Proceedings of ASME Turbo Expo 2011
GT2011
June 6-10, 2011, Vancouver, British Columbia, Canada

GT2011-46545

DUAL-SOLUTION AND CHOKED FLOW TREATMEMT IN A STREAMLINE CURVATURE THROUGHFLOW SOLVER

Prashant Tiwari[§]
GE Global Research Center
Niskayuna, NY, USA

Alex Stein
GE Energy
Greenville, SC, USA

Yu-Liang Lin
GE Infrastructure Technology
Cincinnati, OH, USA

ABSTRACT

In most turbomachinery design systems streamline curvature based throughflow calculations makes the backbone of aero design process. The fast, reliable and easy to understand solution is especially useful in performing several multistage design iterations in a short period of time. Although the streamline curvature based technique enjoys many benefits for subsonic applications there are some challenges for transonic and supersonic flow applications, which is the focus of this paper

In this work it is concluded that three key improvements are required to handle transonic flows in a streamline curvature throughflow solver. These are: 1) ability to overcome dual sub- and supersonic solution and guide the solver towards supersonic flow solution where applicable; 2) suitable technique to calculate the streamline curvature gradient term which can avoid singularity at sonic meridional Mach number and high gradient values in transonic flows; 3) suitable technique to handle choked flow in the turbomachinery flowpath.

Solution procedures for "dual-solution" and choked flow treatment are new and developed as part of this work. However, procedure for calculating streamline curvature gradient is leveraged from earlier work done by Denton [1] and Came [2].

Implementation of these improvements is performed in a streamline curvature based throughflow solver. Numerical improvements presented here have been tested for a range of compressor and turbine cases (both subsonic and supersonic).

It is shown that the numerical improvements presented in this paper resulted in an enhanced version of streamline curvature throughflow solver. The new code produces consistent solution for subsonic applications with no sacrifice in accuracy of the solver. However, considerable robustness improvements are achieved for transonic turbine cases.

[§] Corresponding author: prashant.tiwari@research.ge.com

KEYWORDS

Radial Equilibrium Equation (REE), Streamline curvature, 2D Throughflow solver, transonic flows, high Mach, choked flow, localized choked flow treatment, dual solution

NOMENCLATURE

a	=	Speed of sound
A	=	Area
C_u	=	Circumferential absolute velocity
C_m	=	Meridional absolute velocity
dA	=	Differential area normal to the streamtube ($dA=nda$)
f_q	=	Blade force term
H	=	Total enthalpy
i	=	Unit vector
$\dot{m}$	=	Overall mass flow rate in the annulus
M_m	=	Meridional Mach number ($M_m=V_m/a$)
M_r	=	Relative Mach number
m	=	Distance along meridional direction
n	=	Normal vector
p	=	Static pressure
P_t	=	Stagnation pressure
R	=	Gas constant

r = Radius
r_c = Radius of curvature
S = Entropy
S_2 = Distance along the station. Sometimes also referred to as QO (q), $S_2 = -q$
T = Temperature
U = Blade speed
V_m = Relative velocity in meridional direction
W_u = Circumferential relative velocity

Greek Symbols
σ = Angle between station and vertical plane
ϕ = Angle between streamline and horizontal plane
β = Relative flow angle
λ = Blockage
ρ = Density
ψ = Stream function
γ = Ratio of specific heats ($\gamma = C_p/C_v$)

Subscripts
i = Station index
j = Streamline index
m = Meridional direction
n = Iteration index
u = Circumferential direction
z = Axial direction

1. INTRODUCTION

In most turbomachinery design systems the meridional throughflow calculation is the backbone of the design process. It is fast, reliable, easy to understand, deals easily with multiple blade rows and includes empirical loss, deviation and blockage correlations. Performance and experience from earlier machines can then be easily taken into account in the preliminary design phase. In recent years, full three-dimensional CFD methods are developed, but their applications for detailed analysis remain limited to single blade row calculations, even though multistage CFD is becoming more popular. The application of 3D CFD methods in design iterations of multistage turbomachinery meets serious limitations due to computational cost and complexity. For multistage calculations, throughflow method remains much faster than full 3D CFD calculations.

Throughflow theory is based on axisymmetric treatment of the circumferentially averaged flow [3]. Over the period of many years two main types of throughflow calculations have evolved: the streamline curvature [4] and the matrix (also known as stream function) throughflow [5]. For subsonic applications these two techniques have very small differences. However, stream function (or matrix) methods cannot easily be used for analysis programs when the flow within the blade rows is partly supersonic. This is because for any stream function distribution there exist two possible velocity fields, and there is no way of deciding which solution should be chosen. Similar limitation exists for the streamline curvature method as well [6]. However, the streamline curvature method can easily overcome this problem, as described in this paper. Additional limitation of stream function method is that the flow between the bladerows is restricted to relative Mach number equal to or less than 1, while streamline curvature method do not have this restriction. Marsh [6] described that for transonic flows, two possible solutions (sub and supersonic) satisfy numerically the governing equations on each streamtube. This is known as "*dual-solution*" issue and a possible method of overcoming this issue is described in this paper.

Most throughflow codes use the streamline curvature (SLC) method developed by Smith [4] and Novak [7] and based on the general S1/S2 theory of Wu [3]. Wu [3] was the first to formally derive a consistent model for both blade-to-blade flow (S1) and meridional throughflow (S2) using quasi-3D stream surface description. The SLC method for meridional throughflow takes its name from the prominent role of radius of curvature in the solution of meridional streamlines. A brief summary of streamline curvature based throughflow method is also given in this paper.

Although SLC methods enjoy many advantages there are few disadvantages as well. Three main disadvantages are described below. Firstly, it allows no reverse flow in the meridional plane. Hence, separated flows are best resolved using a fully viscous 3D CFD solution rather than with a 2D throughflow method. Secondly, the method suffers from a sharp increase in calculation time on grids with finely spaced quasi-orthogonals owing to the stability requirements of the streamline curvature calculation [8,9]. Thirdly, the method suffers significant robustness issue for transonic and supersonic flows, which is the focus of this paper.

