A Review of Some Key Issues in CFD-Based Throughflow Simulation

Zhihao Hu[1], Qitian Tao[1], and Xiaohua Liu[1,2](✉)

[1] Shanghai Jiao Tong University, Shanghai, China
{alanryen,sjtu-tqt,Xiaohua-Liu}@sjtu.edu.cn
[2] Key Laboratory (Fluid Machinery and Engineering Research Base) of Sichuan Province, Chengdu, China

Abstract. Throughflow simulation method is the backbone of compressor design and analysis. The CFD-based throughflow is a method developed in the recent decades. Comparisons between traditional throughflow method and CFD-based throughflow method are concerned in the present paper, and especially the development of the CFD-based method has been summarized. Some key issues of the CFD-based method are present. Modeling of the discontinuity model at leading and trailing edge, the flow losses and deviation angle are three key issues of the CFD-based method. Empirical flow losses and deviation angle models may not be accurate for advanced compressor throughflow simulation. New method of getting new empirical model is thus discussed. Four discontinuity models, the automatic incidence correlation model, the momentum flux discontinuity model, actuator disk model, and arbitrary discontinuity breakdown model, are presented, compared and the scope of application of each are discussed.

Keywords: Axial compressor · CFD-based Throughflow · Leading and Trailing Edge Discontinuity · Deviation Angle · Loss

1 Introduction

The numerical simulation method is in of the research methods which wildly used in many areas. The design and analysis of the multistage axial compressor involves numerical simulation. Since Wu (1952) proposed the general three-dimensional theory, the numerical simulation of the axial compressor flow field has been simplified the inviscid flow to a two kinds of relative stream surface iteration problem. As shown in the Fig. 1, the two kinds of stream surfaces are the S1 stream surfaces, blade-to-blade plane, and the S2 stream surfaces, meridian plane, respectively.

Throughflow calculation calculate the flow on the meridian surface developed based on the Wu's theory. Nowadays, three kinds of throughflow methods have developed and used, which are the stream function method, the streamline curvature method and CFD-based throughflow method. Throughflow simulation computer programs based on each method for either design and analysis problem came into being.

Z. Jing and X. Zhan (Eds.): ICASSE 2023, LNEE 1153, pp. 159–177, 2024.
https://doi.org/10.1007/978-981-97-0550-4_13

Even with the booming of computation ability and the development of the numerical method, the throughflow method is still the backbone of compressor design and analysis and wildly used in the axial multistage compressor design and analysis process (Denton and Dawes 1999). The aerodynamic design is a repetitive process of design and numerical analysis, it maybe dozens even hundreds round of test and adaptive before the compressor finally complete. Use the full three-dimensional (3D) simulation at the primary stage of design consumes a lot of time, which not only extend the design cycle, but also expends a plenty of computational resources. Therefore, the throughflow simulation is still a useful tool at the primary stage of compressor design (Jin 2011).

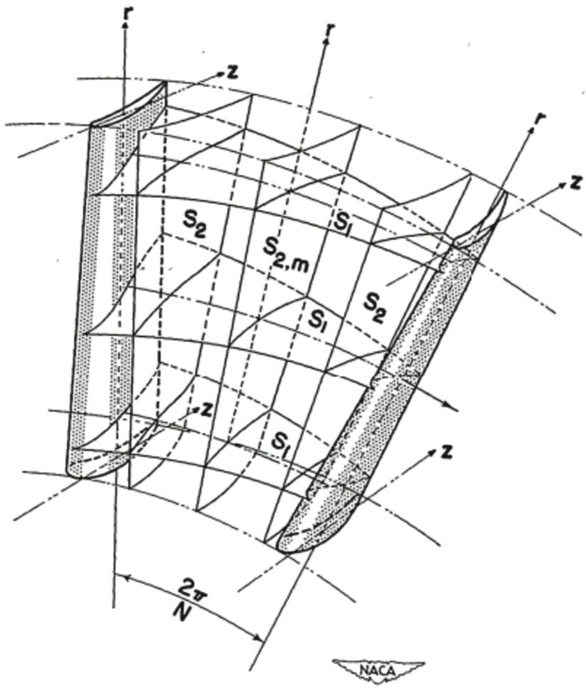

Fig. 1. Intersecting S1 and S2 surfaces in a blade row. (Wu 1952)

Any aero dynamic design or analysis system based on simplifications used hypothesis to simplify the flow field. A lot of experimental data and empirical models are applied for the design or analysis. The more degrees a flow field is simplified the more data are needed. Any kind of high accuracy numerical calculation method may fail owing to the mismatch between current problem with the previous data and empirical models. Meanwhile, some of the empirical models are based on the experimental results of low-load and low-speed cascade experiment. With the stage load and inlet Mach number of the compressors increasing, and the new blade shapes like lean and sweep are being used, the accuracy of the model is challenged. To get higher prediction accuracy of the flow-field in the throughflow, the feasibility of neural network method applied in the acquiring the up-to-date empirical models are discussed.

