is set to zero. A new wall function, to obtain the skin friction coefficient is also available. This is used if YPLUSWALL is greater than 5.0 and uses an assumption that the first grid point is in the laminar sub layer at the specified value of YPLUS. This simplifies the calculation of skin friction with no loss of accuracy.   However, the latter model does not allow the skin friction to change with Reynolds number and the chosen value of YPLUSWALL must be adjusted manually to allow for different Reynolds numbers. Hence the first (original) method is generally preferred. Boundary layer transition can be specified on any surface or can be predicted very approximately by a simple transition model.

A newer wall function based on that of Shih et al, NASA TM-1999-209398, is available in version 17.5. This includes a more complex fit to the log law and an effect of the pressure gradient on the wall shear stress. The pressure gradient term is regarded by the author as rather dubious since it can switch direct very suddenly at a separation point. This model is used without the pressure gradient term if YPLUSWALL is set = -1.0 . Both velocity and pressure gradient terms are used if YPLISWALL = -11.0 . So far the indications are that this model gives very similar results to the original, much simpler, model

Earlier versions of this program omitted shear work and internal heat flows because they may be shown to cancel in a flow with uniform stagnation temperature and a Prandtl number of unity. However, for cooling flows that model is not realistic and inclusion of shear work and internal heat flows is standard in all recent versions. All surfaces are, however, assumed to be adiabatic so heat transfer from solid to the gas cannot be calculated.

In early versions the laminar viscosity was obtained from an input value of Reynolds number since it was felt that a turbomachinery designer is more likely to know that that the actual viscosity. However, all recent versions allow

the dynamic viscosity to be input rather than the Reynolds number. This option is used if the input value of Reynolds number is negative in which case the viscosity at 288K is the (absolute value of Re) x 10^{-5}. If this is done then the input value viscosity is automatically scaled by a power law to allow for its variation with local temperature. E.g. for air the viscosity would be input as -1.9. This option is especially useful for multistage high speed machines where the temperature, and hence viscosity, varies significantly.

IMULTALL-08 and above INCLUDES an option to allow for the increased skin friction due to surface roughness. Different levels of roughness can be specified on each surface.

Boundary layer transition can be specified at a fixed "J" value in both NEW_LOSS and SPAL_LOSS in exactly the same way as for the original mixing length method. However, the prediction of transition by specifying the variable FTRANS is not possible with the SPAL_LOSS model although it is still usable with NEW_LOSS.

Versions 14.2 onwards have an option to reduce the turbulent viscosity for grid points within the laminar sub layer and buffer layer, typically up to yplus = 25. This only has any effect if more than 2 points are within these layers, which is only likely to occur when very fine grids are used. This has little effect on the SA model but makes solutions with LOSS10 or NEW_LOSS more grid independent.

TIP LEAKAGE FLOWS

Tip leakage for plain tip clearances is calculated using the "pinched tip model" in which the blade is thinned towards the tip and periodicity is applied across the tip gap. This model seems to give very realistic results although the actual tip gap calculated may need to be less than the physical gap to allow for the contraction of the leakage jet. A reduction in the tip gap to about 0.6 of the physical gap is recommended. The tip clearance can be specified as a fraction of the local span at both the leading and trailing edges of the blade, and the grid is automatically adjusted to fit the gap.

SHROUD LEAKAGE FLOWS

MULTALL contains a shroud leakage model that models the leakage flow and loss for shrouded turbine blades. The leakage mass flow is estimated from the seal clearance, the number of seals, the upstream stagnation pressure and the downstream static pressure and is bled off from the main flow. The change of

angular momentum of the leakage flow due to friction on the shroud and casing are estimated using input values of the skin friction coefficient. The work done on the shroud by the leakage flow is also calculated. The leakage flow is then injected into the main flow downstream of the blade row and the conservation equations determine the mixing loss. Either hub or tip leakage can be calculated and the flow may be from upstream of the blade row to downstream, as in turbine blades, or from downstream to upstream as in compressor blades.

GENERATION OF MODIFIED GRIDS

The grid spacing in the pitchwise (I) and spanwise (K) directions can be easily changed in the input data. The spacing in the streamwise (J or meridional direction) can also be changed using subroutine NEWGRID. This enables a new grid with a different number of points to be generated from the input data. This option has also been improved compared to previous versions. Previously the *relative* meridional spacing of all the new grid points had to be input as data. This is retained as an option but it is now possible to specify the *relative* spacing at just a few points as a function of the meridional distance and the program interpolates in these few points to obtain all the new grid spacings and hence the new grid points.

CUSP GENERATION

Although it is not essential to use cusps at the blade leading and trailing edges it is strongly recommended that a cusp is used at the trailing edge. A flexible method of cusp generation at blade trailing edges is included in the program. The default is the original cusp (or very close to it) but options have been added to allow the shape and length of the cusp to be chosen and also to allow the cusp to be treated as a part of the blade so that it carries load. Cusps are not necessary at the blade leading edge as long as sufficient grid points are used to define the flow around the leading edge circle. A very fine grid is needed to achieve this, typically about 6-10 points on the leading edge circle, this number may be reduced by using a cusp at the leading edge.

MULTALL-09 and above include an option force the flow at the trailing edge to separate by a body force. This allows a fine grid to be used around a thick trailing edge without the solution generating unrealistic reverse loading at the TE. See Note 21. This option is seldom used.

EXIT FLOW THROTTLING

A new addition (version 08) is the possibility to use a "perforated plate" type of downstream boundary condition. This is used when there is a separated flow at the downstream boundary so that the meridional velocity is negative and the standard boundary condition becomes ill conditioned. This can be prevented by simulating the presence of a flow resistance, such as a gauze or a perforated plate, at the boundary. This effectively adds a pressure drop equal to THROTLE_PLATE x $0.5*\rho*(Vm^2 - Vm_{mid}^2)$ at the exit. Thus it should not change the average exit pressure but it does force the flow to become more uniform at the exit boundary. This is thought to be preferable to using the previous option of SFEXIT. A typical value of THROTLE_PLATE is about 2. If THROTLE_PLATE is set to zero then the option is not used.

Version 09 includes an option the use a throttle boundary condition to vary the exit static pressure linearly in proportion to the exit mass flow rate. This gives improved stability near to the stall point for compressors. This has been improved in version 14.2 and above in which the exit static pressure is made to vary parabolically with the exit mass flow according to:

$$Pexit = THROTTLE\_PRES * (m_{exit}/THROTTLE\_MAS)^2$$

Where THROTTLE_PRES is the required exit pressure in N/m^2 and THROTTLE_MAS is the expected exit mass flow rate in Kg/sec. Changes in exit pressure are relaxed by RFTHROTL which, together with THROTTLE_PRES and THROTTLE_MAS, is input as data if THROTTLE_ALL is non zero. Increasing the value of THROTTLE_PRES will increase the back pressure and move a compressor towards stall. However, forcing the exit flow to lie on a parabolic pressure:mass flow characteristic through the specified point allows a closer approach to the true stall point that was previously possible. Use of this option is recommended for all compressor calculations. THROTTLE_ALL must be set to zero to prevent the use of this option.

BLADE AND ENDWALL COOLING FLOWS

Cooling flows can be added at any point on the blade and endwall surfaces. The flow is added through a series of "patches" whose I,J,K boundaries are specified in the input data. If the "mixing plane" falls within a region where coolant is being added then two separate patches, one upstream and one downstream of the "mixing plane" must be used. The coolant mass flow, stagnation temperature, stagnation pressure, ejection Mach number and flow

directions must be specified for each patch. If it is required to model individual cooling holes then each patch may be one grid cell in size, but this requires a great deal of input data and it is more usual to specify a single patch to cover multiple cooling flows. The overall total-to-total efficiency is calculated and printed out, allowing for the potential work of all the cooling flows. However, the polytropic efficiencies, which are also printed out, relate the mass averaged inlet and outlet flow conditions and are not meaningful when cooling flows are added.

BLEED FLOWS

Flow can be bled from the machine (as is common in steam turbines for feed heating or in gas turbine compressors for turbine cooling) at any point on the hub and casing. As with coolant flows the flow is bled off from a "patch" whose I,J,K boundaries must be specified by the user. However, the bled flow is always assumed to have the flow properties at the point where it is bled off and so these cannot be specified by the user.nThe machine power output/input is calculated allowing for the bled flows but the efficiencies which are printed out are defined by using the inlet and outlet states of the flow remaining in the machine and do not allow for the bleed flows.

LOW SPEED FLOWS

The program has been extended to use the artificial compressibility method to work with very low speed or incompressible flows. This works be inputting an artificial speed of sound, VSOUND, and calculating pressure changes directly from changes in an artificial density using   DP = VSOUND**2 x DRHO. The change DRHO in the artificial density is obtained from the continuity equation in the usual way. The true density is then calculated from the pressure and internal energy and this density is used to find the flow velocities.

 The velocity of sound should be chosen to be about twice the maximum velocity expected in the whole flow field so that the program runs at an effective Mach number of about 0.5. This method can be used to speed up convergence for flows with Mach number less than about 0.25 or it can be used for extremely low Mach numbers less than 0.05 where the basic program may not converge, or for completely incompressible flows. The low Mach number option is called if ITIMST = 5 and the completely incompressible option is used if ITIMST = 6. If either option is used than the artificial speed of sound and a relaxation factor on changes in true density must be input in the

last line of data. The value of the relaxation factor should be about 0.05. For completely incompressible flow then the fixed value of fluid density must also be input in the last card.

REAL GAS PROPERTIES

Previous versions of the program have always used perfect gas properties. Version 7.11 and above allows real gas properties where the gas constant is independent of temperature but the specific heat capacity, Cp, varies with temperature. The value of Cp is taken to vary quadratically with temperature according to

$$ Cp \;=\; Cp_1 \;+\; Cp_2(T - T_{ref}) \;+\; Cp_3(T - T_{ref})^2 $$

The values of Cp_1, CP_2, Cp_3 and T_{ref} are input as data. For combustion products the values of Cp_1, Cp_2 and Cp_3 are approximately 1272.5, 0.2125 and 0.000015625 J/kg K at Tref = 1400 K . The gas constant is closely 287.5 .

This option is used if the value of Cp input in the standard data set is negative, in which case values of Cp_1, CP_2, Cp_3 , T_{ref} and RGAS are input in the next line of data.

REDESIGN OPTION

This new feature is introduced in version 07.11 . It is possible to perform a limited redesign the blade sections on the input stream surfaces by generating new camber lines and thickness distributions. The axial and radial coordinates of the leading edge and trailing edge of the section, and the stream surface on which it is input, are not changed so that the blade axial chord remains constant. The method of specifying the new blade section is very similar to that in the author's blade design system, STAGEN .

A different option is to restagger the input blade, in which case blade is simply rotated about a specified axis so that its leading and training edge coordinates will change. The blade can also be leaned in the tangential direction.

REPEATING STAGE OPTION

In many multistage machines the flow repeats from stage to stage with the velocity profiles and stagnation pressure profiles remaining almost constant

and only the stagnation pressure level changing. The shape of the stagnation temperature profile cannot remain constant since to do so would imply the same entropy rise for all streamlines. In reality the stagnation temperature will increase near the end walls relative to that near mid span. When designing a repeating stage it is desirable to try to satisfy this condition and this can be done by automatically feeding back the exit stagnation pressure profile and yaw angle profile to the inlet boundary conditions. The stagnation temperature profile and pitch angle profile are not fed back but remain at the values input in the original data set. When run in this way the program naturally takes longer to converge but it saves multiple runs with manual adjustment of the boundary conditions.

QUAS1 3D FLOW

Versions 14.2 onwards allow quasi-3D flow on a blade-to-blade stream surface to be calculated with only a single cell, i.e. two grid points, being used in the spanwise direction. To use this option set KM = 2. This enables Q3D solutions for a single blade row to be obtained in the order of 15 seconds. The stream surface is defined by the axial and radial coordinates of a single surface on which the blade coordinates are input and table of the relative stream surface thicknesses against meridional distance. It is important to realise that the stream surface thickness has a very large influence on the flow pattern and so it should be chosen realistically. The flow is forced to follow the stream surface by using a body force acting perpendicular to the surface so that it does not influence the flow on the stream surface. The value of this body force is controlled by the input value of Q3DFORCE but the value does not seem to be critical and values between 0.1 and 1.0 are usually acceptable.

To use this option set KM = 2 and input the value of Q3DFORCE in CARD 17C.

THROUGHFLOW MODE

The option to perform an axisymmetric throughflow calculation was introduced in version 17.1 . It uses two grid points in the pitchwise direction and so is invoked by setting IM = 2. The full blade geometry must still be input but the number of grid points in the meridional and spanwise directions can be greatly reduced. Typically JM = 30 and KM = 19 could be used. Run times are then of order 10 seconds per blade row, the speed is increase if all the array dimensions in the I direction as set = 2.

The flow is forced to follow a mean stream surface, which is defined as the blade centre line plus a deviation angle, which is input as data. The deviation from the centre line angle varies linearly from zero at the leading edge to the input deviation at the trailing edge. Viscous effects are estimated by setting the value of YPLUSWALL. This gives a skin friction coefficient of $2/(YPLUSWALL)^2$ on all solid surfaces so a value of YPLUSWALL = 20 would give a typical skin friction coefficient of 0.005 . Tip leakage flows may be included in the usual way, the leakage flow will be undeflected and will produce a mixing loss as it mixes with the mainstream flow downstream. Choking of blade rows and supersonic deviation is predicted but all shock waves are treated as normal shocks and so may not be realistic.

To use this option set IM = 2 and input data for the deviation, etc, in cards 18AA – 18C and 47 .

COMPILING THE PROGRAM

The program is written in very standard FORTRAN77 and should compile and run on any computer with a FORTRAN compiler. The only non-standard routine is the timing call, which evaluates the CPU time per time step. This will vary for different compilers and operating systems. At present the "MCLOCK" routines for the LINUX g77 and gfortran compilers are coded but these may need changing. The timing calls are not essential for the running of the program and can be deleted or commented out if not required. To do this look for all 3 occurrences of the string "MCLOCK" and either delete them or place a "C" as the first character of the line in which they occur.

All the variables are declared and dimensioned within a single common block, e.g. "commall-open-17.1", which must be present in the same directory as the program when is compiled. The sizes of the arrays are declared by PARAMETER statements in "commall-open-17.1". Typical maximum dimensions of the grid would be 65 points in the pitchwise and spanwise directions and 1000 points in the streamwise (J) direction. With these dimensions the program uses about 750 MBytes of memory. The large number of streamwise grid points is only needed for multistage calculations with many blade rows, typically about 125 points per blade row is sufficient.

The speed of execution is generally increased by using the highest level of compiler optimization available.

RUNNING THE PROGRAM AND INPUT AND OUTPUT FILES

The input data is read from FORTRAN unit 5. The file name is not specified within the program and so the program is most easily run using a command such as

MULTALL.X < DATA.IN

Where MULTALL.X is the name of the compiled executable code and DATA.IN is the name of the data file. The program will run either until the average residual reaches the specified tolerance, CONLIM, or for the maximum number of time steps specified, NMAX.