The robustness issue in a SLC solver, for transonic flows, involves mathematical challenges. Although there is a smooth physical transition from a subsonic to a supersonic flow, the properties of the fluid in the two regions are quite different. In the subsonic region, the equations governing the steady flow are elliptic whereas in the supersonic region they become hyperbolic [10]. The mathematical difficulties arise in the transonic region, where both subsonic and supersonic flows exist and any method of solution must preserve the properties of both flows. Hence, the choice of solver and careful programming is required for solving streamline curvature based throughflow equations.

This paper deals with extending the capability of a generic streamline curvature based throughflow solver for transonic flows. Some of the transonic flow treatments tested in the past incorporates artificial density [11] or using upwind differentiation to deal with sudden discontinuity and shock effects [12]. However, incorporating artificial density often makes the solver unstable as this introduces artificial streamtube choking.

Some of the common techniques used to overcome transonic flow issues in a throughflow codes are described by Denton [1], Came [2], and Casey & Robinson [13].

In addition, Sayari & Bölce [14] also described a suitable approach for transonic compressors for nearly normal shock and choking prediction. This approach is based on an integral method where the averaging method works normal to the central streamline.

In this paper, the turbomachinery choked flow is divided into two categories: 1. Annular choke – when all the streamtubes are choked, 2. Localized choke – when only few streamtubes are choked in the annulus. It is emphasized that the problem of annular and localized streamtube choking is very similar. Hence, a brief theory of supersonic flow in a throughflow system is given in this paper. However, the solution of overcoming localized choke can be very different from annular choke. For localized choked flow a new and rather simple technique is developed in this paper.

Usually by definition the localized choking is limited to few streamtubes on the station. The problem is easier to overcome by identifying the sub and supersonic flow at the station and also identify the station where localized choking is likely to occur (or throat location). The technique described in this paper uses the average turning distribution, for the bladerow, to guide the calculation of REE equation and also solve other properties on the choked station.

It is shown that the numerical improvements presented here results in an enhanced version of a 2D streamline curvature based throughflow solver. The new code produces consistent solution for subsonic application with no sacrifice in accuracy of the solver. However, considerable robustness improvements are achieved for transonic and supersonic applications.

2. THEORY OF STREAMLINE CURVATURE (SLC) BASED THROUGHFLOW METHOD

The throughflow calculation method of an axial-flow compressor & turbine is based upon a meridional perspective of the flow. The full solution of radial equilibrium equation (REE) for turbomachinery is given by Smith [4], hence only a brief overview is given in this paper. The SLC based throughflow method, in particular, considers the flow within the turbomachinery bladerow as axisymmetric, compressible and inviscid. Due to lack of viscous effect, all throughflow solvers need to be coupled with separate viscous models, which bring the effect of loss, turning and blockage [15,16]. In many cases the effect of secondary flow mixing also needs to be accounted for using empirical models [17].

The grid for calculating REE is based on fixed stations (also called quasi-orthogonals or QOs) and floating streamlines. The streamlines are allowed to float based on the circumferentially averaged solution of REE. The curvature gradient calculated from movement of these floating streamlines forms the key contribution in the REE equation. A typical throughflow grid is shown in Figure 1. Where the x-axis is the axial distance and y-axis is the radial distance. The streamlines, denoted by normalized stream function (ψ) floats as a result of REE solution, with the exception of hub and casing streamlines which are kept fixed.

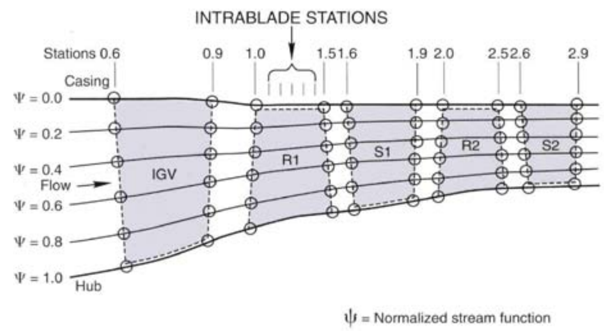

Figure 1: Schematic representation of a throughflow computational grid

Schematic representation of quasi-orthogonals and streamlines is given in Figure 2, where the red lines show arbitrary stations and blue lines show arbitrary streamlines.

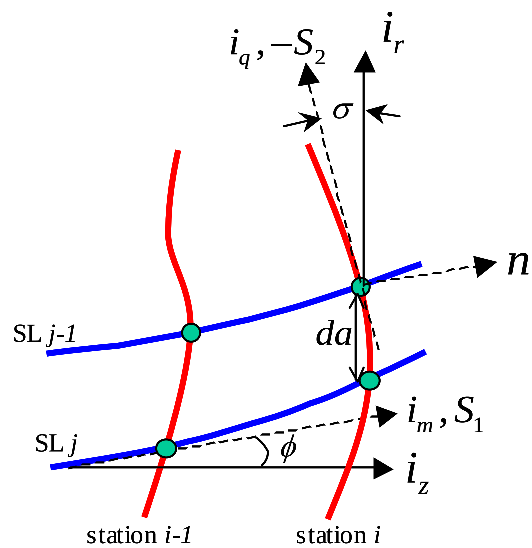

Figure 2: Streamlines and quasi-orthogonals (S_2) representation for the throughflow method

The final REE that makes the heart of any throughflow solver reflects the equation of the pressure forces to the inertial forces. A generic REE is given in Eq.(1), which represents the gradient of meridional velocity (V_m) in span-wise direction and needs to be solved using iterative procedure.

$$V_m \frac{\partial V_m}{\partial S_2} = V_m^2 \left\{ \frac{\cos(\phi-\sigma)}{r_c} - \sin(\phi-\sigma) \frac{1}{V_m} \frac{\partial V_m}{\partial m} \right\} + \frac{\partial H}{\partial S_2} - T_s \frac{\partial S}{\partial S_2} + f_q - \frac{1}{2r^2} \frac{\partial (rC_u)^2}{\partial S_2} \quad (1)$$

By integrating the REE presented in Eq.(1), we can obtain the final solution for meridional velocity at each station and streamline junction. This requires us to re-write Eq.(1) as a linear ordinary differential equation (ODE). The solution procedure depends on the mode of the throughflow calculation. These modes are: (1) Analysis mode - where calculations meet the imposed relative flow angle (β); (2) Design mode – where the prescribed swirl or rC_u distribution is met.