The CFD-based throughflow method using the circumferential averaged Reynolds averaged Navier-Stokes throughflow equations have a more complete set of control equation comparing to the traditional streamline curvature throughflow method. The more complete the equation a throughflow method uses, the less empirical inputs it demands. Meanwhile, for transonic flow simulation, the CFD-based throughflow is capable of maintaining the stability of the control system. In consequence, the CFD-based throughflow method is more and more welcomed among the compressor designers.

Losses and deviation angle have long been the focus among the throughflow method. Since CFD-based throughflow method uses a different set of equations compared with the classical streamline curvature method, the derivation and models of the losses and deviation angle in CFD-based throughflow method should be introduce.

2 Three Throughflow Methods

2.1 Stream Function Method

Marsh (1966) firstly conducted the stream function throughflow method use a matrix method to solve the axisymmetric function. The stream function method transformed the differential form of stream function into the difference form by the finite difference approximation. The result showed the matrix method can provide a good prediction for the axial velocity profile. However, the stream function method in not wildly used for turbomachinery for its incapable to solve the transonic function and it cannot tell form the subsonic solution and supersonic solution (Denton and Dawes 1999). Marsh also confirmed use the matrix to solve the stream function was limited for it requires a high-speed computer with relatively large storage to conduct the analysis. Both factors setback the further the utility of the matrix method on solving the stream function.

2.2 Streamline Curvature Method

Streamline curvature throughflow method is a very popular throughflow method. The streamline curvature method introduces the concept of streamline curvature, which assume the location of the initial streamline, iterate the radial equilibrium equations and the equations of conservation of mass and energy. Such method has advantages of clear physical concept, simple formulars, and easily to program. In consequence, the streamline curvature method is wildly used in the engineering field. Novak (1967) is the pioneer to develop the throughflow simulation program using streamline curvature method. Several simplify assumption has made in streamline curvature method. (1) flow is adiabatic. (2) flow is steady. (3) the viscous forces are neglected in the momentum equation, the loss is presented by the increase of the entropy. The entropy increases are calculated by the total pressure recovery coefficient and efficiency. (4) the mass flow coefficients are introduced owing to the influence of viscous. (5) the fluid is assumed as the prefect gas. The meridional distribution of streamline is shown in Fig. 2. However, the streamline curvature has a major setback. When calculate supersonic flow field, the channel may be partially or completely choked. The maximum flow rate is only decided by stagnation temperature, stagnation pressure and transaction area. When the mass flow rate does not match such relation, the simulation will be divergent.

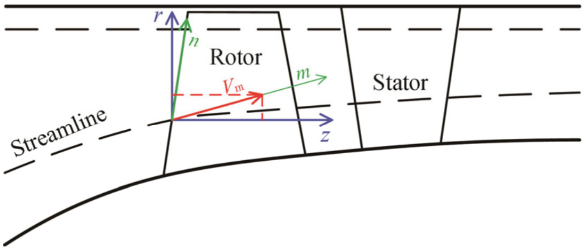

Fig. 2. Meridional distribution of the streamline. (Fei 2021)

Owing to the assumptions of the stream function methods and streamline curvature method, a lot of empirical models are applied. To lessen the dependent of the empirical models, throughflow methods with high order of dimensions of control equations should be investigated.

2.3 CFD-Based Throughflow Method

With the development of computer and numerical method, the CFD-based throughflow simulation program came into being. The CFD-based throughflow method, also called time-marching throughflow. This kind of model is based on Euler equation or Navier-Stokes(N-S) equations and solved by time-marching finite volume method. The CFD-based throughflow method has the following capabilities. (1) this method can predict the mass flow rate at the choke state for the mass flow is calculated form the flow field. (2) it can capture some kind of shock wave. (3) the method can be used for subsonic, transonic, and supersonic simulation simultaneously. (4) Owing to the time-marching method, the throughflow simulation inherent unsteady (Jin 2011).