Note that most of the input data in DATA.IN must be formatted. This may easily be changed to unformatted data if required but experience is that it is much easier to spot errors in formatted data than in unformatted. With formatted data if a value is not present it is taken as zero.

Several different output files are produced. The standard output is to FORTRAN unit 6, which defaults to the screen, this gives a summary of the convergence history every 5 time steps and a more detailed average of the flow properties every 200 time steps. If it is required to send this output to a file then use a command such as

MULTALL < DATA.IN > OUTPUT.OUT

Where OUTPUT.OUT is the name of the output file.

Other output files are set within the program and are produced automatically. These are:

"stage.log " A formatted file containing the convergence history with values of the rms error, continuity error and inlet mass flow is written to fortran unit 4 output every 5 time steps. This file may be used to plot out the convergence history.

"flow_out" An unformatted file containing the all the flow properties is written to fortran unit 7. This file may be output after specified numbers of time step. It is also automatically output on convergence or on reaching the specified maximum number of time steps. This file, together with the file "grid_out" may be used to plot out the results. The same file is also used as a restart file if a restart is requested.

"grid_out" An unformatted file containing the grid coordinates is written to fortran unit 21. It is used together with "flow_out" to plot out the results.

'global.plt" An unformatted file containing the one-dimensional mass averaged flow data is written to fortran unit 11. This may be used to plot out the one-dimensional variation of mass flow, stagnation pressure, stagnation temperature entropy or lost efficiency along the flow path.

"results.out"   A formatted file containing selected flow properties is written to fortran unit 3. The properties may be selected as described in Note

13. This file may be very large and so the output requested should be chosen carefully.

"loss-co.plt" A formatted file which contains the loss of isentropic efficiency at every "J' station is written to fortran unit 23 at the end of any run and may be used to plot lost efficiency against meridional distance.

"mixbconds"   A formatted file written to fortran unit 12. This contains the mixed out values of the flow properties at each mixing plane at every spanwise (K)   grid point. It may be used to provide the inlet boundary conditions to a subsequent calculation on an individual blade row or smaller group of blade rows.

The plotting programs that use some of these files are based on the HGRAPH plotting package and so are not publicly available.

If any of these output files are not required then the relevant parts of the code can be commented out or removed. The plotting programs referred to above use the graphics library HGRAPH which was written by Prof H.P. Hodson at the Whittle lab.

There is an option to stop the program and write out the results and a plot file at any time. Every 10 time steps the program opens and reads a file named "stopit" which may be opened and edited by the user whilst the program is running. This file contains a single number, if the number is zero the execution will continue, if it is 1   then the execution ends and the results files are written. To stop execution, pause the program, edit "stopit" and type 1 in place of the 0 then restart execution of the program .

CARD LIST

<u>Card 1</u>   NAME

Format (A72)

NAME   Title of run - any alphanumeric characters in columns 1 - 72.

<u>CARD 2</u>   IM, JDUM, KM, IF_ROUGH, NMAX, IFCOOL, IFBLEED, NOSECT, NROWS, IFMIX, ISHIFT, KIN, NEXTRAP_LE, NEXTRAP_TE, NCHANGE
Format (16 I5)

IM =   Number of grid nodes in the circumferential (I) direction.

JDUM =   Dummy input, no longer used.

KM =   Number of grid nodes in the spanwise (K) direction.

IF_ROUGH   =0 for smooth surfaces, = 1 if any surface is to be treated as rough, extra data is then needed in Card 46C.

NMAX=   maximum number of iterations, typically = 3000 for a single blade row, more for multiple rows.

IFCOOL   = 0 if no cooling flows, = 1 if cooling flows are to be specified on any blade row.

IFBLEED   = 0 if no bleed flows, = 1 if bleed flows are to be specified on any blade row.

NOSECT=   Number of streamwise surfaces on which the blade
                  geometry data is to be input. If NOSECT is made
                  negative then different numbers of input surfaces
                  may be used on different rows with the actual
                  number being input in CARD 6AA.
        Note that NOSECT must be less than or equal to KM.

NROWS   Number of blade rows to be calculated. The
                  maximum number permitted is determined by the
                  parameter NRS in the array dimensions.

IFMIX   Use circumferential averaging and flux
                  extrapolation at the mixing planes if
                  IFMIX =1. If IFMIX = 0 do not average but
                  feed the circumferential variation from
                  one row into the next row. Usually use
                  IFMIX=1. See also FEXTRAP in Card 44.

ISHIFT   Provides a facility for automatically
                  shifting the axial coordinates of the
                  data read in and also for making the
                  mesh spacing uniform or to vary
                  geometrically in the gap between blade
                  rows. **See note 18.**

  ISHIFT = 2 or 4 is strongly recommended as they automatically
set the grid line at the mixing plane and the one immediately
downstream of it to be very nearly coincident. ISHIFT =4 makes the grid
surfaces smooth between the blade rows without changing the hub or
casing shape.

KIN   Is the number of K points used for input
                  of the inlet boundary conditions. This normally =
                  KM   but use of KIN permits a different number of
                  points (KIN) to be input. If the number of input

points equals KM but their spanwise spacing is different from the spanwise grid spacing then set KIN = -KM

NEXTRAP_LE	If ISHIFT = 2 or 3 then the grid lines upstream of the leading edge are made parallel to a line joining the leading edge point to the point at JLE + NEXTRAP_LE. Typically NEXTRAP_LE = 5 but increase for a highly cambered leading edge.
NEXTRAP_TE	If ISHIFT = 2 or 3 then the grid lines downstream of the leading edge are made parallel to a line joining the trailing edge point to the point at JTE – NEXTRAP_TE. Typically NEXTRAP_TE = 5 but increase for a highly cambered trailing edge.
NCHANGE	The smoothing and the damping are increased for the first NCHANGE steps. This gives increased stability during the initial transients. Typically set NCHANGE = NMAX/4. This is the default value if NCHANGE is set to zero.

<u>Card 3</u>   IN_VTAN, INSURF, IN_VR, IITIMST, IDUMMY, IPOUT, IN_FLOW
ILOS, NLOS, IF_RESTART, IDUMY, IBOUND, IF_REPEAT
Format (13 I5)

IN_VTAN	Defines the type of inlet boundary condition i.e. relative flow angle, absolute flow angle or absolute tangential velocity fixed at the inlet boundary. **See Note 1.**
INPUT	Defines the type of data input for the blade geometry. **See Note 2.**
IN_VR	= 1 means the inlet meridional pitch angle is specified and fixed. = 0 means the inlet meridional pitch angle is obtained using the condition $dV_r/dm = 0$. = 3 means a circumferentially uniform inflow. = 4 means a uniform static pressure at inlet. **See Note 3 for more details** .
ITIMST	Defines the type of timestep to be used. If the values are positive then the standard SCREE scheme is used, if they are negative then use the SSS scheme with standard coefficients. =3 0r -3 -> Non-uniform time steps updated as a function of Mach number. =4 or -4-> Uses the SSS scheme with the coefficients read in as data in card 3A. =5 or -5-> Low speed flow using an artificial speed of sound. Use if the maximum Mach number is less than about 0.25. Extra data is then needed in CARD 46D.

=6- or -6 >   Fully incompressible flow with constant density. Extra data is then needed in CARD 46E.

IDUMMY

Not used in this version but a value of 0 must be input to maintain compatibility with previous versions.

IPOUT-

= 1 means the exit static pressure is input at the hub and tip only and a linear variation with span is assumed,

= 0 means the exit pressure is only fixed on the hub and radial equilibrium is used,

= -1 means the exit pressure is only fixed at the tip and radial equilibrium is used,

= 3   means the exit pressure field is read in as data in card 31.

IN_FLOW

 Enables the inlet mass flow to be specified.

= 0 -> mass flow is fixed by the input pressure ratio. This is by far the most usual option.

= 2 -> mass flow fixed by the pressure ratio but forced towards an average value, possibly giving faster convergence.

= 3 -> mass flow is forced towards the value input in card 42 .

IN_FLOW =0 is usual. See Note 17 for more details.

ILOS

Defines whether or not to call the viscous LOSS subroutines.

Set = 0 for an inviscid flow.

To use the original LOSS model with neglect of viscous shear work and internal heat flows set = 9, to use LOSS with inclusion of viscous shear work and internal heat flows set = 10.

To us loss model NEW_LOSS set ILOS = 100.
To use the Spalart-Allmaras model set ILOS = 200.

NLOS   The viscous forces are updated every NLOS steps. The typical value is NLOS 5 or 10 . Larger values use less CPU time.

IF_RESTART   Determines whether or not a restart file is to be read from Fortran unit 7 to overwrite the usual initial guess. Read and overwrite if IF_RESTART is not equal to zero.

IDUMY   Dummy variable, no longer used.

IBOUND   Used to turn off the viscous shear on one or both endwall boundaries.
IBOUND = 0 gives viscous shear on both end walls,
= 1 gives no shear on the hub,
= 2 gives no shear on the casing,
= 3 gives no shear on either end wall.

IF_REPEAT   = 0 for standard solution
= 1 to use the option to change the inlet boundary conditions to satisfy the repeating stage condition. See Card 46F .

CARD 3A   Only needed if ITIMST =   4 or -4

F1, F2eff, F3, RSMTH, NRSMTH
Free Format

F1   The first coefficient of the SSS scheme.
Usual value = 2.0

F2eff   The effective second coefficient of the
SSS scheme. Usual value = -1.0

F3   The third coefficient of the SSS scheme.
Usual value = -0.65 .

RSMTH   The smoothing factor for the residual
averaging. Usual value = 0.4.

NRSMTH The number of times the residual
averaging is called . Usual value = 1.

Card 4   IR, JR, KR, IRBB, JRBB, KRBB
                    Format (6 I5)

IR	Multigrid block size in the I direction for the first level of multigrid.
JR	Multigrid block size in the J direction for the first level of multigrid.
KR	Multigrid block size in the K direction for the first level of multigrid.
IRBB	Multigrid block size in the I direction for the second level of multigrid.
JRBB	Multigrid block size in the J direction for the second level of multigrid.
KRBB	Multigrid block size in the K direction for the second level of multigrid.

Typically IR, JR, KR   all = 3 and IRBB, JRBB, KRBB all = 9.

See Note 14 for details.

NOTE: The block sizes for the third level of multigrid, which is a one-dimensional calculation, are generated automatically.

IMPORTANT   NOTE

 REPEAT CARDS 5-23 FOR EVERY ONE OF 'NROWS' BLADE ROWS.
(see Note 15).

****
CARD 5   ROW TITLE .
                      Free Format

                      Any title for the current blade row.
                      This is used only to help to space out the data for each
                      blade row.

CARD 6   JMROW, JLEROW, JTEROW, NBLROW, KTIPS, KTIPE, JROTHS,
                      JROTHE, JROTTS, JROTTE, NEWGRD, JTRANS, JTRANSP,
                      JTRANH, JTRANT, IFCUSP
                      Format (16 I5)

              JMROW   Number of streamwise (J) grid points
                           on the current blade row.
              JLEROW   J value of the first grid point on the blade. i.e. of
                          the leading edge point.
              JTEROW   J value of the last grid point on the blade. i.e. of
                          the trailing edge point.
              NBLROW   Number of blades in the current blade row.
              KTIPS   K value of the point where the tip clearance starts.
                          i.e. = 1 for hub clearance, = last K point on the
                          blade for casing clearance.
                          Set KTIPS = 0 for no hub or casing
                          clearance.
                          Set KTIPS = -1 for a blade with a shrouded
                          tip seal.
                          See CARDS 59 to 61.

KTIPE	K value of the point where the tip clearance ends. Set = KM for casing clearance . Set = the K value of the first point on the blade for hub clearance. **Set KTIPE = 0 for no tip clearance**
JROTHS	J value of point where the hub starts to rotate at RPMHUB.
JROTHE	J value of point where the hub stops rotating at RPMHUB .
JROTTS	J value of point where the casing starts to rotate at the RPMROW of this blade row.
JROTTE	J value of point where the casing stops rotating at the RPMROW of this blade row.
NEWGRD	Call subroutine NEWGRID to interpolate in the initial grid and set up a new grid if NEWGRD is not = 0. **See cards 22-26.**
JTRAN_I1	J value beyond which the transition condition is applied on the lower (I=1) blade surface.
JTRAN_IM	J value beyond which the transition condition is applied on the upper (I=IM) blade surface.
JTRAN_K1	J value beyond which the transition condition is applied on the hub.
JTRAN_KM	J value beyond which the transition condition is applied on the casing.
IFCUSP	If IFCUSP = 0 the cusp generation is the default which is a cusp 2 grid spacings in length centred on an extrapolation of the blade centre line. If IFCUSP is not zero read in details of the cusp from the next card but one.

For J less than the above values of JTRAN_I1, etc, the boundary layers will be laminar. For J values greater than the values above the transition criterion FTRANS (Card 43) will be applied.

CARD 6AA   Only used if NOSECT, CARD 2, is negative.

                    NSEC_NOW
                    Free Format

       NSEC_NOW   The number of stream surfaces used for blade geometry input on this blade row.

IF IFCUSP = 1 The size and length of the cusp can be varied.Read in:

CARD 6A   This is only used if IFCUSP in Card 6 = 1.

                ICUSP(NR), LCUSP(NR), LCUSPUP(NR)
                Free Format.

      ICUSP(NR)   If ICUSP = 0 the cusp is centred on an extrapolation of the blade centre line. If ICUSP = 1 the cusp is formed with the I=1 surface of the cusp continuous with that of the blade I=1 surface. If ICUSP = -1 it is formed with the I=IM surface continuous.

      LCUSP(NR)   Is the length of the cusp in terms of number of J cells. The usual value is LCUSP = 2.

      LCUSPUP(NR)   Is the number of grid points upstream of the trailing edge where the cusp starts. Typically = 0. If LCUSPUP = LCUSP the cusp is generated upstream of the trailing edge and is treated as part of the blade, i.e. it carries load.

IF IFCUSP = 2. NO cusp is used and instead a body force is used to make the flow separate at a thick trailing edge. This should be used instead of a cusp for a blade with a thick trailing edge. In which case read in:

CARD 6AB
NUP_I1, NUP_IM, N_WAKE, SEP_THICK, SEP_DRAG
Free Format

NUP_I1   The body forces starts NUP_I1 points upstream of the trailing edge on the I=1 blade surface.

NUP_IM   The body forces starts NUP_IM points upstream of the trailing edge on the I=IM blade surface.

N_WAKE   The body force extends N_WAKE grid points downstream of the trailing edge. N_WAKE may be negative.

SEP_THICK   The body force is applied to all grid points within a distance SEP_THICK*(Local blade thickness) of the extrapolated blade surface. Typical value = 0.01. The value can be made negative to increase the pitchwise extent of the body force field.

SEP_DRAG   The magnitude of the body force is proportional to (1-SEP_DRAG). Typical value = 0.99. Reducing this value increases the body force.