1^{st} order ODE forms of Eq.(1) for analysis and design modes are given below [18].

Analysis Mode

$$\frac{dV_m^2}{dS_2} = A(S_2) \cdot V_m^2 + B(S_2) \cdot V_m + C(S_2) \quad (2)$$

Design Mode

$$\frac{dV_m^2}{dS_2} = P(S_2) \cdot V_m^2 + Q(S_2) \quad (3)$$

where, the coefficients $A(S_2)$, $B(S_2)$, $C(S_2)$, $P(S_2)$ & $Q(S_2)$ are "non-constant" constants.

For simplicity, here we describe the solution procedure for design mode only. An analytical solution of Eq.(3) can be found using the standard "*integrating factor*" approach. The final solution of Eq. (3) in form of meridional velocity is given by

$$V_m = \left[\frac{1}{\exp^{\int_0^{S_2} -P(S_2) dS_2}} \int_0^{S_2} \exp^{\int_0^{S_2} -P(S_2) dS_2} Q(S_2) dS_2 \right]^{1/2} \quad (4)$$

The solution starts with guessed value of meridional velocity (which can be either at meanline or blade tip) and then integrated along the entire span. The two coefficients ($P(S_2)$ and $Q(S_2)$) are determined based on solution from previous iteration.

This solution of meridional velocity is obtained in conjunction with continuity equation, Eq.(5), for each station to satisfy mass conservation.

$$\dot{m} = \int_{tip}^{hub} \lambda \rho V_m dA \quad (5)$$

As mentioned earlier, the solution procedure is iterative in nature and as the REE equation is highly non-linear, care must be needed to successfully obtain the final solution. In our procedure the predicted meridional velocity, streamline curvature and movement are damped, to improve solver robustness. Denton [1] emphasized the need to only damp streamline curvature value as all other quantities are related to it. However, we find it useful to also damp meridional velocity and streamline movement to ensure convergence and robustness in solution procedure. A dynamic damping coefficient control algorithm, similar to what described by Pachidis et al. [19], is also implemented in the solver.

3. IMPROVEMENTS FOR TRANSONIC FLOWS

Based on the study performed in this work, it is found that in order to overcome the transonic flow issue in a SLC based throughflow solver following three improvements are essential.

1. **Dual-Solution Approach**: Ability to overcome dual sub- and super-sonic solution by guiding solver towards supersonic flow solution where applicable.
2. **SLC Gradient**: Ability to calculate streamline curvature gradient term in order to avoid singularity at sonic meridional Mach number and high gradient values for transonic flow.
3. **Choked Flow Treatment**: A suitable technique to handle localized choked flow and turning distribution at throat station.

Subsequent sections describe solution procedure for each of the three improvements needed. Solution procedures for "dual-solution" and choked flow treatment are new and developed as part of this work. However, procedure for calculating SLC gradient is leveraged from earlier work done by Denton [1] and Came [2].

3.1. Dual-Solution Approach

As described in Section 2, a throughflow solver finds meridional flow solution by combining the continuity and momentum equation for a streamtube. Usually for subsonic flows it is numerically easier to obtain the solution, as most of the throughflow solvers are coded to seek the subsonic solution. However for high Mach number flows, two possible solutions (subsonic and supersonic) satisfy numerically the governing equations on each streamtube. Sometimes when supersonic solution is desired it becomes harder for the solver to automatically select the nature of flow a priori. Some techniques for seeking the supersonic solution are described in literature [14,20].

In this section a new and relatively simple method is described to help a throughflow solver seek the supersonic solution, when desired. To best of author's knowledge this technique has not been published.

Figure 3 shows the variation of bladerow exit angle with the meanline average turning ($\overline{rC_u}$) or total pressure ($\overline{P_t}$). It is clear that for the same angle level both subsonic and supersonic conditions can be satisfied, with the maximum angle level at the choked flow condition. This is referred as "*dual solution*" domain in the literature [21].

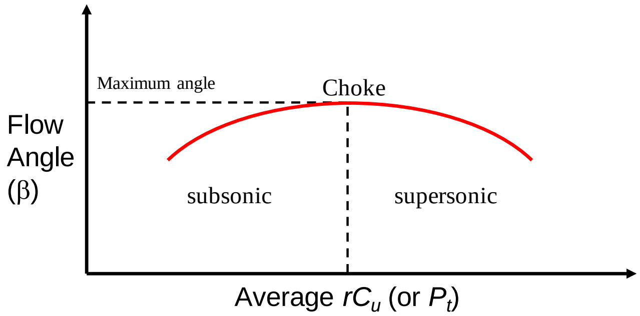

Figure 3: Variation of bladerow exit flow angle (β) with average turning (rC_u) or total pressure (P_t)

The problem of dual-solution will mainly occur when the throughflow solver is used in the analysis mode, because the exit flow angle (β) is prescribed. The design mode can easily overcome this issue as prescribed turning (rC_u) can be made to reflect the desired sub- or supersonic solution. For analysis mode, what we proposed here that for the station where supersonic flow is experienced user should also prescribe the value of average meanline turning ($\overline{rC_u}$) or total pressure ($\overline{P_t}$). This is in addition to the desired exit flow angle (β) distribution, which is a 2D profile. These meanline average turning ($\overline{rC_u}$) or total pressure ($\overline{P_t}$) values are easy to obtain, as these are generally known from 1D meanline preliminary design calculations or design guidelines [22].