The first CFD-based throughflow program was conducted by Spurr (1980). Spurr use the time-marching throughflow program to solve the circumferential averaged Euler equation and iterative with a blade-to-blade simulation program to perform simulation on a transonic nozzle. His result matches well with that from the full three-dimensional Euler method. Dawes (1992) modified a three-dimensional Navier-Stokes solver to make it possible to solve blade rows in multistage compressors. Dawes also explored the feasibility of combination of full three-dimensional solver and throughflow solver. Yao and Hirsh (1995) added body force, friction force and introduced losses and deviation angle data for each blade subroutines to a three-dimensional Euler/N-S solver and extend the model to design problem. And they discussed the effect of flow angles and blockage distribution on the blade rows. Body-force term represent blade force and viscous losses in the Euler solver for design and analysis, which is capable of solving subsonic, transonic and supersonic flow with a same set of code as well as capture shock (Damle 1996). Baralon et al. (1998) modeled the profile losses, deviation angle, end-wall skin friction in Euler throughflow model and imported the spanwise mixing model developed by Gallimore (1986). Their model treats the axisymmetric shock as normal blade passage shock through their theoretical and numerical study. They also investigated the method of shock wave capture and proposed a novel method to calculate the cascade blockage factor. They use a three-stage transonic fan to validate the models, the validation result showed the solver is capable of providing a meridional picture of transonic flow for different operating points. Pacciani et al. (2016) develop an axisymmetric Euler solver to naturally fit the secondary and tip leakage effects into throughflow solvers.

Throughflow simulation models based on the circumferential averaged Euler equation neglect the shear stresses. Losses are presented by total pressure drops and entropy changes (Fay et al. 1999). To better represent the actual flow field, more through elements of the flow field should be take into consideration. In consequence, the time-marching Navier-Stokes throughflow method are researched.

Fay et al. (1999) developed a method numerically solve the viscous equations of the meridional flow. They also model the secondary flow due to vortices lead by the tip clearance. And by radial mixing approach, the incoming shear layer vorticity are added. Their model is capture to represent the important feature of viscous flow inside the cascade. Sturmayr and Hirsch (1999) extend a multigrid N-S solver to include throughflow model, investigated shock and associated losses. In their solver, the blades are described by distributed profile blade thickness, deviation angle, and losses. Their model, capture quasi-normal shocks in analysis mode and axisymmetric shock in design mode. Simon et al. (2007) detailed develop a N-S throughflow solver based on the circumferential averaged N-S equation. Their solver use circumferential averaged Reynolds averaged N-S equations. In their throughflow model both Reynolds stress term from Reynolds average and circumferential unevenness term from circumferential average are included. They also discussed the influence of blade force, circumferential stress, and circumferential unevenness term to the flow field. They found the inviscid blade force have the largest impact, the circumferential unevenness secondary to it, and the viscous blade force take the least effect. Taddeo and Larocca (2008) continuing conduct the research on the time-marching throughflow. They applied the distributed loss model and actuator model in the time-marching throughflow. The distributed loss model acts as a compact blade force that turns the flow without producing the unphysical entropy. They also explore the feasibility of combining the time-marching throughflow and empirical models. They found the combination can not only increase the numerical accuracy, but also apply the existing compressor design experiences.

In China, Ji et al. (1999) are the pioneer in researching the CFD-based throughflow. They explored the feasibility of time-marching N-S throughflow in a high load transonic turbomachinery, and verified by design and analysis of an axial supersonic inflow shock rotor. They also announced, the time-marching method inherited unsteady term which can be used for the exploring of the flow stability. Jin (2011) developed a time-marching throughflow solver, which can do both time-marching Euler simulation and N-S simulation. The inviscid blade force is associated with the circumferential pressure, and the inviscid blade force is associated with viscous stress. Jin also declared that at the off-design result from the circumferential averaged N-S simulation differ a lot from the full three-dimensional results owing to the loss and deviation applied for the off-design points have not been fully investigated. Wan et al. (2013) investigated high loaded transonic single-stage fan a four-stage fan flow field performance by both Euler and N-S throughflow solver. By comparing with three-dimensional simulation and experimental results, the throughflow models can provides a credible performance characteristics and better prediction in end-wall regions. Wu (2019) based on the time-marching N-S throughflow, developed the incident, deviation angle, losses and spanwise distribution model at the design point for analysis cases and design point deviation angle and losses model at off-design points. Yang et al. (2019) develop an off-design performance analysis solver, employ novel inviscid blade force model to achieve desired flow deflection, use cubic spline interpolation to solve the discontinuity problems at the leading and trailing edge of the blade rows, and integrated empirical loss model to simulate real three-dimensional viscous force effects. Liu et al. (2021) uses a liner cascade based on N-S throughflow and NUMECA to modify the inviscid blade force, which previously based on the assumption that flow thorough the averaged stream surface with no entropy increasing. Their model decreases 4 times of the prediction error of the adiabatic efficiency than the previous method. Tao et al. (2022) use a distributed inviscid blade force and viscid blade force to reproduce the flow deflection and loss and use uniform inlet 3-D simulation results to interpolate the deviation angle and loss parameters. Then they computed a multistage fan and booster with inlet radial and circumferential distortion.