See Note 21 for more details.

<u>CARD6B</u>

RPMROW, PUPROW, PLEROW, PTEROW, PDNROW,
FRACTIP, RPMHUB
Format (7 F10.5)

RPMROW — Rotational speed of present blade row in rpm. Positive in the direction of increasing theta coordinate.

WARNING! RPMROW may need to be negative.

PUPROW — **Initial Guess** of the static pressure at mid-span at the upstream boundary to the current blade row, in N/m^2.

PLEROW — **Initial Guess** of the static pressure at mid-span at the leading edge of the current blade row, in N/m^2.

PTEROW — **Initial Guess** of the static pressure at mid-span at the trailing edge of the current blade row, in N/m^2.

PDNROW — **Initial Guess** of the static pressure at mid-span at the downstream boundary to the current blade row, in N/m^2.

FRACTIP — The tip clearance of this blade row as a fraction of the QO span. Use an average value if this varies through the row. The grid spacing at the tip is automatically adjusted to give this value of gap. If FRACTIP is negative different values at the leading and trailing edges can be input in the next card.
This is not used for shrouded blades

RPMHUB — The rotational speed of the hub of the present blade row in RPM. Positive in the direction of increasing theta coordinate.

WARNING ! RPMHUB may need to be negative.

CARD 7   This Card is only needed if FRACTIP in the last card was negative

　　　　　　　　FRACTIP1,FRACTIP2
　　　　　　　　Free Format

　　　　　　　　FRACTIP1　　　　　　The tip clearance at the blade leading edge as a fraction of the local span.

　　　　　　　　FRACTIP2　　　　　　The tip clearance at the blade trailing edge as a fraction of the local span.

CARD 8 　　THIS CARD IS ONLY NEEDED IF KTIPS IS NOT = 0.

　　　　　　　FTHICK(NR,K) K=1,KM
　　　　　　　Format (8F10.5)

　　　　　　　FTHICK(NR,K)　　　　Is a multiplying factor on the blade thickness as input via RT_THICK,J,K) It is mainly used to reduce the blade thickness to zero in the tip region and to thin it towards the tip. Set FTHICK = 0 for the last point on the blade and above the tip gap.

IMPORTANT NOTE:

REPEAT CARDS 9 TO 21 FOR EACH OF "NOSECT" INPUT SECTIONS OF THE CURRENT BLADE ROW

IF KM= 2, FOR A QUASI-3D CALCULATION, THEN ONLY A SINGLE BLADE SECTION MUST BE INPUT.

Card 9   FAC1, XSHIFT, IF_DESIGN, IF_RESTAGGER, IF_LEAN
                           Format (5 F10.5)

FAC1 =   Is the multiplying factor on the input axial
                    coordinates.

XSHIFT   Is the shift of origin for the x coordinate. Note that
                    the shift is done before multiplying by FAC1.

IF_DESIGN   A new blade section is to be designed for this
                    section using data input at the end of the data
                    description.

IF_RESTAGGER   The blade section is to be restaggered by an angle
                    input in Card 17A.

IF_LEAN   The blade section is to be leaned relative to the
                    hub section by an angle to be input in Card 17B .

****
DO NOT READ IN CARDS 10 to 16 IF "IF_DESIGN" IS NOT
= 0 AS A NEW BLADE SECTION WILL BE DESIGNED.
****

Card 10   XSURF(J,K), J = I, JM
Format (8F10.5)

XSURF(J,K)   is the axial coordinate of the J'th point on the K'th input section. In <u>metres</u>.

Card 11   FAC2, TSHIFT
Format (4 F10.5)

FAC2   is the multiplying factor on the input circumferential coordinates.

TSHIFT   is the origin shift for circumferential coordinates. (Note that the shift is made before multiplying by FAC2).

Card 12   RT_UPP(J,K), J = 1, JM
Format (8F10.5)

RT_UPP,J,K)   Is the circumferential coordinate ($r\theta$) of the grid points on the blade surface <u>**with the largest value of θ**</u> (i.e. on the side of the blade to blade passage with the lowest value of theta) at the J'th point on the K'th input section. The origin of θ is not important.   <u>In metres</u>.

<u>Card 13</u>   FAC3, BETAUP, BETADWN1,BETADWN2
                        Format (4 F10.5)

    FAC3   Is the multiplying factor on the input blade
                        tangential thicknesses.

    BETAUP   Is the grid angle (in degrees) that can be set
                        upstream of the leading edge. **See note 19.**

    BETADWN1   Is the grid angle (in degrees) that can be set
                        immediately downstream of the trailing edge. **See
                        note 19.**

    BETADWN2   Is the grid angle (in degrees) that can be set at the
                        end of the grid on the current blade row . **See
                        note 19.**

<u>Card 14</u>   RT_THICK,J,K), J = 1,JM
                        Format (8F10.5)

    RT_THICKJ,K)   Is the <u>tangential</u> thickness ($\Delta r\theta$) of the blade at
                        the J'th grid point on the K'th input section. (set
                        the thickness = 0.0 off the blade). In metres.
                        It is scaled by FTHICK(K) and by FAC3.

<u>Card 15</u>   FAC4, RSHIFT
                        Format (2 F10.5)

    FAC4   is the multiplying factor on the input radii.

    RSHIFT   is the shift of the origin for radial coordinates. Note
                        the coordinates are shifted before scaling by FAC4.

Card 16   <u>This card is not needed if INSURF = 0</u>

RSURF(J,K), J = 1,JM
Format (8F10.5)

RSURF(J,K)   Is the radius of the J'th point on the K'th input
section. <u>In metres</u>.

Card 17   <u>This card is only needed if INSURF = 0</u>

RCYL(K)
Format (F10.5)

RCYL(K)   Is the radius of the K'th cylindrical surface used for
data input if INSURF = 0. <u>In metres</u>.

```
***********************************************************************
```
RE ENTER HERE IF IF_DESIGN WAS NOT ZERO
```
***********************************************************************
```


```
***********************************************************************
```
IF "IF_REDESIGN" IS NOT ZERO THEN INSERT CARDS IFDES1 TO IFDES5 HERE FOR THE RE_DESIGN OPTION GIVEN AT THE END OF THIS DATA DESCRIPTION.
```
***********************************************************************
```


If "IF_RESTAGGER" was not zero then insert the following cards to restagger the blade section

CARD 17AA   COMMENT CARD

CARD 17A

ROTATE, FRACX_ROT
Free Format

ROTATE   The angle of clockwise rotation of the blade section, in degrees.

FRACX_ROT The centre of rotation as a fraction of the axial chord. Usually = 0.5

Note that this option changes all the local blade angles on the stream surface by ROTATE degrees whilst keeping the stream surface geometry unchanged. It can be used for both axial and radial flow machines

Defaults are ROTATE = 0.0 and FRACX_ROT = 0.5.
```
***********************************************************************
```

If "IF_LEAN" was not zero then insert the following card to lean the blade.

CARD 17BB   COMMENT CARD

CARD 17B

ANGLEAN
Free Format

ANGLEAN   The angle by which this blade section is leaned relative to the first section. In degrees. Positive if the lean is in the positive theta direction.

Default is ANGLEAN = 0.0
****

IF KM = 2 input the following data, Cards 17CC to 17F, for a quasi-3D blade-to-blade calculation. Note that in this case the blade geometry must be input on only a single stream surface.

CARD 17 CC   COMMENT CARD

CARD 17C   Q3DFORCE
                                     Free Format

                     Q3DFORCE The is the scaling function on the pressure changes used to keep the flow on the stream surface. Standard value = 1.0 but higher values are usually stable and may give faster convergence.

CARD 17D   NSS
                                     Free Format
                     NSS   The number of points in the following table of stream surface distance and thickness.

CARD 17E   FRACSS(N), N= 1,NSS
                                     Free Format

                     FRACSS   Is the fraction of the meridional distance along the stream surface

CARD 17F   TKSS(N), N=1,NSS
                                     Free Format

                     TKSS   Is the relative stream surface thickness at FRACSS, i.e. $TKSS/TKSS_{ref}$. Only the relative values are used, the absolute value is not important.

END OF KM= 2, Q3D STREAM SURFACE, DATA

IF IM = 2, So that a throughflow calculation is being performed, input the following data to specify the exit flow angle from the current blade row. The flow angle is positive if the exit velocity has a positive tangential component. The deviation angle is always positive.

CARD 18AA   COMMENT CARD

CARD18A   ANGL_TYP, NANGLES
                     Free Format

              ANGL_TYP   The blade exit angle is specified as a flow angle if ANGL_TYP = 'A', as a deviation angle if ANGL_TYP = 'D'.

              NANGLES   The number of spanwise positions at which the exit angle will be specified.

CARD 18B   FRAC_SPAN(N), N=1,NANGLES
                     Free Format

              FRAC_SPAN   A table of the fraction of span at which the angles will be specified.

CARD 18C   EXIT_ANGL(N), N=1,NANGLES
                     Free Format

              EXIT_ANGL   A table of the exit flow angle or deviation angle at FRAC_SPAN. In degrees.

NOTE Cards 18 - 21 give the coordinates of the hub and casing. They are not needed if INSURF = 2 which is the most usual option

<u>Card 18</u>   XH(J), J = 1,JM
                                 Format (8F10.5)

   XH(J)   Is the axial coordinate of the hub at the J'th grid point. (It is scaled by FAC1 and moved by XSHIFT). <u>In metres.</u>

<u>Card 19</u>   RH(J), J = 1, JM
                                 Format (8F10.5)

   RH(J)   is the radius of the hub at the J'th point.
                                 (It is scaled by FAC4 and moved by RSHIFT).
                                 <u>Metres.</u>

<u>Card 20</u>   XT(J), J = 1, JM
                                 Format (8F10.5)

   XT(J)   is the axial coordinate of the casing at the J'th grid point. (It is scaled by FAC1and moved by XSHIFT). <u>Metres.</u>

<u>Card 21</u>   RT(J), J = 1,JM
                                 Format (8F10.5)

   RT(J)   is the radial coordinate of the casing at the J'th grid point.(It is scaled by FAC4 and moved by RSHIFT).   <u>Metres.</u>

REMINDER   Cards 18 - 21 were not needed if "INSURF" was = 2
****

THIS IS THE END OF THE BLADE GEOMETRY DATA INPUT FOR THE
CURRENT BLADE SECTION.

RETURN TO CARD 9 FOR THE NEXT SECTION UNLESS THIS IS THE LAST
SECTION ON THE CURRENT BLADE ROW.
IF KM = 2 THEN ONLY ONE SECTION IS INPUT.
****

****
THE FOLLOWING CARDS (22-26) ARE ONLY NEEDED IF 'NEWGRD' IN
CARD 6 IS NOT ZERO. NEW GRID SPACINGS IN THE STREAMWISE (J)
DIRECTION ARE GENERATED FOR THE CURRENT BLADE ROW IF "NEWGRD"
IS NOT ZERO.
****

CARD 22   NUP, NON, NDOWN
                          Free Format

        NUP   Is the number of grid points upstream of the blade
                          required in the new grid. Set = 0 to generate the
                          grid automatically as in CARD 23A

        NON   Is the number of grid points on the blade in the
                          new grid. Set = 0 to generate the grid
                          automatically as in CARD 24A.

        NDOWN   Is the number of grid points downstream of the
                          blade in the new grid. Set = 0 to generate the grid
                          automatically as in CARD 25A.

IF NUP IS NOT ZERO USE THE FOLLOWING CARD

CARD 23   UPF(J),J=1,NUP
         Free Format

  UPF(J)   Is the **relative** spacing of the grid points upstream
         of the blade in order of increasing J value.

IF NUP IS ZERO USE THE FOLLOWING TWO CARDS

CARD 23 A   NUP
         Free Format

  NUP   is the number of grid points required upstream of
         the blade row.

CARDS 23 B   XFRACUP, RELSPUP
         Free Format.

  XFRACUP   = Fraction of axial upstream distance
  RELSPUP   = Relative grid spacing at XFRACUP

Continue inputting a table of values of XFRACUP and RELSPUP until a value
XFRACUP greater than 1.0 is found. A minimum of 2 cards are needed.

IF NON IS NOT ZERO USE THE FOLLOWING CARD

CARD 24   ONF(J),J=1,NON
         Free Format

  ONF(J)   Is the **relative** spacing of the grid points on the
         blade in order of increasing J value.

IF NON IS ZERO USE THE FOLLOWING TWO CARDS

CARD 24 A   NON

                                Free Format

        NON   is the number of grid points required upstream of
                        the blade row.

CARDS 24 B   XFRACON, RELSPON

                                Free Format.

        XFRACON   = Fraction of axial distance on the blade.
        RELSPON   = Relative grid spacing at XFRACON

Continue inputting a table of values of XFRACON and RELSPON until a value
XFRACON greater than 1 is found. A minimum of 2 cards are needed.

IF NDOWN IS NOT ZERO USE THE FOLLOWING CARD

CARD 25   DOWNF(J), J =1,NDOWN

                                Free Format

        DOWNF(J)   Is the **relative** spacing of the grid points
                                downstream of the blade in order of increasing J
                                value.

IF NDOWN IS ZERO USE THE FOLLOWING TWO CARDS

CARD 25 A   NDOWN

                                Free Format

        NDOWN   is the number of grid points required downstream
                                of the blade row.

CARDS 25 B   XFRACDWN, RELSPDWN
                              Free Format.

   XFRACDWN   = Fraction of axial distance downstream of the
                                   blade.
   RELSPDWN   = **Relative** grid spacing at XFRACDWN

Continue inputting a table of values of XFRACDWN and RELSPDWN until a value
XFRACON greater than 1 is found. A minimum of 2 cards are needed.

CARD 26   UPEXT, DWNEXT
                              Free Format

   UPEXT   Multiplies the original length of the grid upstream
                             of the leading edge by UPEXT .

   DWNEXT   Multiplles the original length of the grid
                             downstream of the trailing edge by DWNEXT.

THIS IS THE END OF DATA INPUT ON THE CURRENT BLADE ROW.

RETURN TO CARD 5 TO START THE NEXT BLADE ROW UNLESS THIS IS THE LAST ROW.

IF THE LAST BLADE ROW HAS BEEN COMPLETED. CARDS, 27 - 45 MUST NOW BE INPUT. THESE GIVE DATA FOR THE WHOLE MACHINE.

Card 27   CP, GA, CFL, SFT, SFX, MACHLIM
                            Format (8F10.5) .

CP   =   Gas specific heat at constant pressure,
                            Joules / Kg K. = 1005 for air.
                            Input a negative value to use real gas properties
                            which are then input in the next card.

GA   =   Ratio of specific heats. = 1.4 for air.

CFL   =   Timestep multiplier. Typical value = 0.4 for the
                            Scree scheme, 0.7 for the SSS scheme. **See Note 6.**

SFT   =   Smoothing factor in pitchwise and spanwise
                            directions. Typical value 0.005. **See Note 7.**

SFX   =   Streamwise smoothing factor. Typical value 0.005.
                            See Note 8.