However, satisfying both the flow angle distribution and average meanline turning is not possible as this over specifies the problem. And this is also not the intent of this technique. In the analysis mode, the throughflow solver only satisfies the 2D exit flow angle. But the average turning value is used to iteratively guide the solver towards supersonic solution. This is done by adding an additional inner loop which checks whether the calculated average turning value is equal to the prescribed value (within the allowable tolerance limit). If this condition is not met then the difference in turning is used to calculate the delta flow angle ($\Delta\beta$). This delta flow angle ($\Delta\beta$) value is then used to update the prescribed 2D flow angle distribution. Same $\Delta\beta$ value is used from hub to tip in order to preserve the given flow angle distribution. This is done to preserve the shape of angle profile, as this is usually the design intent in the vortexing studies. Usually this loop is very efficient and within few iterations both the flow angle distribution and given meanline turning value is satisfied.

The shift in given flow angle profile is calculated using a relationship of flow angle and average turning. Here is the brief description of the mathematical procedure.

Let's assume that user has supplied the mass averaged turning value at the station where supersonic flow is desired, which is $\left.\overline{rC_u}\right|_{desired}$. Now after solving REE on the station we can calculate the mass averaged turning ($\overline{rC_u}$) for the station using equation

$$\overline{rC_u} = \int_0^1 rC_u \, d\psi \tag{6}$$

Substituting $C_u = U + W_u$ and differentiating Eq.(6) with respect to exit flow angle (β) we obtain

$$\frac{d\overline{rC_u}}{d\beta} = \int_0^1 r\left(\frac{dW_u}{d\beta}\right) d\psi \tag{7}$$

Now at each iteration the difference between the desired and calculated meanline turning is determined using following equation

$$\left.\overline{rC_u}\right|_{desired} - \overline{rC_u} = \Delta\overline{rC_u} \tag{8}$$

Now prescribing the difference in turning ($\Delta\overline{rC_u}$), in terms of difference in flow angle ($\Delta\beta$), as

$$\Delta\overline{rC_u} = \left(\frac{d\overline{rC_u}}{d\beta}\right)\Delta\beta \tag{9}$$

and using Eqs. (7) & (8) we can calculate the desired value by which the flow angle should be updated at each iteration.

$$\Delta\beta = \frac{\Delta\overline{rC_u}}{\dfrac{d\overline{rC_u}}{d\beta}} = \frac{\left.\overline{rC_u}\right|_{desired} - \overline{rC_u}}{\displaystyle\int_0^1 r\left(\frac{dW_u}{d\beta}\right) d\psi} \tag{10}$$

At each iteration the above calculated difference in flow angle ($\Delta\beta$) is added to prescribed flow angle (β_j) distribution. Same value is added at each streamline to preserve the shape of the prescribed angle profile.

$$\left.\beta_j\right|_{n+1} = \left.\beta_j\right|_n + \left.\Delta\beta\right|_n \tag{11}$$

The above described procedure easily rectifies the dual solution issue in throughflow solvers. However, the choice of mass averaged quantity does require special care. Some care should be followed to determine the desired mass averaged turning ($\overline{rC_u}\big|_{desired}$) or total pressure ($\overline{P_t}\big|_{desired}$) data. Also it should not be assumed that the mass averaged quantity can be changed indefinitely to get any form of supersonic or hypersonic solution. The underlying condition for this technique is that meridional Mach number (M_m) should remain under unity, as this would represent maximum mass flow through the streamtube. However, relative Mach number (M_r) can be supersonic. No solution would be possible if user would try to specify mass flow above the choked mass flow limit, as this is unphysical and above described method will not be able to handle this type of input, unless the mass flow is corrected using the technique shown in Section 3.3.

3.2. Streamline Curvature (SLC) Gradient

Examination of Eq.(1) shows that all quantities in the bracket on the RHS side can be assumed constant at a particular iteration. In particular the quantity $\frac{1}{V_m}\frac{\partial V_m}{\partial m}$ is called the streamline curvature gradient term, which is the key component of REE. We can obtain the analytical solution of streamline curvature gradient term shown by Smith [4], as

$$\frac{1}{V_m}\frac{\partial V_m}{\partial m} = \frac{-1}{1-M_m^2}\left[\begin{array}{l}\frac{(U-C_u)}{r(\gamma RT)}\frac{d(rC_m)}{dm}+\left(1+\frac{C_u^2}{\gamma RT}\right)\frac{\sin(\phi)}{r}\\ +\frac{1}{\lambda}\frac{d\lambda}{dm}+\frac{d\phi}{dm}-\frac{1}{R}\frac{dS}{dm}\end{array}\right] \tag{12}$$

It should be noted that since the $\left(1-M_m^2\right)$ term is present in the denominator of Eq.(12) it leads to singularity when meridional Mach number tends to be unity ($M_m \to 1$). This is represented in Figure 4. Marsh [6] mathematically showed that the REE changes its form from elliptic to hyperbolic, when meridional Mach number becomes sonic. This poses significant challenge in solving the REE.

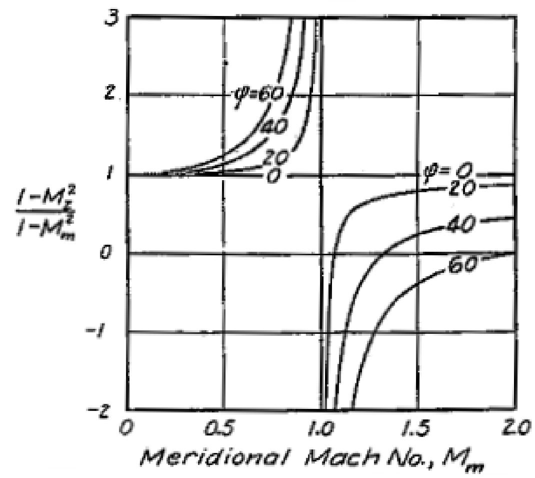

Figure 4: Streamline curvature gradient singularity shown at M_m = 1 (plot reproduced from Smith [4])

Denton [1] showed that the problem of singularity at $M_m \to 1$ could be removed by calculating the streamline curvature gradient term from previous iteration. This is achieved by decoupling the continuity equation with the momentum equation. This technique is implemented in the throughflow solver using the approach shown below.