The time-marching throughflow has developed for about 30 years, and has been widely used in the primary design and analysis phase of the compressor. Several issues have not yet been solved, which will be discussed in the next section.

2.4 Several Issues of CFD-Based Throughflow Method

As conclude by Yang et al. (2017), there are several issues should be further investigated to set up the high accuracy time-marching throughflow method.

Firstly, standard radial blockage factor calculation should be set up to avoid artificial adjustment for getting the accuracy geometry description of the flow field to reflect the physical blockage phenomenon more accurately. At present, the blockage factor has two kinds. One is the circumferential blockage factor, directly derived from the circumferential averaged N-S equation, has a significant influence on the simulation of the mass flow rate and efficient at the choke state. The other is the normal blockage factor, which is based on the geometry of the blade cascade, which can be applied for the mean flow surface throughflow.

Secondly, boundary layer thickness model should be added to get a much accurate reflection of the blockage. Meanwhile, the relation between the boundary layer thickness and the blade force should be further investigated.

Thirdly, for transonic/supersonic flow, the capture of the shock wave should be further investigated, the incapability of capturing shock wave other than the normal shock wave in both design and analysis, which will not being able to simulate all kinds of shock in the flow field.

Fourthly, the flow discontinuity at the leading edge and trailing edge should be further considered. At the leading and trailing edge, instantaneously turn of flow happened. Sever methods have been proposed and discussed. This issue will be fully discussed in next chapter.

3 Leading and Trailing Edge Discontinuity Models

The incidence at the leading edge and deviation at the trailing edge is a key issue of the CFD-based throughflow simulation. Owing to the axisymmetric hypothesis, the CFD-based throughflow simulation cannot sense the blade at the off-blade area to get the pre-rotational velocity. However, once enter the blade region the flow must tangent to the stream surface, the circumferential velocity is constrained by the averaged blade flow surface angle, hence velocity discontinuity at the grid interface of blade leading edge happens. If the blade force term is used to simulate the velocity transition of the leading edge, the sudden change of the volume force will occur, even if the mesh is continuously densified, it will not help. In practice, it is found that as long as the incidence of the incoming flow at the leading edge is greater than 2°, there will be obvious numerical loss. However, the entropy increase corresponding to this kind of loss is non-physical. For the trailing edge of the blade, similar discontinuities will also occur.

3.1 Automatic Incidence Correction Model

Baralon proposed the automatic incidence correction technique (Baralon et al. 1998). In this technique, the blade angles of the first 20% of the chord are modified linearly, parabolically in (x, rθ)coordinates, so that as to adapt progressively to the flow angle upstream of the leading edge. This approach can be defined conceptually as introducing a length scale into the singularity problem. Indeed, when there is very little incidence, the blade force applied to the cell just downstream of the leading edge do not cause any oscillation thereby no entropy. Hence, it appeared natural to solve the singularity approach by applying a progressive change in blade angles, in volume forces. The incidence correction is applied periodically during the time-marching of the computation. The computed flow then adapts itself smoothly to the changes caused by the incidence correction.

The blade shape modifications may have a local effect on the blade loading, but the uncertainty in blade loading is large anyway because the mean line constraint of the throughflow approach is already only an approximation of the real mean flow. Furthermore, the turning that is of interest for performing the work is the turning that is of the flow itself and not of the blade.

Nevertheless, even if the automatic incidence correction is very efficient for subsonic flows, this approach may cause problems for supersonic flows in terms of stream tube are variation.

In case of significant incidence, the modification of the blade geometry at the leading edge may trigger a strong normal shock which is naturally followed by a second one for reasons of stream tube variation. This behavior could be associated with the real blade-to-blade flow of typical transonic rotor blades where, for high mass flows, a first bow shock occurs at the leading edge followed by a weaker normal shock. However, the blade-to-blade bow shock is usually an oblique shock. Therefore, it is believed that the two strong normal shocks obtained in the throughflow solution may correspond to an overproduction of shock losses in this case. It should be emphasized that the problem of successive shocks is depending on the upstream Mach number, the blade geometry and the local incidence. It may not arise in a different configuration.