MACHLIM =   A limiting value of Mach number beyond which the
                            relative velocities will be frozen. This helps to limit
                            localised very high velocity spikes.
                            Typical value = 2.0 but reduce this in cases with
                            very high localized Mach numbers.
                            The value defaults to 2.0 if this number is omitted
                            from the data set.

If CP input in card 27 was negative input the following real gas properties.

CARD 27A   CP1, CP2, CP3, TREF, RGAS
                 Free Format

         CP1   The value of CP at TREF in J/kg K

         CP2   The rate of change of CP with temperature at
                     TREF
         CP3   The second derivative of the variation of CP
                     with temperature at TREF.

         TREF   The reference temperature in K .

         RGAS   The constant value of gas constant in J/kg k.

The specific heat is taken to vary as:

$$Cp \;=\; Cp_1 \;+\; Cp_2(T - T_{ref}) \;+\; Cp_3(T - T_{ref})^2$$

For combustion products typical values are TREF = 1400, CP1 = 1272.5, CP2 = 0.2125, CP3 = 0.000015625,   RGAS = 287.5.

CARD 28   DAMP, DUMMY, FBLK1, FBLK2, FBLK3, SFEX, CLIM, RFIN
                        Format (8F 10.5)

DAMP =   Factor controlling the negative feedback. Typical
                        range = 10 -> 25 . **See Note 10.**

DUMMY   = Not used in this version but a number must be
                        input to maintain compatibility with previous
                        versions.

FBLK1   Scaling factor on changes produced by the first
                        level of multigrid. Typical value = 0.40.

FBLK2   Scaling factor on changes produced by the second
                        level of multigrid. Typical value = 0.2

FBLK3   Scaling factor on changes produced by the third
                        level of multigrid. Typical value = 0.1

SFEX   Smoothing factor on the exit flow field. Usually =
                        0.0 but may be set = 0.5 if reverse flow is causing
                        instability at the exit boundary.
                        If SFEX is not zero then NSFEXIT must be input in
                        the next card. **See note 11.**

CLIM   Convergence criterion as the average **percentage**
                        change in the meridional velocity per time step.
                        Usually set = 0.005.

RFIN   Relaxation factor on changes in inlet pressure.
                        Usually set = 0.1 but reduce to 0.02 or less if the
                        inlet flow becomes unstable.

 CARD 28A   Only needed if SFEX in the last card is not zero.

NSFEXIT
Free Format

The exit flow smoothing is applied over NSFEXIT grid points upstream from the exit boundary. Typically use 5 to 10 upstream points with a value of SFEXIT = 0.1.

<u>Card 29</u>   PUPHUB, PUPTIP, PDHUB, PDTIP, THROTLE_PLATE,
                              THROTTLE_ALL, DUMMY, F_PDOWN
                              Format   (8F10.5)

PUPHUB	Initial <u>guess</u> of static pressure on the hub at inlet to the whole machine. in N/m^2
PUPTIP	Initial <u>guess</u> of static pressure at the tip at inlet to the whole machine. In N/m^2
PDHUB	Static pressure on the hub at exit from the whole machine. This is a fixed value if IPOUT = 0, or 1 but only an initial guess if IPOUT = -1 . In N/m^2
PDTIP	Static pressure at the tip at exit from the whole machine. This is only an initial guess if IPOUT = 0, but a fixed value if IPOUT = 1 or = -1 .In N/m^2
THROTLE_PLATE	If this is zero this has no effect. If it is greater than zero then the downstream boundary behaves as a perforated plate with loss coefficient = THROTLE_PLATE. This can be used to prevent reverse flow crossing the downstream boundary. A typical value of THROTLE_PLATE, if used, is 2
THROTTLE_ALL	This can be used to set a throttle boundary condition for the whole flow. The exit pressure is made to vary parabolically with the exit mass flow rate. If it is non-zero then extra data is read in in the next card. Set = 0.0 if no throttle is to be used.
DUMMY	No longer used but a value must be input.
FP_DOWN	Allows a downwinding of the pressure used in the momentum equations. This may be used to increase the shock smearing and reduce the undershoots and overshoots in pressure at a shock wave. Set = 0.1 - 0.2 to use this option. Set = zero if not used, as is usual in all subsonic flows.

Note that the above values of exit pressure are not used if IPOUT = 3 when values in card 31 are used instead.

CARD 29A Only needed in THROTTLE_ALL is non-zero.

THROTTLE_PRES, THROTTLE_MAS, RFTHROTL
Free Format

THROTTLE_PRES The required value of exit static pressure in N/m**2

THROTTLE_MAS The expected value of exit mass flow rate in kg/sec.

RFTHROTL A relaxation factor on changes in exit pressure, typical value = 0.1   .

****

CARDS 30 to 36   give data on the inlet and exit flow on KIN stream surfaces whose relative spacing is given by FR(K) in Card 34 . IF KIN = KM then these are the same as the streamwise surfaces of the computational grid. If KIN is not equal to KM then the boundary conditions are generated by interpolating in this data.

Card 30 PO1(K), K = 1,KIN
Format (8F10.5)

PO1(K) = Value of the inlet **absolute** inlet stagnation pressure (in N/m^2) at inlet on the K'th input surface at J=1.

Card 31   ONLY NEEDED IF IPOUT =3.

PD(K), K=1,KIN
    Format (8F10.5)

PD(K) =   The downstream static pressure on the K'th input
                         surface in N/m^2. This is held constant If IPOUT =
                         3. Do not input this card if IPOUT is not = 3.

Card 32   TO1(K), K = 1, KIN
                Format (8F10.5)

TO1(K) =   Value of the **absolute** inlet stagnation
                         temperature (in degrees K) at inlet on the K'th
                         input surface at J = 1.

Card 33   VTIN(K), K = 1, KIN
                Format (8F10.5)

VTIN(K) =   **Absolute** tangential velocity at inlet (in _m/sec)_
                         on the K'th input surface.
                         _Only an initial guess unless IN_VTAN =1_

Card 34   VM1(K), K = 1, KIN
                Format (8F10.5)

VM1(K) =   Initial guess of the meridional velocity at inlet on
                         the K'th input surface. _This is only an initial guess
                         but should be as accurate as possible as it fixes the
                         initial mass flow rate_ . m/sec.

<u>Card 35</u>   BS(K), K = 1, KIN
                               Format (8F10.5)

     BS(K) =   Absolute or relative swirl angle (in degrees) at inlet
                               depending on the value of IN_VTAN .
                               $= \tan^{-1} (V_\theta /V_m)$ if IN_VTAN = 0.

                               $= \tan^{-1} (W_\theta/V_m)$ if IN_VTAN = 2.

                               Not used if IN_VTAN = 1 but values must still be
                               input.

<u>Card 36</u>   BR(K), K = 1, KIN
                               Format (8F10.5)

     BR(K) =   Meridional pitch angle at inlet .
                               (i.e. $\tan^{-1} (V_r / V_x)$) in <u>degrees</u>.

                               This is held constant if IN_VR = 1, 3, or 4 .
                               <u>But is only an initial guess if IN_VR = 0.</u>

<u>Card 37</u>   FR(K), K = 1, KIN-1
                               Format (8F10.5)

     FR(K) =   <u>Relative</u> spacing of the boundary condition **input**
                               surfaces in the spanwise direction. This is also
                               taken to be the spacing of the grid surfaces if KIN
                               = KM.

                               See Note 12 which also describes how the
                               spacing can be generated automatically within the
                               program.

Card 38 FP(I), I = 1, IM-1
Format (8F10.5)

FP(I) = <u>Relative</u> spacing of the grid points in the pitchwise direction.

See Note 12 which also describes how the spacing can be generated automatically within the program.

Card 39 IPRINT(L), L=1,5
Format (5 I5)

IPRINT(L) = Five values of the time step number on which a full printout and plot file will be produced as determined by IOUT and KOUT.
Note IPRINT(L) must be a multiple of 5.
IPRINT(1) = 0 gives a printout and plot file of the initial guess.

Card 40 IOUT(I), I = 1, 13
Format (13 I2)

IOUT(I) Determines which flow variables are printed out to UNIT 6 at the end of a run or as determined by IPRINT. It also determines whether output is printed at every pitchwise point or only a pitchwise mass average value is output. **Usually set = 0 as this is very seldom used. See Note 13.**

Card 41   KOUT(K), K = 1, KM
                                    Format (40 I2)

        KOUT(K)   Determines on which streamwise (K) surfaces
                            output is printed to UNIT 6.   KOUT(K) = 0 gives no
                            output on that K line. If KOUT(K) = 1 then output
                            is printed on the K line as determined by IPRINT
                            and IOUT. Usually set = 0 as this is very
                            seldom used. See note 13.

CARD 42

        This card is not needed unless IN_FLOW = 2 or 3. See Note
 17.

                        FLOWIN, RFLOW
                                Format(2F10.3)

        FLOWIN =   Required total mass flow for the whole annulus
                            in Kg/sec. This is only used if IN_FLOW=3.

        RFLOW   =   Relaxation factor on mass flow forcing, typically
                            0.10. Used if IN_FLOW= 2 or = 3.

<u>CARD 43</u>   REYNO, REL, FTRANS, FACSEC, TURBVIS_LIM, PRANDTL,
                    YPLUSWALL
                              Format(8F10.5)

REYNO   Reynolds number based on the exit velocity at mid-
                    span and the meridional chord **of the <u>first blade
                    row</u>**. If the value input is less than 100 then the
                    dynamic viscosity is taken as REYNO x10^{-5}.
                    If the value input is negative then its absolute
                    value is the (dynamic viscosity at temperature
                    288K) x 10^5 . This viscosity is then automatically
                    varied with the local temperature. For air a typical
                    input value would be -1.9 .

REL   Relaxation factor on changes in the viscous source
                    terms. Typical value = 0.50.

FTRANS   Is a simple boundary layer transition criterion as
                    suggested by Baldwin-Lomax. A low FTRANS, e.g.
                    = 0.0001, gives a fully turbulent boundary layer. A
                    high value, e.g. 10000. , gives a fully laminar one .
                    The typical value for correct transition = about 15
                    but this is not considered to be a very
                    reliable transition criterion.
                    Note FTRANS should be used in conjunction with
                    JTRANS, CARD 43

FACSEC   Proportion of the fourth order smoothing. Typical
                    value = 0.8 .

TURBVIS_LIM   The limiting value of turbulent viscosity as a
                    multiple of the local laminar viscosity. Typical value
                    = 1000.

PRANDTL   The Prandtl number of the fluid. Usually set = 1 for
                    turbulent flow.

YPLUSWALL   Is the value of Y^+ used to calculate the wall skin friction. The first grid point is taken to be in the laminar sub-layer at this value of Y^+ and this is used to obtain the skin friction. A typical value of YPLUSWALL, if it is to be used, is 11.

Set this = 0.0 to use the original wall functions which are considered more reliable.

Set YPLUSWALL to -5.0 to use the Shih et al wall functions with only the velocity terms. Set = -15.0 to use both the velocity and pressure terms.

CARD 43A

Only needed if the Spalart-Allmaras turbulence model is used. i.e. if ILOS = 200. Defaults are used if this card replaced by a Comment statement e.g. "C" .

FAC_STMIX, FAC_ST0, FAC_ST1, FAC_ST2, FAC_ST3, FAC_SFVIS, FAC_VORT, FAC_PGRAD
Free Format

FAC_STMIX	The scaling factor on the mixing length model source term. Usually set = 0
FAC_ST0	The scaling factor on the main SA generation source term. Usually set = 1.0
FAC_ST1	The scaling factor on the first SA diffusion term. Usually set = 1.0
FAC_ST2	The scaling factor on the second SA diffusion term. Usually set = 1.0
FAC_ST3	The scaling factor on the first SA destruction term. Usually set = 1.0

FAC_SFVIS   Is used to increase the smoothing factor for the turbulent viscosity in the SA model. The value of SFX is multiplied by FAC_SFVIS. The usual value is 2.0 but increase it to increase the stability of the SA model which is tends to be less robust than the mixing length model.

FAC_VORT   This is new to version 17.5. It allows the main source term for turbulent viscosity to be increased by streamwise vorticity as suggested by Lee et al in ASME GT2017-63245 . The increase is limited to (1 + FAC_VORT) x the original source term. Lee et al suggest a value = 0.9191 for FAC_VORT. The default value = 1.0 . So far there is little experience of using this option.

FAC_PGRAD   This is also new to version 17.5. The main source term for turbulent viscosity is increased by adverse pressure gradients, again as suggested by Lee et al. The increase is limit to (1 + FAC_PGRAD) x the original term. Lee et al suggest a value of 0.6565 for FAC_PGRAD. The default value = 1.0 . So far there is little experience of using this option. Both FAC_VORT and FAC_PGRAD can be used together to increase the source term by their product.

CARD 44 This card is only needed if IFMIX is not = zero.

        RFMIX, FEXTRAP, FSMTHB, FANGLE
        Free Format

        RFMIX   Is a relaxation factor on the flow properties calculated from an isentropic expansion at the mixing plane. Usually use   RFMIX = 0.025 but

reduce to 0.01 if there is any instability at the mixing plane.

FEXTRAP Is a scaling factor on the extrapolation of the fluxes at the mixing plane and also of the circumferential variations at the downstream boundary. The amplitude of flux variation at the mixing plane is FEXTRAP x the amplitude at the next grid point upstream. A typical value = 0.9. Default value = 0.9. Use larger values if the grid spacing is small and vice-versa. See note 22.

FSMTHB Is a factor multiplying the value of SFX (the axial smoothing factor) used at the mixing plane. It is not usually necessary to increase the smoothing there so the typical value is 1.0 but increase this if there are signs of instability at the mixing plane.

FANGLE The pitchwise flow angle on the downstream side of the mixing plane is taken as FANGLE x (the average flow angle from downstream) + (1-FANGLE) x (the flow angle on the mixing plane). See note 22.
Typical value =0.9, but increase this if the grid points are closely spaced and reduce it if they are widely spaced.

THE NEXT CARD IS ONLY NEEDED IF KIN IS NOT = KM OR IF KIN = -KM

<u>CARD 45</u>   FRNEW(K), K=1,KM-1
                            Format (8F10.3).

FRNEW(K)   is the <u>relative</u> spacing of the <u>**grid** points </u>in the
                        spanwise, K, direction. This is the actual grid point
                        spacing whilst FR(K), card 37, is the inlet boundary
                        condition spacing. These are taken to be the same
                        if KIN = KM.
                        An option to generate the spacing automatically is
                        described in **Note12.**

If ILOS is not zero then the following data must be input after CARD 45.

CARD 46   Only used if ILOS is 10 to use the original mixing length model.

One card for each blade row.

For each blade row input the following mixing length limits.