In order to obtain the streamline curvature gradient contribution we used the finite difference method by accounting contributions from the neighboring upstream station (Eq.(13)). All the quantities at RHS are taken from previous iteration.

$$\frac{1}{V_m}\frac{\partial V_m}{\partial m}\bigg|_{n+1} = \frac{d(\ln V_m)}{dm}\bigg|_{n+1} = \frac{\ln(V_{m,i})-\ln(V_{m,i-1})}{m_i - m_{i-1}}\bigg|_{n} \tag{13}$$

Since the solution of SLC based throughflow codes highly depend on the calculation of the streamline curvature gradient term, high gradients pose a serious robustness issue in the solver. In order to avoid this issue an inner loop is suggested to determine if meridional Mach number approaches unity (or any other maximum limit assigned to the solver) anywhere on a station. In literature it is recommended that the optimum maximum value of meridional Mach number for using numerical treatment should be around $M_{m,max} \geq 0.9$ [20]. If the maximum Mach number condition has been met even at a single streamline on a station, above-mentioned procedure is used to calculate the streamline curvature gradient term for the entire station.

This approach adds considerable robustness to throughflow solver for high Mach number flows until the streamtubes choke. A separate treatment is developed to overcome the challenge of streamtube choking, which is discussed in Sections 3.3 & 3.4.

3.3. Choked Flow Physics

Let's consider very important behavior for the compressible flow, which is the variation of mass flow with pressure ratio. We know that, in converging-diverging nozzle, as the exit pressure is decreased the flow velocity in the throat increases; hence the mass flow increases. Once the sonic flow is achieved, at the throat, then mass flow rate is given by $m = \rho^* u^* A^*$. Where, star (*) indicates properties at sonic conditions. Now even if exit pressure is further decreased the conditions at the throat remain unchanged. The maximum mass flow can be calculated by Eq.(14) using the properties at the throat.

$$m = \rho^* u^* A^* = \rho u A \tag{14}$$

The Mach number at the throat cannot exceed 1. Hence, the mass flow will remain unchanged even if the exit pressure is lowered. This is shown in Figure 5.

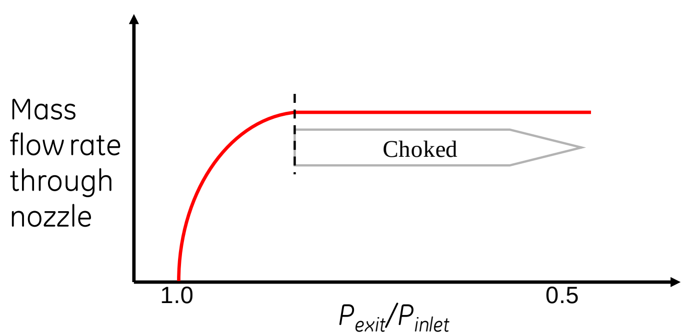

Figure 5: Variation of mass flow with exit pressure. Plot also shows choked flow condition

Once the flow becomes sonic at the throat, disturbances cannot work their way upstream of the throat. Hence, the flow in the convergent section of nozzle no longer communicates with the exit pressure and has no way of knowing that the exit pressure is further decreasing. This situation where the flow goes sonic at the throat and mass flow remains constant even when exit pressure is reduced is called *choked flow*.

In this study we have classified choked flow in a turbomachinery system as *annular* and *localized* choke. Annular flow is classified when all the streamtubes are choked and flow reach the maximum value in the entire flowpath. However, in a streamline curvature approach a flowpath contains many individual streamtubes. It is very common that flow in these individual streamtubes can reach a local maximum value. It is possible that one or more streamtubes experience choked flow, but still the entire flowpath is not choked. This type of flow regime is classified as localized choked flow in this study. It is important to note that if all the streamtubes in the turbomachinery annulus choke then the condition automatically leads to *annular choked flow* as described above. Many turbomachinery applications do not often operate in the annular choked flow mode. On the other hand localized choked flow is encountered very commonly.

Understanding this particular behavior of choked flow is very important for developing a suitable solution scheme for throughflow calculations.

Here is a brief description of *annular choked flow* solution procedure. Detail description is given by Denton [1] & Came [2]. The approach is defined as "*target pressure*" approach. The core basis of this scheme is to invert the solution scheme such that we start with approximately the correct pressure ratio across all bladerows first and hence mass flow rates are determined implicitly with reasonable approximation. This is done by incorporating an inner iteration loop. Once the first approximation is done continuity equation is used to determine the mass flow mismatch. This mass flow mismatch is then used to update the upstream target pressure using equation

$$p_{nte-1}^{n+1} = p_{nte-1}^{n} \left(\frac{\dot{m}_{inlet}}{\dot{m}_{nte}} \right) \tag{15}$$

Eventually, after few loops all bladerows reach closer to desired mass flow and pressure ratio, which leads to converged solution.

As mentioned earlier that most of the turbomachinery applications do not operate when the entire flowpath is choked. However, the above-described procedure is useful to determine the maximum possible mass flow (i.e. chocked mass flow) through the turbomachinery flowpath. This is also useful when the input mass flow is higher than the choked mass flow. In that scenario above mentioned technique is used to iteratively correct the mass flow.

3.4. Localized Choked Flow Treatment

The localized choked flow physics works similar to annular choke flow physics described above. A turbomachinery annulus contains many streamtubes, and since there is no mass transfer between these streamtubes, each streamtube can be considered as the converging-diverging nozzle. The throat location in a streamtube is not fixed, rather variable which is a function of flow solution.