3.2 Momentum Flux Discontinuity Model

In order to exert a full control on the leading-edge flow without modifying the blade shape, an approach based on a discontinuity in momentum is proposed (Baralon et al. 1998).

The conservation of mass and energy are respected but additional source terms are introduced in the momentum equations in order to control both the flow angle downstream of the discontinuity and generation of entropy.

The technique can be divided in two stages: first, we must derive the target forces that will be included in the momentum fluxes to obtain the desired conditions at the discontinuity. Afterwards, we must derive at the flux calculation level the flow state downstream of the leading edge with the forces included.

This technique has been validated on subsonic and supersonic applications using a quasi-one-dimensional code with blade modelling included. Mach number as high as 3 and incidence angles up to 30° were computed successfully with this 1D code. Computed total pressure and leading-edge flow angle conditions were exactly equal to those specified.

This method solves the leading and trailing edge problems through the discontinuous relationship, but there are still problems in practical application, that is, the iterative solution of flow parameters is difficult in mathematics, especially when the Mach number is close to 1, so it cannot be applied to transonic flow, such as transonic rotor. In addition, this method also requires manual input of additional parameters, namely total pressure loss.

3.3 Actuator Disk Model

An isentropic flow through a linear cascade can be described using 1D Euler equations with a distribution of inviscid blade force that is responsible for turning the flow (Taddei and Larocca 2014a, 2014b). A further source term includes blade blockage effects through a prescribed blockage factor, $h(x) = 1 - \sigma_x \Delta z(x)/(x_{TE}\text{-}x_{LE})$. The equations have the divergence form

$$\frac{\partial W}{\partial t} + \frac{\partial F_c}{\partial x} = Q_i + Q_h \tag{1}$$

where

$$W = \left\{ \begin{array}{c} \rho \\ \rho V_x \\ \rho V_z \\ \rho E^0 \end{array} \right\} F_c = \left\{ \begin{array}{c} \rho V_x \\ p + \rho V_x^2 \\ \rho V_x V_z \\ \rho V_x H^0 \end{array} \right\} Q_i = \left\{ \begin{array}{c} 0 \\ f_{ix} \\ f_{iz} \\ \vec{f_i} \cdot \vec{v} \end{array} \right\} Q_h = -\frac{\rho V_x}{h} \frac{\partial h}{\partial x} \left\{ \begin{array}{c} \frac{1}{V_x} \\ \frac{V_z}{H^0} \end{array} \right\}$$

A given streamline $z(x)$ is tangent to the relative flow velocity and orthogonal to the inviscid blade force vector

$$\frac{\partial z}{\partial x} = \frac{V_z - V}{V_x} = -\frac{f_{ix}}{f_{iz}} \tag{2}$$

Equations (2) close the system in Eq. (1) together with the usual thermodynamic relations. Outside of the cascade, since both the source terms vanish, a uniform flow solution satisfies Eq. (1) and the flow angle is only determined by the upstream conditions. Inside of the cascade, if streamline z(x) is assumed to coincide with the blade camber line, the relative flow angle is constrained by the blade angle. Whenever the incoming flow angle does not match the LE angle, a discontinuity occurs at the LE and unphysical production of entropy will appear in the numerical solution of Eq. (1). Furthermore, the flow leaves the cascade at the TE blade angle and no deviation can be introduced. The most common method to overcome the drawback follows a distributed approach. A modified streamline geometry is assumed, which (i) fits the direction of the incoming flow and (ii) gives the outgoing flow the desired deviation from the TE blade angle.

The main idea underlying the proposed model is to concentrate all the incidence and deviation at the LE and TE, respectively, with no production of unphysical entropy. This can be done by treating the edges as ADs. Due to hyperbolic nature of the system in Eq. (1), the most physically consistent approach to its time-marching solution is to recast it in the form of a conservation law and integrate this form using upwind finite-volume schemes. For the purpose of the present work, a flux difference splitting technique is adopted. The numerical flux vector at a generic interface between two contiguous cells is evaluated using an approximate Riemann solver. Let i + 1/2 be the interface at the LE and TE. A modified Riemann solver is adopted, which places a discontinuity exactly at the interface (Fig. 3). The relations of a quasi-steady compressible AD are assumed to be valid across the discontinuity.