XLLIM_I1, XLLIM_IM, XLLIM_K1, XLLIM_KM, XLLIM_DWN, XLLIM_UP
Free Format

XLLIM_I1	Is the mixing length limit on the I=1 blade surface as a fraction of the mid-span pitch. Typical value = 0.03 .
XLLIM_IM	Is the mixing length limit on the I=IM blade surface as a fraction of the mid-span pitch. Typical value = 0.03 .
XLLIM_K1	Is the mixing length limit on the hub (K=1) as a fraction of the mid-span pitch. Typical value = 0.03
XLLIM_KM	Is the mixing length limit on the casing (K=KM) as a fraction of the mid-span pitch. Typical value = 0.03
XLLIM_DWN	Is the mixing length limit downstream of the blade row as a fraction of the mid-span pitch. Typical value = 0.03 .
XLLIM_UP	Is the mixing length limit upstream of the leading edge as a fraction of the mid-span pitch. Typical value = 0.03 .

CARD 46 A   Only used if the NEW_LOSS or SPAL_LOSS routines are used, i.e. if ILOS = 100 or ILOS = 200.

For each blade row input the following mixing length limits.

XLLIM_IN, XLLIM_LE, XLLIM_TE, XLLIM_DN, FSTURB, TURBVIS_DAMP
Free Format

XLLIM_IN   The mixing length limit at the upstream boundary of the blade row. Typical value = 0.02.

XLLIM_LE   The mixing length limit at the leading edge of the blade row. Typical value = 0.03.

XLLIM_TE   The mixing length limit at the trailing edge of the blade row. Typical value = 0.04.

XLLIM_DN   The mixing length limit at the downstream boundary of the blade row. Typical value = 0.04 .

FSTURB   The free stream turbulent viscosity in the blade row as a multiple of the local laminar viscosity. e.g. 2.0 .

TURBVIS_DAMP
            The fraction of the pitchwise average turbulent viscosity which is transferred across the mixing plane downstream of the blade row. Typical value = 0.5.

CARD 46B   FACMIXUP, NMIXUP
                             Free Format

FACMIXUP   The mixing lengths in CARD 46 are initially multiplied
                  by FACMIXUP. They are then gradually reduced to the
                  input values over the first NMIXUP time steps. This is
                  done to prevent stall of compressor stages during the
                  initial transient. Typically FACMIXUP = 2.0

NMIXUP   The mixing length increase is reduced to zero over the
                  first NMIXUP time steps. Typically set NMIXUP = 1 for
                  turbines but = NMAX/2 for compressors.

CARD 46C   IF IF_ROUGH = 1 Then, for each of NROWS blade rows input

ROUGH_H(N), ROUGH_T(N), ROUGH_L(N), ROUGH_U(N)
Free Format

ROUGH_H(N)   The roughness on the hub. Actual roughness height in microns.

ROUGH_T(N)   The roughness on the casing. Actual roughness height in microns.

ROUGH_L(N)   The roughness on the lower, $I=1$, blade surface. Actual roughness height in microns.

ROUGH_U(N)   The roughness on the upper, $I = IM$, blade surface . Actual roughness height in microns.

IF ITIMST = 5 or -5 input the following Card.

        CARD 46D   VSOUND, RF_PTRU, RF_VSOUND, VS_VMAX
                                   Free Format

           VSOUND   An initial guess of the artificial speed of sound. It should be about twice the expected maximum relative velocity. It is only an initial guess and is automatically changed by the VS_VMAX ratio during the calculation. Default = 150 m/s .

           RF_PTRU   A relaxation factor on the changes in the calculated pressure. Typical value = 0.01 .

           RF_VSOUND A relaxation factor on the changes in the calculated artificial speed of sound. Typical value = 0.002 .

           VS_VMAX   The ratio of the calculated sound speed to the calculated maximum flow speed. Typical value = 2.0 .

IF ITIMST = 6 or -6 input the following Card

        CARD 46E   VSOUND, RF_PTRU, RF_VSOUND, VS_VMAX ,
                                   DENSTY
                                   Free Format

             VSOUND, RFP_TRU, RF_VSOUND, VS_VMAX
                          All as in the previous card.

           DENSTY   The fixed value of density to be used for fully incompressible flow. Kg/m^3

If IF_REPEAT Card 3, is not zero input the following card to use the repeating stage option.

CARD 46F   NINMOD, RFINBC
                         Free Format

    NINMOD   The inlet boundary conditions are updated every
                         NINMOD time steps. Typical value = 10 .

    RFINBC   The relaxation factor on the change in inlet boundary
                         conditions. Typical value = 0.025. Reduce if any signs
                         of instability.

CARD 46G   NSTAGE(N), N= 1,NROWS
                         Free Format

                         NSTAGE(N) is the stage number to which blade row
                         number N belongs.

    e.g.   For a 2 stage machine with one stator and one rotor in each
                         stage   input   1 1 2 2 .   The default is the each stage
                         has two blade rows.

IF IM = 2 so performing a throughflow calculation input the following data to specify the rate of updating of the pressure distribution used to keep the flow on the stream surface and the smoothing of the blade surface pressure distribution.

CARD 47A   COMMENT CARD

CARD 47   Q3DFORCE, SFPBLD, NSFPBLD
         Free Format

        Q3DFORCE   The scaling factor on the change in pressure distribution. Standard value = 1.0 but larger values are usually stable and give faster convergence.

        SFPBLD   The smoothing factor applied to the calculated blade pressure distribution. Standard value = 1.0 . Larger values are more stable but may make the flow deviate from the specified stream surface.

        NSFPBLD   The number of times that smoothing is applied on each time step. Standard value = 2. Higher values are more stable but may make the flow deviate from the specified stream surface.

CARD 47A   YPLAM, YPTURB
                           Free Format

                YPLAM   The value of yplus below which the
                                     turbulent viscosity is set to zero

                YPTURB   The value of yplus above which the
                                     turbulent viscosity is not changed.

Typical values are YPLAM = 7.5, YPTURB = 25.0

To use the calculated value of turbulent viscosity without any reduction in the sublayer or buffer layer set YPLAM = 0.01, YPTURB = 0.02. The values must not be equal otherwise an error will occur.

This card must always be input.

THE FOLLOWING DATA, CARDS 48 TO 55, ARE NEEEDED IF
ANY BLADE ROW HAS COOLING FLOWS ADDED THROUGH THE
BLADE OR ENDWALL SURFACES, i.e. WHEN "IFCOOL", AS INPUT
IN CARD 2, IS NOT ZERO.

THIS DATA IS READ IN BY SUBROUTINE "COOLIN" JUST BEFORE
STARTING THE MAIN TIME STEPPING LOOP.

For every blade row (index NR) read in the following cards 48 to 55.

CARD 48   TITLE
                                    Format A72

                                    Any title to identify the blade row in columns 1-72.

CARD 49   NCWLBLADE, NCWLWALL
                                    Free Format

        NCWLBLADE   = Number of cooling patches on the blade suction
                                      and pressure surfaces.
        NCWLWALL   = Number of cooling patches on the hub and
                                      casing.

CARD 50   TITLE
                                    Format A72

                                    Any Title to identify the blade or endwall surface
                                    in columns 1-72

INPUT CARDS 51 to 52 FOR EVERY ONE OF THE "NCWLBLADE" COOLING PATCHES ON THE BLADE SURFACES OF THE CURRENT BLADE ROW

<u>CARD 51</u>   IC, JCBS, JCBE, KCBS, KCBE
                       Free Format

    IC   = I value of the blade surface on which the patch is located. Must be either I = 1 or IM.

    JCBS   = J value of the start of the patch, defined relative to the upstream mixing plane of the current blade row.

    JCBE   = J value of the end of the patch, defined relative to the upstream mixing plane of the current blade row.

    KCBS   = K value of the start of the patch.

    KCBE   = K value of the end of the patch.

<u>CARD 52</u>   CFLOWB, TOCOOLB, POCOOLB, MACHCOOL, SANGLEB, XANGLEB, RVT_IN, RPM_COOL
                       Free Format

    CFLOWB   = the mass flow rate of coolant through the current patch. In Kg/sec.

    TOCOOLB   = the **absolute** stagnation temperature at which the coolant is supplied to the blade row. In K.

    POCOOLB   = the **absolute** stagnation pressure at which the coolant is supplied to the blade row. In N/m**2.

MACHCOOL = the **relative** Mach number at which the coolant leaves the blade surface.

SANGLEB = the angle between the coolant jet and the plane which is locally tangent to the blade surface. In degrees. See Note 20

XANGLEB = the angle between the projection of the cooling jet onto the blade surface and a line which is the intersection of the blade surface with a surface of constant radius, i.e. with a cylindrical surface. In degrees. See Note 20

RVT_IN Is the angular momentum (radius x tangential velocity) with which the coolant flow is supplied to the blade row by any pre-swirl system. Set = zero if no pre-swirl system.

RPM_COOL Is the rotational speed , in RPM, of the disc through which the coolant flow is supplied to the blade row. RVT_IN and RPM_COOL are used to find the pumping work on the coolant between its supply condition to the disc and the point where it enters the mainstream. Set = zero except for coolant which is supplied to a rotating blade row or disc.

CARD 53   TITLE
                                Format A72

                                Any Title to identify the blade or endwall surface
                                in columns 1-72

**INPUT CARDS 54 to 55 FOR EVERY ONE OF THE "NCWLWALL"
COOLING PATCHES ON THE ENDWALL SURFACES OF THE
CURRENT BLADE ROW**

CARD 54   KC , JCWS, JCWE, ICWS, ICWE
                                Free Format

    KC   = K value of the blade surface on which the patch
                                is located. Must be either K = 1 or KM.

    CWS   = J value of the start of the patch, defined relative
                                to the upstream mixing plane of the current blade
                                row.

    JCWE   = J value of the end of the patch, defined relative
                                to the upstream mixing plane of the current blade
                                row.

    ICWS   = I value of the start of the patch.

    ICWE   = I value of the end of the patch.

<u>CARD 55</u>   CFLOWW, TOCOOLW, POCOOLW, MACHCOOL, SANGLEW, XANGLEW, RVT_IN, RPM_COOL
    Free Format

CFLOWW   = the mass flow rate of coolant through the current patch. In Kg/sec.

TOCOOLW   = the **absolute** stagnation temperature at which the coolant is supplied to the blade row. In K.

POCOOLW   = the **absolute** stagnation pressure at which the coolant is supplied to the blade row. In N/m**2.

MACHCOOL   = the **relative** Mach number at which the coolant leaves the blade surface.

SANGLEW   = the angle between the coolant jet and the plane which is locally tangent to the endwall surface. In degrees. See Note 20.

XANGLEW   = the angle between the projection of the cooling jet onto the endwall surface and a line which is the intersection of the blade surface with a surface of constant circumferential coordinate, i.e. with the axial-radial plane θ = constant. In degrees. See Note 20.

RVT_IN   Is the angular momentum (radius x tangential velocity) with which the coolant flow is supplied to the blade row by any pre-swirl system. Set = zero if no pre-swirl system.

RPM_COOL   Is the rotational speed , in RPM, of the disc through which the coolant flow is supplied to the blade row. RVT_IN and RPM_COOL are used to find the pumping work on the

coolant between its supply condition to the disc and the point where it enters the mainstream. Set = zero except for coolant which is supplied to a rotating blade row or disc.

RETURN TO CARD 48 TO INPUT COOLING FLOW DATA FOR THE NEXT BLADE ROW

THE FOLLOWING DATA , CARDS 56 TO 58 IS ONLY NEEDED IF THERE ARE ANY BLEED FLOWS IN THE MACHINE. FLOW CAN BE BLED FROM THE HUB AND CASING AND FROM THE BLADE SURFACES. I.E. IF "IFBLEED" IN CARD 2 IS NOT ZERO.

The data is read in by subroutine "BLEEDOUT" just before the start of the main time-stepping loop.

READ IN CARDS 56 TO 58 FOR EVERY BLADE ROW IF "IFBLEED" IS NOT ZERO.

CARD 56
TITLE
Format A72

TITLE   Any characters in columns 1-72 to identify the blade row.

CARD 57
NBLEED
Free Format

NBLEED Is the number of bleed flows on the current blade row.

**READ IN CARD 58 FOR EACH OF THE "NBLEED" BLEED FLOWS
ON THE CURRENT BLADE ROW.**

<u>CARD 58</u>   IBLDS, IBLDE, JBLDS, JBLDE, KBLDS, KBLDE, MASSBLED
                          Free Format

IBLDS   = the I value where the bleed starts.
IBLDE   = the I value where the bleed ends.
JBLDS   = the J value where the bleed starts, defined
                        relative to the start of the current blade row.
JBLDE   = the J value where the bleed ends, defined
                        relative to the start of the current blade row.
KBLDS   = the K value where the bleed starts.
KBLDE   = the K value where the bleed ends.

MASSBLED   = the mass flow bled off in Kg/s.

**RETURN TO CARD 56 TO INPUT THE BLEED FLOW DATA FOR
THE NEXT BLADE ROW.**

THE FOLLOWING DATA, CARDS 59-61, IS ONLY NEEDED FOR
SHROUDED BLADES WITH SHROUD LEAKAGE. FOR EVERY BLADE
ROW FOR WHICH "KTIPS(NR)" WAS SET NEGATIVE IN CARD 6
INPUT THE FOLLOWING 3 CARDS.

The data is read in in by subroutine "SHROUDFLOW' on the first
time it is called from the main time stepping loop.
Note NR is the blade row number.

CARD 59   KSHROUD(NR), JLEAKS(NR), JLEAKE(NR), JLKINS(NR), JLKINE(NR)
                              Free Format

          KSHROUD   = 1 for A hub shroud,
                       = KM for a casing shroud.

          JLEAKS   = J Value where the leakage starts leaving the
                         mainstream flow. (J is defined relative to the start
                         of the current blade row)

          JLEAKE   = J Value where the leakage stops leaving the
                         mainstream flow (J is defined relative to the start
                         of the current blade row)

          JLKINS   = J Value where the leakage starts to re-enter the
                         main flow behind the blade row. (J is defined
                         relative to the start of the current blade row)

          JLKINE   = J Value where the leakage stops entering the
                         main flow behind the blade row. (J is defined
                         relative to the start of the current blade row).

Note that shroud leakage flow from downstream of the blade to upstream, as in
compressors, can now be handled. In this case JLEAKS and JLEAKE should be greater
than JTE, and JLEAKINS and JLEAKINE should be less than JLE.

CARD 60   SEALGAP(NR), NSEAL(NR), CFSHROUD(NR), CFCASING(NR)
                        Free Format

    SEALGAP   = Shroud seal clearance in metres

    NSEAL   = Number of tip seals.

    CFSHROUD   = Skin friction coefficient on the shroud attached
                          to the blade row. A typical value = 0.0025

    CFCASING   = Skin friction coefficient on the hub or casing
                          adjacent to the shroud. A typical value =
                          0.0025

CARD 61   WCASE(NR), PITCHIN(NR)
                        Free Format

    WCASE   = Rotational speed of the hub or casing adjacent to
                          the shroud. This is not the rotational speed of
                          the shroud, which is taken to be the same as
                          that of the blade row. in RPM.
                          Note this may be negative.

    PITCHIN   = Angle in the meridional plane at which the
                          leakage flow re-enters the mainstream. The
                          angle is measured from a tangent to the hub or
                          casing and is always treated as positive. In
                          degrees. Guess = 45deg if not known.