The situation of dealing with localized choking is little easier and hence a new and rather simple technique is described here. The premise of this concept is to preserve the average turning required by the bladerow. Similar to dual-solution approach, described in Section 3.1, this requires providing an additional constraint in terms of average turning ($\overline{rC_u}$). Generally a designer knows the average flow turning required by the bladerow. A sample turning distribution is shown in Figure 6.

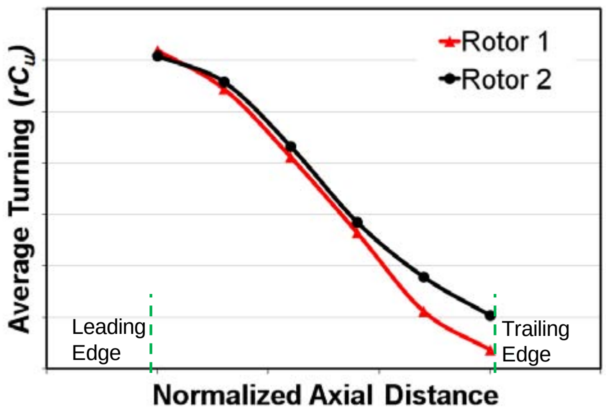

Figure 6: Sample meanline bladerow turning distribution for two rotors

In this procedure, when localized choking occur on a streamtube, we assume that the sonic point lies on the closest upstream station. The condition that leads to local choking is when relative Mach number increases to sonic limit. Generally calculated meridional Mach number or turning (rC_u) distribution using REE at sonic point leads to unphysical values. This is shown as blue line in Figure 7. Once again, this problem will mainly occur when the throughflow solver is used in the analysis mode. In order to rectify this issue we provide additional constraint at the station. We can use the average meanline turning distribution as the additional constraint, but this information is also not sufficient as we can fit many radial distributions through a single average value. We need additional constraint to define a unique radial distribution. For this we make the assumption that the bladerow tip makes the same proportional turning as the meanline distribution. This assumption is not unreasonable for hub strong flows we often experience in many commercial gas turbines. Hence, following equation is used to calculate the tip turning,

$$rC_u\big|_{throat}^{tip} = rC_u\big|_{upstream}^{tip} + \left[\frac{\overline{rC_u}\big|_{throat} - \overline{rC_u}\big|_{upstream}}{\overline{rC_u}\big|_{downstream} - \overline{rC_u}\big|_{upstream}}\right]\left(rC_u\big|_{downstream}^{tip} - rC_u\big|_{upstream}^{tip}\right) \quad (16)$$

In Eq.(16) all the quantities with overbar ($\overline{rC_u}$) represent the average quantity for the station and quantities without overbar represent the tip streamline specific quantity. The quantity in the square bracket is the desired average fractional turning. Once the tip turning is determined the radial distribution can be calculated, for rest of the streamlines, using the assigned average turning for the station ($\overline{rC_u}$). The slope value by which the turning distribution varies from tip to hub may depend on the type of flow and desired vortexing behavior designer want from the bladerows. In this study, for simplicity, a constant slope is assumed.

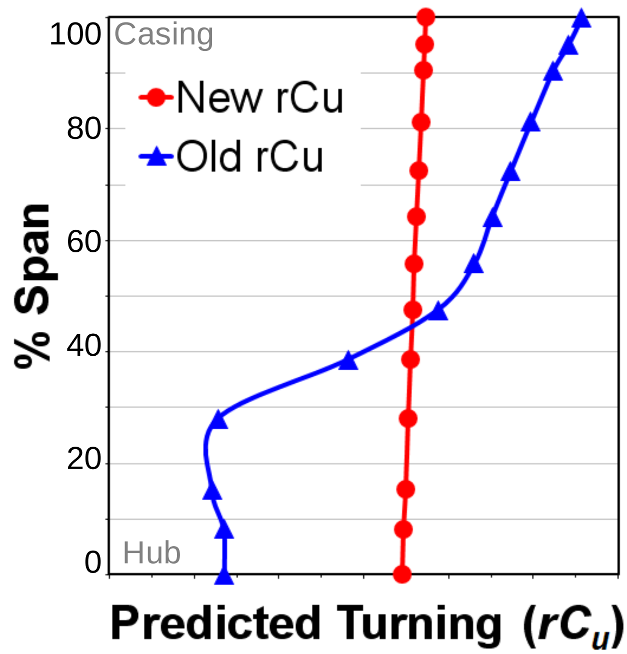

Figure 7: Plot showing the radial distribution of turning (rC_u) for throat station under localized streamtube choking. Red: after correction; Blue: before correction.

Using this constraint of average turning ($\overline{rC_u}$), the flow angle distribution for the throat station is calculated in the analysis mode. This procedure is only applied to the throat station to overcome the localized choked flow issue in the throughflow solver. The final turning (rC_u) distribution after imposing the constraint is given as red line in Figure 7.

4. RESULTS AND DISCUSSION

The improvements described in Section 3 are implemented in a throughflow solver. The improved solver is used to perform throughflow calculations for a range of compressor and turbine cases (both subsonic and transonic). Only few of these results are presented here. For subsonic flow condition, as shown earlier, the streamline curvature gradient term can be defined using the analytical solution (given by Eq.(12)). Eq.(12) is also implemented in the throughflow solver and used to compare the solution against the finite difference based approach (Eq.(13)) shown in Section 3.2. For transonic cases the solution of throughflow solver is compared with the full 3D CFD calculations.