$$
\begin{aligned}
&\rho_1 V_{x1} = \rho_2 V_{x2} \\
&\frac{a_1^2}{\gamma - 1} + \frac{V_{x1}^2 + (V_{z1} - v)^2}{2} = \frac{a_2^2}{\gamma - 1} + \frac{V_{x2}^2 + (V_{z2} - v)^2}{2} \\
&S_1 = S_2 \\
&V_{z2} = v + V_{x2} \tan \beta
\end{aligned}
\tag{3}
$$

where subscripts 1 and 2 refer to the flow states upstream and downstream of the AD, respectively. The first, second, and third equations of the system in Eq. (3) prescribe conservation of mass flow rate, relative total enthalpy, and entropy between states 1 and 2 (note that h = 1 at the LEs and TEs). In particular, the third equation avoids production of entropy across the interface. The fourth equation forces the relative flow in state 2 to take a specified β angle. At the LE, this angle will be the LE blade angle. At the TE, it will be the TE blade angle plus the desired deviation. Equation (3) is combined with the eight compatibility equations of the approximate Riemann solver across waves $V_x - a$ and $V_x + a$ and contact surface V_x. In particular velocity V_z is assumed to be simply conveyed, together with entropy, along contact surfaces. If U stands for the primitive variable vector {a V_x V_z S} T, one obtains a nonlinear system that provides U_1 and U_2 for a given β angle. As in the conventional Riemann problem first- or higher-order approximate solutions can be performed. Second-order accuracy is achieved, in the spirit of ENO approaches, through definition of a linear distribution of the primitive variables over the cells and use of slope limiters. Equation (3) involves the assumption of an unchoked flow across the AD. However, Eq. (3). Can be modified to treat choked flows. Due to its importance, this topic will be addressed in a separate contribution.

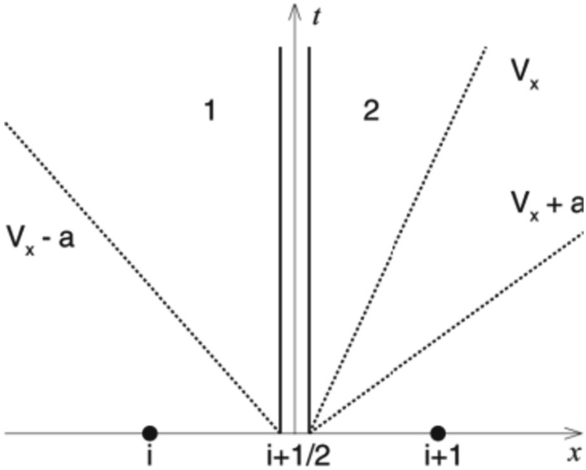

Fig. 3. Representation of an AD at a LE or TE (Taddei and Larocca 2014a, 2014b).

3.4 Arbitrary Discontinuity Breakdown Model

The equations describing gas flows in bladed subdomains and in axisymmetric regions are different. What is more, small perturbances propagate in these subdomains with different velocities (Nigmatullin et al. 1994). Indeed, in bladed regions local acoustic perturbances propagate in the direction perpendicular to grid lines $\xi = const$ with the velocities $\frac{U_\xi}{C_\xi} \pm a$.

It is evident that, first, these boundaries are in common case the discontinuity surfaces and secondly, it is necessary to set enough boundary conditions at these surfaces which depends on number of coming and leaving perturbances. In order words it is necessary to consider an arbitrary discontinuity breakdown problem at these boundaries.

Let us first consider the boundary corresponding to leading edges. A treatment of the boundary conditions means that the usual arbitrary discontinuity breakdown procedure must be placed by the special one described below.

If it is necessary the corresponding right or left limit is used. The parameters at the bladed side will be marked by index 1 and parameters at the axisymmetric subdomain will be marked by index 2. Large left and right parameter (at both sides of the surface) will have the indices L and R.

It's convenient to consider leading triangle drawn at Fig. 5. This triangle may be corresponded to some real triangle in blade-to-blade space. Similar to the trailing edge boundary. Similar to leading edge case one may define trailing edge triangle.

The leading and trailing edge discontinuity processing model is equivalent to modifying the intermediate wave in the general Riemannian problem. The intermediate wave here is no longer a contact discontinuity, but an "intermediate" wave that considers the flow transition and flow picture between L and R, and has the property of entropy conservation under normal conditions. At the same time, the treatment methods for various special cases such as the presence of detached shock wave, leading edge blockage, trailing edge blockage and backflow at the leading edge, so that the leading and trailing edge discontinuity problems can be well handled under any flow state (Fig. 4).