BLADE REDESIGN DATA

****

****

The following cards number IFDES1 to IFDES5 are input if using the blade section redesign option. These are needed for every blade section for which IF_DESIGN was non-zero.

****

****

CARD IFDES1A   **COMMENT CARD**

CARD IFDES1

   N_SS, N_LE, N_TE

N_SS	The number of points used to define the new stream surface. Typically about 8 points are sufficient.
N_LE	The number of the leading edge point in the N_SS input points.
N_TE	The number of the trailing edge point in the N_SS input points.

Note that there must be an input point at the leading edge and one at the trailing edge.

****

CARD IFDES2A   COMMENT CARD

CARD IFDES2

> For N = 1, N_SS input
> XSS(N), RSS(N), RELSPACE(N)

> > XSS   The axial coordinate of the point on the stream surface. In metres.

> > RSS   The radius of the point on the stream surface. In metres.

> > RELSPACE   The relative grid spacing of the final grid points at the point on the stream surface. Only the relative values are needed the absolute values are not used.

Note interpolation between the points is used so that the input points must define a smooth surface and the spacing of the points should note change suddenly.
****

CARD IFDES3A   COMMENT CARD

CARD IFDES3 .

> NNEW, NSMOOTH

> > NNEW   The number of points at which new blade geometry will be input in the next card. Typically 5 to 10 points are sufficient.

> > NSMOOTH   The number of times that the new blade data will be smoothed. Typical value = 2 .

> There are no defaults for this data .
****

CARD IFDES4A   COMMENT CARD

CARD IFDES4

   For N = 1 to NNEW input the following data, i.e. NNEW cards in total.
   FRACNEW(N) ,BETANEW(N), THICKUP(N), THICKLOW(N)

   FRACNEW   The fraction of axial chord at which the blade details are
                 given. The first value must be 0.0 and the last value =
                 1.0

   BETANEW   The blade camber line angle at FRACNEW. In degrees. It
                 is positive if a vector in the direction of the angle would
                 have a positive tangential component.

   THICKUP   The blade tangential thickness above the camber line as
                 a fraction of the axial chord.

   THICKLOW   The blade tangential thickness below the camber line as
                 a fraction of the axial chord.

If THICKUP is not equal to THICKLOW then the original blade centre line is not a true
centre line of the new blade.

The stream surface shape is not changed by this redesign procedure.

There are no defaults for this data.
****

CARD IFDES5A   COMMENT CARD

CARD IFDES5

    FRAC_CHORD_UP, FRAC_CHORD_DWN, RTHETA_MID

        FRAC_CHORD_UP The grid extension upstream of the leading edge as a fraction of the meridional chord.

        FRAC_CHORD_DWN   The grid extension downstream of the trailing edge as a fraction of the meridional chord.

        RTHETA_MID   The tangential coordinate of the mid grid point. This may be used to change the blade stacking but is usually set = 0.0

There are no defaults for this data
****

****
****

End of data input for the redesign option. Return to the main input data, Card 17.

****
****

NOTES ON INPUT DATA

These notes are meant to amplify the information given in the card list and are applicable to all versions of the program except where otherwise stated.

NOTE 1 INLET FLOW ANGLE OR TANGENTIAL VELOCITY

IN_VTAN is used to specify the type of inlet boundary condition on the inlet flow direction on the blade-blade stream surface (i.e. on $\tan^{-1} (V_\theta/V_m)$).

> If IN_VTAN = 0, the absolute flow angle is held fixed at the value given by BS(K) . In degrees. (Card 35).

> If IN_VTAN = 2, then the relative flow angle is held fixed and = BS(K). In degrees. (Card(35)).

> If IN_VTAN = 1, then the absolute swirl velocity is held fixed and equal to VTIN(K) (card 33)in m/sec. and the value of BS(K) is ignored.

IN_VTAN =0 is usually the best condition for both fixed and rotating blade rows.

IN_VTAN = 1 should be used if the relative inlet flow is supersonic since a fixed flow direction may then be unstable.

NOTE 2   STREAMWISE SURFACES USED FOR DATA INPUT

 INPUT is used to specify the type of surfaces on which the blade geometry is to be input. The program interpolates in this data to set up the grid required for the calculation.

>>>> If INPUT = 0, then the blade geometry is given on NOSECT cylindrical surfaces (r = const = RCYL). In this case the hub and casing geometry must be input separately in cards 18-21.

>>>> If INPUT = 1, the data is given on NOSECT <u>arbitrary</u> streamwise surfaces starting with the one nearest the hub and ending with the one nearest the casing. The hub and casing grid points must then be put in separately in Cards 18-21. It is highly desirable that the first surface is inboard of the hub and the last surface is outboard of the casing.

>>>> If INPUT = 2, the surfaces are as for INPUT = 1, but the first surface is <u>coincident </u>with the hub and the last surface is <u>coincident</u> with the casing. In this case the hub and casing grids do not need to be put in separately.

INPUT = 2 is by far the most convenient type of data input option. Extrapolation of the grid from the input surfaces to the hub or casing is very dangerous and is likely to cause problems.

The surfaces used for blade geometry input need not be the actual stream surfaces of the flow.

SEE ALSO NOTE 15 FOR FURTHER DATA ON INPUT SECTIONS

<u>Note 3</u>   SPECIFICATION OF THE RADIAL AND STATIC PRESSURE VELOCITY AT
             INLET

IN_VR   is used to specify the inlet boundary condition on the radial velocity and
on the static pressure at inlet. Note IN_VR is changed to IN_PRESS before being
used for compatibility with NEW_READIN .

> If IN_VR = 0 then the radial velocity at inlet will be obtained by
> extrapolation using $dV_r/dm = 0$, and the value of
> BR(K) will be ignored. In this case the inlet static
> pressure will be found from the condition $dP/dm = 0$,
> where m is the meridional distance.

> If IN_VR = 1 or -1 , the inlet pressure is obtained
> from a condition $d^2P/dm^2 = 0$.

> IF IN_VR = +1 then the radial velocity at inlet will be
> obtained from the specified meridional
> pitch angle, BR(K).   <u>This angle must be
> compatible with the hub and casing slopes
> at inlet.</u>

> IF IN_VR = -1 then the radial velocity is obtained by extrapolation
> using dVr/dm =0.

> If IN_VR = 3 or -3   then the upstream boundary
> condition is as with IN_VR = +1 and -1 ,
> but the pressure is circumferentially
> averaged. This may be used to cure an
> instability which sometimes occurs at the
> inlet boundary. This instability takes the
> form of an unsteady wave moving in the
> circumferential direction and only seems
> to occur when the grid is highly distorted
> (i.e. skewed) at inlet.

If IN_VR= 4 or -4   then the boundary condition is as
with IN_VR= + 1 and -1 but the static pressure at the
upstream boundary is taken to be uniform at the value
obtained at the mid-span and mid-pitch point. This option
may be useful for calculating flows with a strong velocity
gradient in the inlet as it prevents failure when the inlet
velocity reverses locally .

Boundary conditions which use d^2P/dm^2 =0, ie ABS(IN_VR) = 1 , 3 or 4 must be used whenever a significant streamwise pressure gradient is expected at the inlet, as in the case of many radial inflow turbines. This may require a lower value of the inlet pressure relaxation factor, RFIN.

If the pitch angle is specified at inlet then it must be compatible with the hub and casing surface angles.

IN_VR = +1 or +3 is generally to be preferred for axial flow machines.

NOTE 4   TIME STEP OPTIONS

CFL is the timestep multiplying factor and is the main parameter controlling stability of the program. The length of timestep taken is given by $\Delta t = CFL.\Delta s/a_0$ where Δs is the minimum length scale of a cell and a_0 is the absolute stagnation speed of sound at inlet.

Local time stepping is always used based on the length scale of each cell and the local average relative Mach number in the cell. The time steps are updated every 5 steps based on the current values of Mach number.

 For the "scree" scheme values of CFL = 0.4 -> 0.5 are typical. To give a margin of safety it is usual to start with CFL = 0.4 and only to increase it if this is found to be stable.

For the "SSS" scheme values of CFL = 0.7 are typical. Larger values are often stable but may give oscillatory residuals without causing failure. In such cases the solution is usually perfectly acceptable.

Values of CFL less than 0.2 should never be needed.

If problems with stability occur then the first remedy tried should be a reduction in CFL.

ITIMST determines the type of time step to be used. The standard option is ITIMST = 3 which means that the timestep is evaluated for each individual cell, using the cell dimensions and Mach number, and the standard "scree" scheme is used.

If ITIMST = -3 then the time step is evaluated as above but the "SSS" scheme is used with standard coefficients.

If ITIMST = 4 or -4 then the coefficients of the "SSS" scheme are read in as data.

If ITIMST = 5 then the low Mach number option is used with the standard "scree" scheme. This uses artificial compressibility to calculate the pressure from an artificially low speed of sound, which is input as data. This speed of sound should be about twice the maximum relative velocity expected in the flow.

If ITIMST = -5 then the low Mach number option is used with the "SSS" scheme.

If ITIMST = 6 then artificial compressibility is used with fully incompressible flow. The required desnsity is read in as data.

If ITIMST = -6 then the incompressible option is used with the "SSS" scheme.

Note 5   FOURTH ORDER SMOOTHING

Combined second and 4th order smoothing is now used as standard without the previous option of using only second order smoothing. The input variable ISMTH is not longer required.

FACSEC   (Card 43) is the proportion of fourth-order smoothing, which should be about 0.8. This means that, if the smoothing factor input is SF, the second order smoothing is applied with a factor of (1. – FACSEC)*SF and combined with fourth order smoothing applied with a smoothing factor of FACSEC*SF.

This introduces much less numerical viscosity than second order smoothing alone. With fourth order smoothing, larger values of SFT and SFX , e.g. 0.02, can be used without significantly affecting the solution. (see Note 7).

To use second order smoothing set FACSEC = 0.0.

NOTE 6   TIME STEP MULTIPLYING FACTOR, CFL .

CFL is the timestep multiplying factor and is the **main parameter controlling stability of the program.** The length of timestep taken is given by Δt = CFL.$\Delta s / a_0$ where Δs is a length determined by ITIMST and a_0 is the absolute stagnation speed of sound at inlet.

For stability CFL should be **less than** approximately $1/(1 + M_{max})$ where M_{max} is the maximum **relative** Mach number expected anywhere in the flow field.

For the Scree scheme values of CFL = 0.4 -> 0.5 are typical. To give a margin of safety it is usual to start with CFL = 0.4 and only to increase it if this is stable.

For the SSS scheme values of CFL = 0.7 are typical. Larger values are often stable but may give oscillatory residuals without causing failure. In such cases the solution is usually perfectly acceptable.

Values of CFL less than 0.2 should never be needed.

If problems with stability occur then the first remedy tried should be a reduction in CFL.

<u>Note 7</u>   SMOOTHING FACTOR IN THE PITCHWISE AND SPANWISE DIRECTIONS, SFT.

The smoothing factor **SFT** controls the smoothing in the pitchwise and spanwise directions. Low values of SFT which are approximately equal to 0.005 will usually provide stability with negligible 'viscous' effect from the smoothing.

Note that in MULTALL-06 and above the input value of SFT is scaled by CFL/0.5 before it is used.

An increase of smoothing should be the second resort (after reducing CFL) if signs of instability occur.

The value of SFT is increased by 0.02 at the start of a calculation and then gradually reduced to the input value over the first NCHANGE steps.

<u>Note 8</u>   AXIAL SMOOTHING FACTOR. SFX.

The axial (strictly streamwise) smoothing using **SFX** is now always combined second and fourth order smoothing and so has very little effect on the solution. The proportion of fourth-order smoothing used is determined by FACSEC which should be about 0.8. A value of SFX = 0.005   is typical.

Note that in MULTALL-06 and above the input value of SFT is scaled by CFL/0.5 before it is used.

 The value of SFX is increased by 0.02 at the start of a calculation and then gradually reduced to the input value over the first NCHANGE steps.

<u>Note 10</u>   NEGATIVE FEEDBACK TO LOCALISE ANY INSTABILITY

DAMP controls the amount of negative feedback.

The maximum change per iteration is limited to approximately DAMP x a reference value of the variable concerned. This prevents local instabilities, such as might occur during the initial transient, growing and causing failure of the whole calculation. It should have no effect at all on the steady solution. A value of DAMP = 10 is usually acceptable and should be regarded as standard. Low values (= 5.) give greater stability but sometimes produce incorrect results and so should be used with caution. Higher values of DAMP will produce faster convergence and DAMP = 99.9 may be used if the initial transients are weak.

 If the value of DAMP is set to be greater than 100 then damping is not used. This is usually possible with the SCREE scheme. However, use of DAMP will give faster convergence with no loss of accuracy.

 The value of DAMP is automatically reduced to ¼ of the input value on starting a calculation and then gradually increased to the input value over the flrst NCHANGE steps.

Set DAMP = 10 for general use.

Note
It has been found that the convergence of some multistage machines is so rapid that convergence of the main flow may occur whilst a few grid points in the viscous layers are still changing. In this case low values of DAMP may force premature convergence and higher values, say DAMP = 50 should be used.

Note 11   EXIT FLOW FIELD SMOOTHING.

SFEXIT is used to provide a powerful smoothing of the exit flow in cases where the latter becomes so non-uniform as to produce back flow at the exit. It has mainly been found to be necessary for some centrifugal compressor calculations where separation occurs in the vaneless diffuser. The smoothing is applied over NSFEXIT points upstream from the exit boundary. A value SFEXIT = 0.5 can be used with NSFEXIT = 1, but SFEXIT = 0.1 and NSFEXIT = 5 -> 10   is usually preferable if there are sufficient grid point downstream of the last blade row. For most machines SFEXIT can usually be set to zero.

See also THROTLE_PLATE which can be used for the same purpose.

NOTE 12   PITCHWISE AND SPANWISE GRID SPACING

 FR(K) and FP(I) control the <u>relative</u> spacing of the grid lines in the spanwise and pitchwise directions respectively. Only the **relative** spacings are needed and the values input are then divided by their sum to give the fraction of the height and gap occupied by each element.

The change in relative spacing between adjacent grid points should not be greater than about 30% (i.e $F(I+1)/F(I)$ < 1.3) because the smoothing routines assume uniform spacing and will produce errors for highly non-uniform spacing. Lower values than this, say 1.25, are really preferable. An expansion ratio of 1.2 is suitable when fine grids are used because the required overall expansion can then be achieved with a lower expansion ratio.

Use of too high an expansion ratio is one of the most frequent mistakes made by users of the program.

The streamwise grid spacing (J direction) should also not vary by a factor of more than 1.3 between adjacent points. Again an expansion ratio of about 1.25 is preferred.