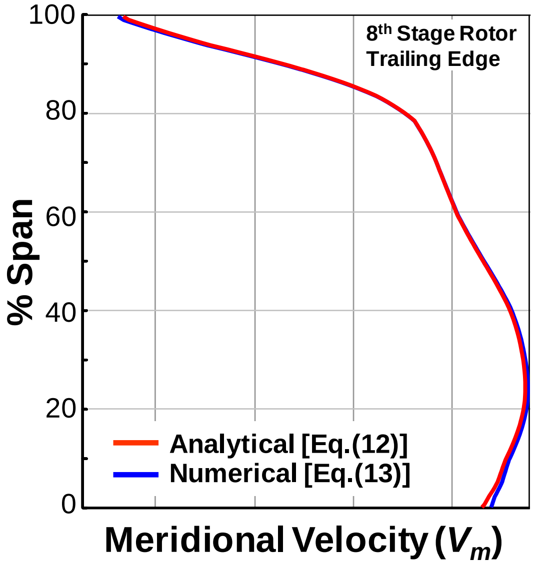

Figure 8: Meridional velocity distribution at TE of 8th stage rotor of the subsonic compressor test case. Results showing solver consistency for a subsonic test case.

Throughflow solver consistency is checked for a subsonic compressor test case, in the analysis mode. The subsonic case used here is an 8-stage aircraft engine compressor. The SLC gradient term treatment, described in Section 3.2, was implemented for all stations regardless of flow Mach number. This solution is compared against the exact solution defined using Eq.(12). Figure 8 shows the fully converged results for the two solution procedure. The meridional velocity distribution at the trailing edge of last stage rotor is presented.

The results clearly show the consistency and accuracy of the results being presented. It can also be inferred that the method of SLC gradient treatment, presented in Section 3.2, does not make any impact on the final solution for the subsonic cases. A small mismatch in the meridional velocity calculations near hub is attributed to high gradient values. Although, all the results were obtained within the prescribed tolerance limit.

Finally the improvements presented here are tested for a transonic case, which require all the three improvements described in Section 3. The transonic case used here is 3-stage low pressure turbine (LPT) of an industrial gas turbine. The focus of this transonic turbine design is to develop high throughflow next-generation gas turbines for IGCC applications. The high mass-flow brings the challenge of pushing high throughflow Mach numbers in last stages of gas turbines very close to $M_m \approx 1$. This brings significant robustness challenges to throughflow solver. Many times the meridional Mach number challenges are coupled with aspect ratio limitations in high annulus area gas or steam turbine last stages [9]. A viable solution for aspect ratio limitation is to provide reasonable spacing between stations, as described by Wilkinson [9].

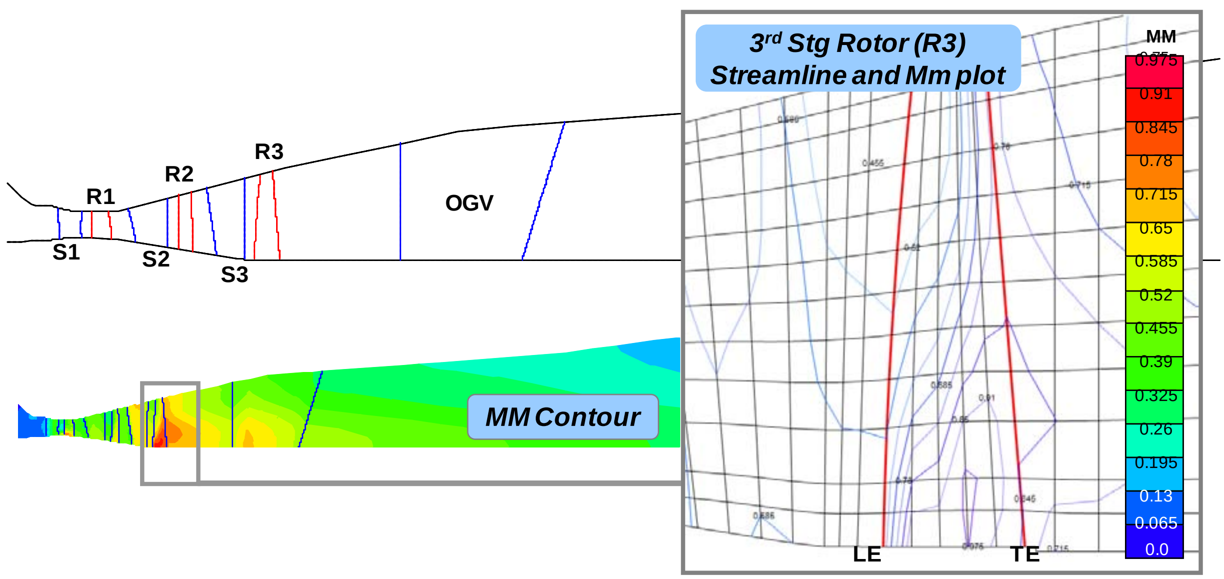

Figure 9: Predicted meridional Mach number shown for a 3-stage transonic turbine test case. Solution obtained using the numerical improvements presented in this paper.

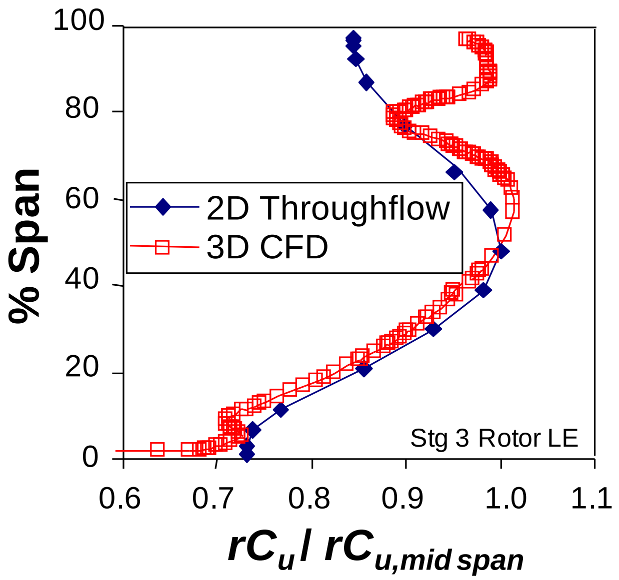

Figure 10: 3D CFD and 2D throughflow comparison for a transonic 3-stage turbine test case

The complete solution of the 3-stage gas turbine test case is shown in Figure 9. The three necessary improvements descried in Section 3 are required to converge this transonic case. The localized streamtube choking treatment helps determine the REE solution for the choked station, which occur in the third stage rotor of the LPT. The hub strong profile with peak meridional Mach number of around ~0.975 is predicted near trailing edge of third stage rotor. Design owners also performed quantitative comparison of throughflow calculations with full 3D CFD solution. The comparison of normalized turning distribution prediction between 3D CFD & 2D throughflow prediction is shown in Figure 10, for the leading edge of last stage rotor. A reasonable comparison is obtained between the two solutions with very good match at mid-span and hub sections. The tip distribution mismatch is attributed to strong tip vortex mixing effect, which were not captured by 2D throughflow solution.