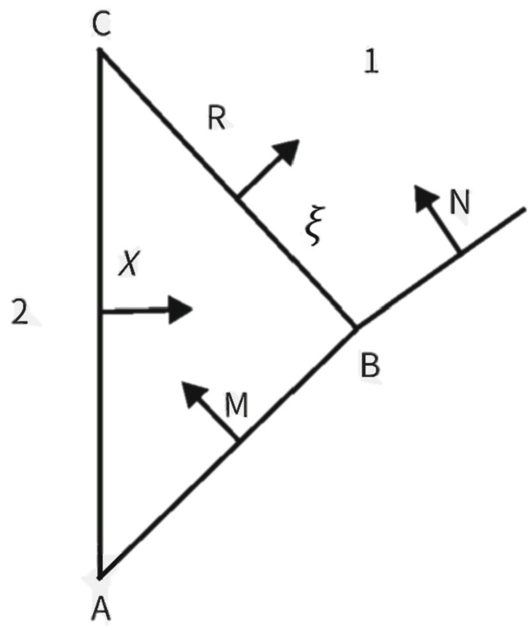

Fig. 4. Arbitrary discontinuity model at the leading edge (Nigmatullin et al. 1994).

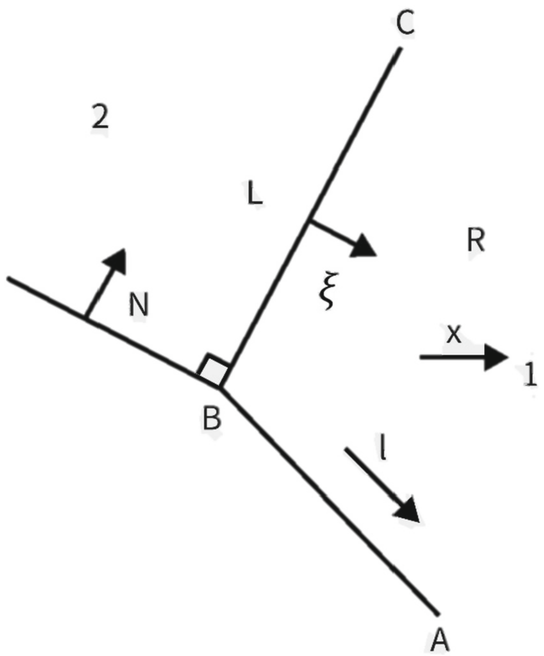

Fig. 5. Arbitrary discontinuity model at the trailing edge (Nigmatullin et al. 1994).

4 Deviation and Losses in CFD-Based Throughflow

Unlike the full three-dimensional simulation, by using throughflow method, the loss and flow deflection can't be simulated directly owing to the reduction of dimension. In consequence, the researchers usually use the empirical model based on the cascade experiment to modify the losses and deviation angle, which represents the deflection.

Most currently developed N-S time-marching throughflow programs are based on the circumferential averaged N-S equations. In the blade passage, the material appearance of the blades does not appear anymore. The effects of the blades are replaced by the force distributions. Usually, flow deflations are modelled by inviscid blade force distribution. While the generated losses are modelled by the distribution of viscous forces. The generated loss is a combination of the blade walls shear stresses and the circumferential stresses (Simon 2007).

4.1 Modeling of Deviation Angle and Losses

At the outlet of blade, the deviation angle is defined as the difference between the flow angle and blade angle. For blade, who has a fixed geometry, the deviation angle represents the flow deflection in the blade region, which represents the work done by the compressor. A good prediction of the deviation angle will lead to much accurate results. As far as the knowledge of author, there are mainly two methods to modify the deviation angle in the time-marching throughflow. One is use inviscid blade force to represent the flow deflation in the blade region (Simon 2007), the other is use an actuator disk model (Taddei and Larocca 2014a, 2014b).

As presented by Simon, in the blade section, several prescriptions have been made to model the deviation angle. Firstly, an orthogonal condition between the blade force and the flow direction are imposed. The flow deflection is generated by the inviscid blade force, which deflects the flow while generating no losses. In the reference frame of the blade, the blade force produces no work. However, a force generates no work and has a component aligned to the flow direction will generate entropy. Therefore, the blade forces are prescribed to be orthogonal to flow. Secondly, as shown in Fig. 6, a relation between the radial component and the circumferential components of the blade force are prescribed by the angle between the normal to the camber surface and the axial direction. Thirdly, the prescribed intensity of the blade force can be obtained by analysis formulation, which take the advantages in the usage of only blade geometry and the robustness in magnitude the blade force than the design formulation and analysis formulation from the circumferential momentum.