Overall very large variations (~ 50:1) in grid spacing may be used as long as the spacing is changed gradually as described above. For example, if IM = 13, and the expansion ratio is 1.4 the values of FP(I) might be:

$$
\begin{array}{lcl}
FP(1) & = & 1.0 \\
FP(2) & = & 1.4 \\
FP(3) & = & 2.0 \\
FP(4) & = & 2.8 \\
FP(5) & = & 4.0 \\
FP(6) & = & 5.0 \\
FP(7) & = & 5.0 \\
FP(8) & = & 4.0 \\
FP(9) & = & 2.8 \\
FP(10) & = & 2.0 \\
FP(11) & = & 1.4 \\
FP(12) & = & 1.0 \\
\end{array}
$$

 Highly non-uniform grid spacing, with the ratio of maximum to minimum spacing = about 25 are usual for viscous calculations .

The grid spacing is generated automatically within the program if FP(3) or FR(3) is set equal to zero. In this case the spacing is varied as a geometrical progression away from the walls with a ratio FR(1) or FP(1) between adjacent points up to a maximum spacing = FR(2) or FP(2) times the spacing at the wall. The other values of FP and FR must still be input but are not used and so can be set equal to zero. For example when using this option the values of FR(K) might be:

    FR(1) = 1.2
    FR(2) = 25.0
    FR(3) = 0.0
    All other values of FR(K) = 0.0

will give a grid expansion ratio of 1.2 up to a maximum value of 25. All other values of FR(K) from K = 4 to K = KM-1 (or FP(I) from I=4 to IMM1) must be input but are not used and so can be set equal to zero.

This option can also be used for FRNEW(K) in card 45 .

Note 13   OUTPUT PRINTING AND PLOTTING OPTIONS

 IOUT(I) determines which flow variables are to be printed out on convergence or as specified by IPRINT and KOUT .

IOUT(I) = 1 or 3 gives a full printout of the
I'th variable,

IOUT(I) = 0 gives no printout.

IOUT(I) = 2 gives a printout of the
circumferentially averaged value of
the variable.

The variables are numbered as follows :

1 = Percentage change in V_m
2 = Axial velocity
3 = Absolute swirl velocity
4 = Radial velocity
5 = Static pressure
6 = Relative Mach number
7 = Absolute stagnation temperature
8 = Meridional velocity
9 = Swirl angle $\tan^{-1}(W_q/V_m)$.
10 = Meridional pitch angle $\tan^{-1}(V_r/V_m)$
11 = Density
12 = Ratio of $P/T^{**}(\gamma/(\gamma-1))$ to the inlet value at mid span and mid pitch. This should =1.0 for isentropic flow and can be thought of as the ratio of the local stagnation pressure to that that would be obtained in an insentropic process.
13 = Pressure coefficient $(P-P_{IN})/(P_{01}-P_{IN})$

A printed output file is <u>very long</u> and is not usually very useful. Graphical inspection of the output is much more efficient.

The above printed output is sent to Fortran unit 6, which defaults to the screen. To send it to a file use a command like:

Multall < data.in > results.out

A separate output file "flow_out" for plotting the solution is automatically sent to Fortran unit 8 on completion of a run. This file may be read and plotted by the plotting program CONTOUR7. The file "flow_out" is also used as a restart file, which can be used to start a new calculation from a previous solution when there are only small changes expected. Use of a restart file saves considerable run time and is recommended whenever possible.

A file called "global.plt" for plotting the mass averaged flow quantities using the program "GLOBPLOT" is sent to FORTRAN unit 11.

Note 14   MULTIGRID LEVELS

Three levels of multigrid are available in the standard program.

IR, JR, KR are the number of individual elements along the I,J,K sides of the **smaller** multigrid blocks.

IRBB, JRBB, KRBB are the numbers of individual elements along the I,J,K sides of the **larger** blocks.

 If IR, JR, KR are **all** = 1, then the multigrid is not used .

It is <u>preferable but by no means essential </u>to use an integral number of blocks in each coordinate direction i.e. (IM-1)/IR,   (IM-1)/(IRBB),(JM-1)/JR, (JM-1) /(JRBB), (KM-1)/KR, and   (KM-1) /(KRBB) should all be integers.

The optimum block size depends on the problem, but IR, JR, KR, all = 3 seems good, as does IRBB, JRBB, KRBB all = 9 . For a coarse 13x73x13 point grid IR=JR=KR =3 and IRBB=JRBB=KRBB =6 would be suitable. For viscous flow calculations IM = KM = 28 with IR = KR= 3 and IRBB = KRBB = 9 seems to work well, as does IM=KM=49 with IR=KR=4 and IRBB=KRBB = 12.

 A third level of multigrid is formed from one-dimensional blocks which fill the whole pitch and whole span.   The block sizes for this are generated automatically without any user input. Five blocks are generated, one upstream of the leading edge, three within the blade row and one downstream of the trailing edge. If the third level of multigrid is used then the second level blocks **should not also** fill the whole pitch and span.

The scaling factors on the changes produced by the multigrid are input as data in card 19. The greater these are the faster will be convergence but the greater the tendency to instability. Hence the values should be significantly less than unity. Typical safe values are FBLK1=0.4, FBLK2=0.2 and FBLK3=0.10 , although larger values can often be used. The third level of multigrid is sometimes prone to instability and should not be used with too high a scaling factor. A factor 0f 0.1 is usual.

<u>Note 15</u>   DATA INPUT SURFACES

 The streamwise or cylindrical surfaces on which blade section data is input need not be the surfaces which are used for the computational mesh. The program will interpolate or extrapolate in the data input to set up the mesh. However, <u>extrapolation is always dangerous</u>, and so the first input section should lie on or inboard of the hub and the last on or outboard of the casing. A single input section is always sufficient for a blade of constant section and two sections for a blade with linear variations in section. The number of sections required depends on how much the blade geometry changes along the span but more than 5 sections are seldom required even for highly twisted blades.

The points at which the blade coordinates are input on each section determine the intersection of the quasi-orthogonal surfaces with that section, and so the number of points (JM) must always equal the number of quasi-orthogonal surfaces (JM). Similarly, the leading edge point (JLE) and trailing edge point (JTE) must be the same on all surfaces. However, the relative spacing of the J points along each surface need not be the same since the quasi-orthogonal surfaces need not intersect the meridional plane in a straight line

NOTE 16   DIMENSIONS OF DATA AND VARIABLES

Physical dimensions of input coordinates and fluid properties should always be **consistent.** If the dimensions are in metres then all fluid properties must be in SI units. However, for a single **stationary blade row** the dimensions of the blade do not affect the magnitude of the velocities and so any convenient units may be used for the blade coordinates.

If it is desired to use British units then **they must be consistent,** if lengths are in ft then pressures must be in poundals/sq ft and temperatures in degrees R and the gas specific heat in ft poundals/ deg R.

Note 17   SPECIFICATION OF THE MASS FLOW RATE

IN_FLOW in Card 3 may be used to obtain a specified mass flow as input in card 33. This option is not generally used as the mass flow forcing introduces artificial changes of stagnation pressure.

If IN_FLOW = 3,   Then the required mass flow
rate for the whole annulus or machine,
in kg/sec, is read in Card 42

If IN_FLOW = 2,   Then the local mass flow is
forced towards the current average
mass flow . This may give faster
convergence whilst the mass flow
will still be fixed by the pressure
ratio.

If IN_FLOW = 0   Then the mass flow is determined by
the input pressure ratio as is usual
for Euler and Navier-Stokes solvers.

The mass forcing function is damped by a factor RFLOW for which a typical value is 0.1 . However, when using IN_FLOW = 2, lower values of RFLOW, say = 0.01, should be used as high values can cause spurious entropy changes even when the mass flow is only forced towards the average value.

 When this option is used then the stagnation pressure change will be adjusted to make the mass flow and pressure ratio specified compatible and conservation of stagnation pressure (or entropy) cannot be expected. Hence the calculated efficiency will not be correct when IN_FLOW = 3 is used.

IN_FLOW =0 should be regarded as standard. However, IN_FLOW =2 may give faster convergence, especially for centrifugal compressors. IN_FLOW=3 should not be used unless the mass flow is known reasonably accurately.

NOTE 18   ISHIFT

This is a special option for shifting the coordinates of the input data for multi-blade row calculations.

IF ISHIFT = 0 then the grid coordinates are used as read in. **Note this option must be not be used unless the mixing plane and next downstream plane were made coincident in the raw data.**

If ISHIFT =1 then the axial coordinates of all but the first blade row are automatically shifted from their input values to make the blade rows line up on the hub stream surface. Only the hub grids will line up and those at other spanwsie positions will not generally do so. Note that only the axial coordinates are shifted so that the radial coordinates must already be compatible, hence the option must be used with great care on radial flow machines.

If ISHIFT = 2 then the blades are shifted so that the grids on all stream surfaces match at the mixing planes. The grid spacing is made vary geometrically between the trailing edge of the upstream row and the mixing plane and between the leading edge of the downstream row and the mixing plane. The mixing plane and the next downstream plane are automatically made to be coincident as is essential. This option gives a good grid between the blade rows.

If ISHIFT = 3 then the shifting is done as with ISHIFT = 2. However, in the meridional view, the hub and casing and the grid will be formed by straight lines (conical stream surfaces) joining the trailing edge of one blade row and the leading edge of the next row. This is also applied to the hub and casing and so can cause small changes of annulus geometry. This option is necessary when the surfaces on which the blade geometry is input are not continuous at the interface plane. i.e. different surfaces are used in different blade rows. This is often the case if the data input is on cylindrical surfaces.

ISHIFT = 4 is the same as ISHIFT = 3 but the hub and casing shapes are not changed, i.e. they are not made conical.

The use of ISHIFT = 2 or = 4 is strongly recommended.

NOTE 19 GRID EXTRAPOLATION WHEN ISHIFT IS NOT = 0 .

When a new grid is generated between blade rows by using ISHIFT = 1 , 2 or 3 then the slope of the periodic boundary (I=1) can be obtained by extrapolating the blade centre lines from the trailing edge to a point NEXTRAP_TE grid points upstream and from the leading edge to a point NEXTRAP_LE points downstream of it. NEXTRAP_LE and NEXTRP_TE are input in CARD 2 and are usually taken to be 5 but this may need to be increased for a highly cambered leading or trailing edge.

In some cases the blade centre line is highly curved at the leading or trailing edges and the above extrapolation may give an incorrectly aligned grid. In such cases the grid can be set at angles BETAUP and BETADWN1 to the meridional direction (i.e. to a line of constant theta coordinate) where BETAUP and BETADWN1 are read in a data in CARD 13. If these angles are set to zero, or are not input in CARD 13, then the grid extrapolation using NEXTRAP_LE and NEXTRAP_TE will be used as described above. If they are input and are non-zero then the input angles will over-ride those calculated using NEXTRAP_LE and NEXTRAP_TE.

Recent versions also enables the grid angle at the end of the current blade row (i.e. at the exit boundary or mixing plane) to be input as BETADWN2 . The grid angle is gradually changed from BETADWN1 at the trailing edge to BETADWN2 at the end of the current blade row. IF both BETADWN1 and BETADWN2 are set to zero they are not used at all. If BETADWN2 is set to zero but BETADWN1 is not = zero, then BETADWN2 is set equal to BETADWN1.

NOTE 20 DEFINITION OF COOLING FLOWS AND OF DIRECTION OF EJECTION

The cooling flows may be ejected through "patches" on the blade and endwall surfaces. The I, J and K grid points covered by the patch are input as data. The coolant stagnation temperature, pressure, velocity and direction of ejection are taken to be constant over each patch. The stagnation temperature and pressure that are input are those at which the coolant is supplied to the blade. The increase of stagnation temperature and pressure due to work done on the coolant by a rotating blade is calculated within the program using the input values of the coolant supply angular momentum (RVT_IN) and disc rotational speed (RPM_COOL). The velocity of ejection **relative** to the blade is then calculated from the specified **relative** Mach number of the flow leaving the cooling holes.

The direction of the cooling jet leaving the blades and endwalls is specified by two angles. The first angle is that between the coolant jet and the plane tangent to the surface through which it is being ejected. This is more easily visualised as (90° - the angle between the coolant jet and the local normal to the surface). See Fig 20a. The second angle is that between the projection of the coolant jet onto the surface and a line in the surface. The definition of this line differs for blade surface or endwall ejection as described below.

For ejection through the blade suction or pressure surfaces the line is a line of constant radius in the surface., i.e. the intersection of the blade surface with a cylindrical surface. See Fig 20b.

For ejection through the hub or casing the line is a line of constant circumferential angle, θ, drawn on the hub or casing, i.e. the intersection of the hub or casing with a plane of constant circumferential angle θ passing through the machine axis.
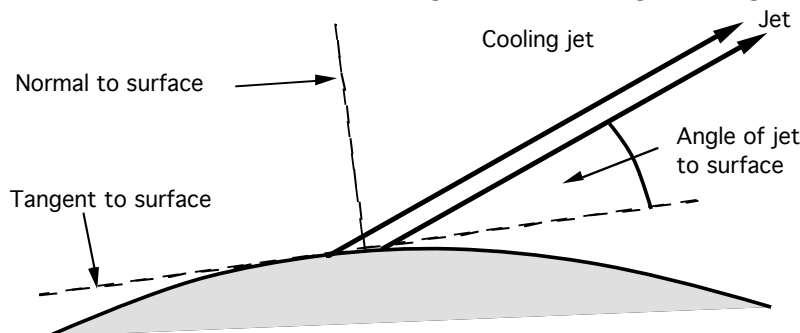

Fig 2oa . Definition of the angle of the coolant
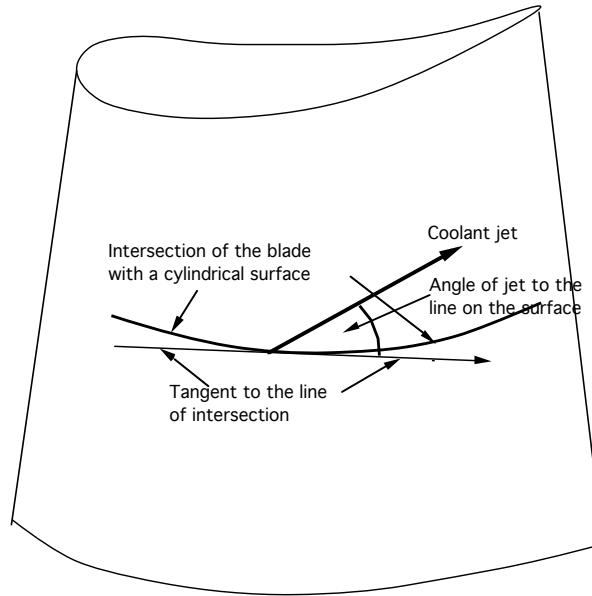jet to the blade or endwall surface.