5. CONCLUSIONS

This paper describes various improvements implemented in a streamline curvature based throughflow solver for extending the capability of the solver for transonic and supersonic flows. It is described that three key improvements are required to handle transonic flows in a streamline curvature based throughflow solver:

1) Ability to guide solver to predict supersonic flow solution when "dual-solution" exists
2) Suitable technique to calculate streamline curvature gradient term and to avoid singularity at $M_m=1$ and high gradient values in transonic flow
3) A suitable technique to handle choked flow and calculate turning distribution at the throat station

Details of the treatment for each of the above areas are described in this paper. Solution procedure for "dual-solution" and choked flow treatment is new and developed as part of this work. However, procedure for calculating SLC gradient is leveraged from earlier work done by Denton [1] and Came [2].

Choked flow in turbine blades is categorized as annular choke (when entire turbomachinery annulus is choked) and localized choke (when few streamtubes are choked). Approaches for handling both annular and localized choked flow are described in this work.

Numerical improvements presented here have been validated and tested for a range of compressor and turbine cases (both subsonic and supersonic). It is shown that the numerical improvements presented here resulted in an enhanced version of the 2D streamline curvature based throughflow solver. The new code produces consistent solution for subsonic application with no sacrifice in accuracy of the solver. However, considerable robustness improvements are achieved for transonic and high Mach number applications.

REFERENCES

1. Denton, J. D., Throughflow calculations for transonic axial flow turbines, Trans ASME, Vol 100, Pg 212-218 (1978).

2. Came, P. M., Streamline curvature throughflow analysis, Proc. of First European Turbomachinery Conference, VDI Berichte 1185, Pg. 291 (1995).

3. Wu, C. H., A general theory of three-dimensional flow in subsonic, and supersonic turbomachines of axial, radial and mixed-flow types, Trans. ASME, 1363-1380 (1952).

4. Smith, L. H. Jr., The radial-equilibrium equation of turbomachinery, J. of Engineering for Power, Pg 1-22 (1966).

5. Marsh, H., A digital computer program for the throughflow fluid mechanics in an arbitrary turbo machine using a matrix method, Aeronaut. Res. Counc. Rep. Memo. 3509 (1968).

6. Marsh, H., The uniqueness of turbomachinery flow calculations using the streamline curvature and matrix through-flow methods, J. Mech. Engg. Sci., Vol 12 (6), 376-379 (1971).

7. Novak, R. A., Streamline curvature computing procedures for fluid-flow problems, Trans. ASME, Vol. 89,478-490 (1967).

8. Bindon, J. P., Stability and convergence of streamline curvature flow analysis procedure, Int. J. Num. Methods in Engg., Vol. 7, 69-83 (1973).

9. Wilkinson, D. H., Stability, convergence, and accuracy of two-dimensional streamline curvature methods using quasi-orthogonals, Proc. Instn. Mach. Engrs, Vol. 184, 108-124 (1970).

10. Stow, P., The solution of Isentropic flow, J. Inst. Maths Applications, Vol 9, 35-46 (1972).

11. Hafez, M., South, J. and Murman, E., Artificial Compressibility Methods for Numerical Solutions of Transonic Full Potential Equation, AIAA J., Vol. 17 (8), 838-844 (1978).

12. Murman, E. M. and Cole, J. D., Calculation of plane steady transonic flows, AIAA J., Vol 9 (1), 114-121 (1971).

13. Casey, M. & Robinson, C., A new streamline curvature throughflow method for radial turbomachinery, Proc. ASME Turbo Expo 2008, GT2008-50187 (2008).

14. Sayari N. & Bölcs A., A new throughflow approach for transonic axial compressor stage Analysis, ASME Paper-95-GT-195 (1995).

15. Denton, J. D., Loss mechanism in turbomachines, Trans. ASME, J. Turbomachinery, Vol. 115, pg 621-656 (1993).

16. Koch, C. C., and Smith, Jr., L. H., Loss Sources and Magnitudes in Axial-Flow Compressors,' ASME J. Eng. Power, Vol. 98, pg 411–424 (1976).

17. Adkins, G. G. and Smith, L. H. Jr., Spanwise mixing in axial flow turbomachines, Trans. ASME, J. of Engineering for Power, Vol. 104, 97-110 (1982).

18. Magnus Genrup, Theory of turbomachinery degradation and monitoring tools, Ph.D. Thesis, Lund University, Sweden (2003).

19. Pachidis, V., Templalexis, I. & Pilidis, P., A dynamic convergence control algorithm for the solution of two-dimensional streamline curvature methods, Proc. of ASME Turbo Expo 2009, GT2009-59758 (2009).

20. Aungier, R. H., Turbine Aerodynamics – Axial-flow and radial-flow turbine design and analysis, Pub.: ASME Press, New York (2006).

21. Anderson, J. D., Jr., Fundamentals of Aeromechanics, 2nd Ed., Pub: McGraw-Hill, Inc. (1991).

22. Craig, H. R. M. & Cox, H. J. A., Performance estimation of axial flow turbines, Proc. Inst. Mech. Engg., Vol. 185, Pg 407-424 (1970).