Fig 20b, Definition of the angle of the jet on the surface. The view is perpendicular to the surface at the point of coolant ejection
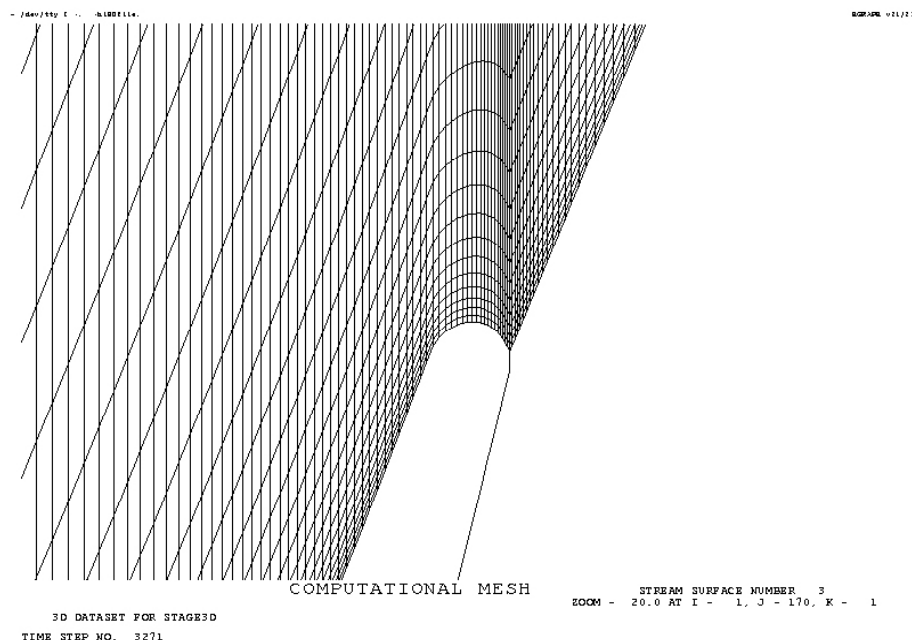
These definitions of cooling flow direction have been chosen because they can be easily applied to both axial and radial flow machines.

Note 21. Use of a body force to make the flow separate at a thick trailing edge.

This option is seldom used.

It is difficult to use a cusp on a blade row with a very thick trailing edge but if the grid is refined around the trailing edge the flow will usually not separate soon enough, leading to a locally low pressure on the pressure surface and negative blade loading at the TE. This is not physically realistic and in practice the flow usually separates at the start of the trailing edge circle (blend point). In MULTALL-09 the TE separation can be forced by a body force filed allowing a fins mesh to be used around the TE. This is invoked by setting IFCUSP = 2.

 NSEP_I1 is the number of grid points upstream of the TE from which the blade I = 1 surface is extrapolated. NSEP_IM is the same for the I = IM surface . The force field extends N_WAKE grid points downstream of the trailing edge, this value may be negative to stop the force field before the TE. The body force is applied over a region which is more than (SEP_THIK x local blade thickness) **inside** the extrapolated region, see the second figure below,. The magnitude of the body force is proportional to (1 – SEP_DRAG) and it acts in the direction of the local velocity. Typically SEP_THIK = +0.01 (it may be positive or negative) and SEP_DRAG = 0.99 . Lower values of SEP_DRAG lead to virtually stagnant flow in the affected region.

COMPUTATIONAL MESH

3D DATASET FOR STAGE3D
TIME STEP NO. 3271

STREAM SURFACE NUMBER   3
ZOOM - 20.0 AT I - 1, J - 170, K - 1

Fine grid around a thick trailing edge.

The body force is applied over the shaded region as illustrated below. The region may be made wider in the pitchwise direction by making SEP_THIK negative.

SEP_THIK

NSEP_I1

N_WAKE

SEP_THIK

NSEP_IM

Extent of the body force field.
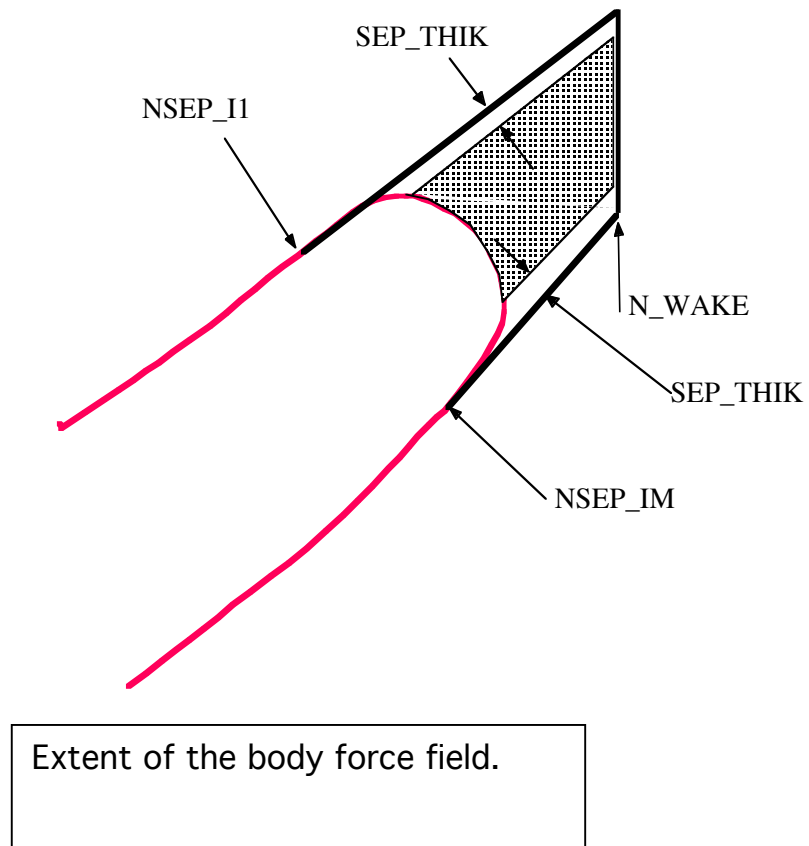
Note 22.   THE MIXING PLANE MODEL

Development of a good mixing plane model is one of the most difficult problems in CFD. The mixing plane is an artificial concept designed to permit steady calculations of blade rows which are in relative motion due their rotation. It is generally assumed that the mixing plane should allow the flow from an upstream blade row to mix out as if it were doing so in a long duct with constant area and it should allow the flow to enter the downstream row as if it had originated from a pitchwise uniform flow far upstream. Hence the mixing plane must transmit the mixed out fluxes of mass momentum and energy from one blade row to the next whilst causing the minimum distortion to the pitchwise non-uniform flows leaving the upstream row and entering the downstream row. The mixing out of a non-uniform flow to a pitchwise uniform flow is generally an irreversible process and although the fluxes of mass, momentum and energy must be conserved in the mixing process the entropy will usually increase. This increase in entropy represents the mixing loss which occurs in the real flow. However, It should be emphasised that this is only a model of reality and it is not obvious that the mixing loss, which occurs in an unsteady flow in a real machine, is the same as that at the mixing plane.

Several different mixing plane model have been used during the development of MULTALL. The latest one in MULTALL-14.6 and later is thought to be the best yet. It is a combination of the one last used in TBLOCK, which is robust and permits reversed flows across the mixing plane, with the flux extrapolation method used in earlier versions of MULTALL. The same model is now used in both MULTALL and TBLOCK.

As in all previous versions there are two coincident "J" grid surfaces at the mixing plane. These are numbered JMIX and JMIX+1. All flow properties are pitchwise uniform and equal on both of these faces, although they will vary in the spanwise (K) direction. The pitchwise uniform value is the mixed out value. The values are made equal on both surfaces by treating the flow from JMIX to JMIX+1 as if it were between two faces of a cell and time stepping the flow between them, exactly as for the cells in the rest of the grid. This ensures that when the solution is converged the fluxes become equal. Since the flows are pitchwise uniform this makes all the flow properties equal on the two faces.

The upstream and downstream faces of the mixing plane are decided by checking the flow direction on each spanwise (K) grid surface and the direction can change from one surface to the next. There is no presumption that the flow is in the positive J direction. Hence the model can allow any amount of reverse flow.

For the cells upstream of the mixing plane, i.e. those between JMIX-1 and JMIX, the fluxes on their upstream face are calculated as normal but the fluxes on their downstream face, i.e. on the mixing plane, JMIX, are obtained by flux extrapolation. If the upstream face is JMIX-1, this involves adding a fraction of the difference between the local flux and the pitchwise averaged flux at JMIX -1 to the flux calculated from the uniform flow at JMIX. i.e.

$$\text{FLUX}_{jmix} = \text{FLUX}_{avg,jmix} + \text{FEXTRAP} \times (\text{FLUX}_{jmix-1} - \text{FLUX}_{avg,jmix-1})$$

Hence the cells between JMIX-1 and JMIX "see" only a fraction (1-FEXTRAP) of the uniform flux at the mixing plane. FEXTRAP is input as data and a typical value is 0.9. The more closely the grid lines JMIX-1 and JMIX are spaced the larger should be FEXTRAP, values of 0.99 can be used for very close spacing. The value should be decreased for wide spacing of the grid points.

The pitchwise average flux at JMIX is not changed by this procedure and so the uniform flow at JMIX satisfies conservation of mass, momentum and energy between JMIX-1 and JMIX , hence it is the mixed out flow corresponding to the non-uniform flow at JMIX-1 .

If FEXTRAP is set to zero then there is no flux extrapolation and the changes in primary variables in the cells immediately upstream of the mixing plane are then made pitchwise uniform. The treatment then becomes the same as in TBLOCK-13 where it was found to be exceptionally robust.

Flux extrapolation was found to be of little benefit and slightly destabilising on the downstream side of the mixing plane and so is not used there. Since JMIX+1 is on the mixing plane the next surface downstream of the mixing plane is JMIX+2. The cells between JMIX+1 and JMIX+2 are updated using the pitchwise uniform flux from the uniform flow at JMIX+1 and the pitchwise average flux at JMIX+2. Hence the changes that points at JMIX+2 receive from their upstream cells is pitchwise uniform, although they also receive a pitchwise non-uniform change from the cells downstream of them so the flow on them is not pitchwise uniform. This ensures conservation of mass momentum and energy between the mixing plane and the downstream flow. This is the treatment used in TBLOCK-13, it is robust and permits reversed flow but does tend to make the flow too uniform when applied close to a leading edge in which case the entropy and enthalpy downstream of the mixing plane may not be pitchwise uniform. This is overcome by smoothing the flow at JMIX+2 towards an isentropic flow which is calculated using with the local static pressure at JMIX+2, the pitchwise

uniform enthalpy and entropy from JMIX+1 and the flow direction from a weighted average of the pitchwise uniform value at the mixing plane and the average of the values at JMIX+3 and JMIX+4 . The fraction of the average downstream angles used is FANGLE, which is input as data in card 44, and for which a typical value is 0.9 . Taking the average flow direction from JMIX+3 and JMIX+4 was found to be more stable and not significantly less accurate than extrapolating the flow direction from JMIX+3 and JMIX+4. On every time step the flow is smoothed towards this isentropic value by a factor RFMIX for which a low value of 0.01 is usually sufficient, although higher values (say 0.05) are usually perfectly stable. If RFMIX = zero then there is no smoothing to isentropic flow and the treatment is the same as in TBLOCK-13.

This procedure works well in all cases except those when the flow relative to the downstream blade row is supersonic so that pressure waves, either expansions or shocks, run into the mixing plane from downstream. In this case the pitchwise variation in flow direction downstream of the mixing plane must be compatible with the Mach number variation, i.e. they must satisfy the Prandtl-Meyer relationship. Whenever the downstream flow is supersonic this relationship is used everywhere except at mid-pitch, where the angle is still set by the angle extrapolation. This allows pressure waves to intersect the mixing plane without reflection as illustrated in the Figure below.

The smoothing of points adjacent to the mixing plane has also been changed so that it does not include values of the variables on the mixing plane, this ensures that the pitchwise uniform values on the mixing plane are not imposed on the rest of the flow. This smoothing is scaled by FSMTHB which is input as data and for which a typical value is 1.0, however, the exact value does not seem to have much effect on the solution.

To use exactly the same mixing plane model as in TBLOCK-13 set both FEXTRAP and RFMIX = 0.0
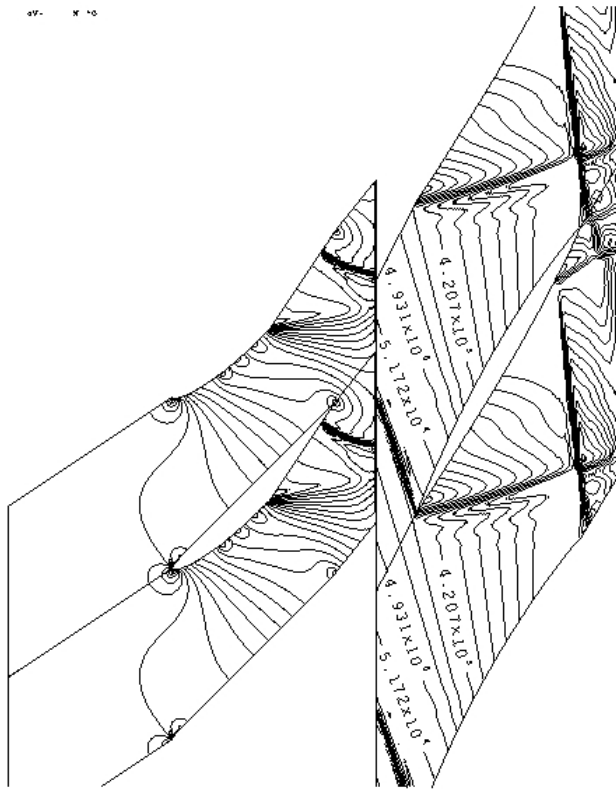
Intersection of pressure waves with the mixing plane.

The model described also works well with reverse flow across the mixing plane as illustrated by the Figure below.

VELOCITY VECTORS
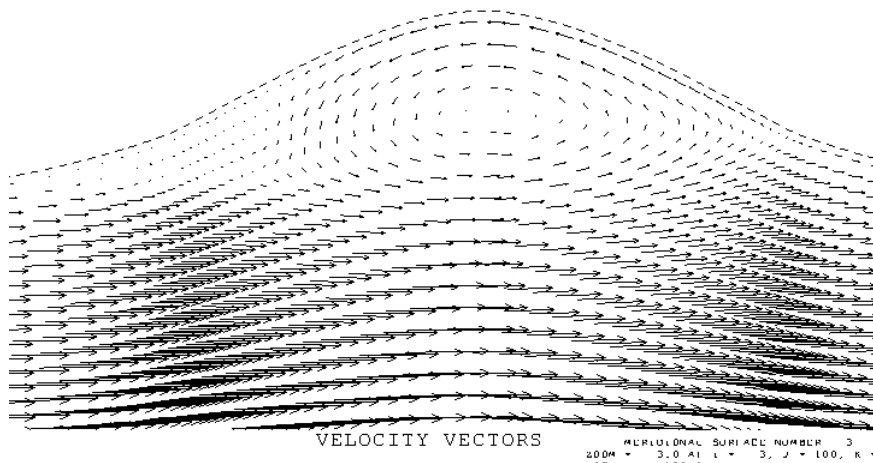
Reverse flow across the mixing plane, which is in the centre of the bulge.

J D DENTON
Updated January 2017