Taddei and Larocca (2014a, 2014b) apply the compressible actuator disk equations to modify the evaluation of the deviation angle. In the blade region, by the assumption of the throughflow, the relative flow angle constrains to the meridional streamline direction surface geometrical angle, while in the passage region, no blade force exists. This arises discontinuity of the flow at the leading and trailing edge of the blade. The actuator disk applied at both the leading edge and the trailing edge of the blade instantaneously turns the flow while not producing unphysical entropy. This method avoids unphysical incidence owing to the discontinuity at the leading edge and provides desired deviation angle at the trailing edge, spares handmade modification of the throughflow surface, and allows coping with strong incidence gradient by not specially treating between the inviscid and viscous meridional flows.

Wu (2019) propose a deviation angle model for off-design states, analytically solved the implicit function, which predicts the deviation angle. This model includes no empirical coefficient, and the model explained the relation between the incidence angle and deviation angle, inconsequence, the deviation angle at all spanwise and all working condition can be applied. However, this model is limited to the rotors and stators included in the airfoil database, whose data is already acquired by the experimental.

The generated loss inside the blade passage can be modelled by the distribution of viscous forces. Both the shear stresses acting on the blade wall as well as the circumferential stresses contributes to the loss. The flow losses model and the deviation angle model are respectively the empirical model used for predicting the total pressure loss and outflow direction in the design of compressor. Denton defined the loss in term of entropy increase (Denton 1993). Denton also classified the loss in the compressor into four kinds, blade profile loss due to diffusion on the surface and thickness at trailing edge, end-wall loss due to boundary layer and clearance, leakage loss, and shock loss. While the end-wall loss and leakage loss can be predicted by the loss spanwise distribution model. The shock loss can be solved directly by the time-marching throughflow simulation. In consequence, the profile loss model, which mainly generated form the viscosity friction and flow separation is the only loss model needed to be discussed in throughflow simulation (Wu 2019).

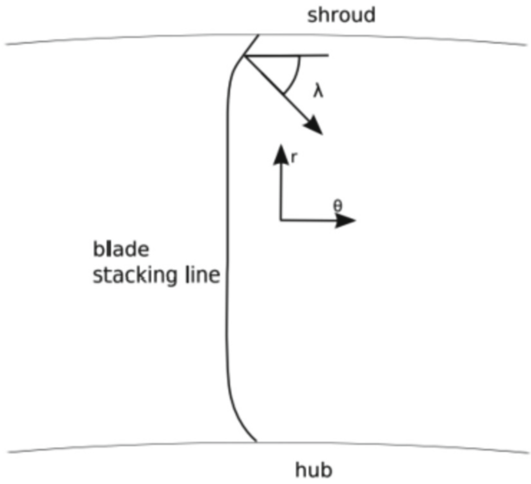

Fig. 6. Definition of the lean angle: camber surface inclination (axial view) (Simon 2007)

Lieblein (1953, 1957) conducted some basic work to predict the profile losses. With the experimental data form low-speed cascade. Lieblein derived the correlation between the momentum thickness and blade diffusion factor, D, or the equivalent diffusion factor, D_{eq}. Lieblein's model is a milestone since then, even though it only valid for low-speed traditional cascade. Koch and Smith (Koch et al. 1976) constructed a more comprehensives losses model based on a large number of cascade, single rotor and compressor experimental result. Their model takes the equivalent diffusion factor, D_{eq}, as a variable and considers the actual flow conditions such as Mach number, Reynolds number, airfoil surface finish, etc. The classical deviation angle model includes the Cater mode (Cater 1950) and Lieblein model (Lieblein 1957). The Cater model is based on the result of the C-4 subsonic airfoil, while the Lieblein model refers the experimental result of the NACA 65 cascade.

By hypothesis two-dimensional flow in the blade-to-blade surface two-dimensional profile losses are generated. The empirical correlations of profiles losses and the deviation angle are computed through design value plus an off-design value.

Deviation angle and losses spanwise distribution model described the effect of the three-dimensional flow phenomenon like end-wall viscosity, leakage flow and secondary flow to the losses and deviation angle Petrovic et al. (2010). Summarized a simplified deviation angle and loss distribution model, which can be applied for reasonably describe the span wise distribution while a leakage of the experimental data.

4.2 Novel Method to Develop Empirical Model

The empirical models discussed above match quit well for classical blade families. However, geometries of modern compressors are generally designed with customized blade, the previous profile loss model may fail to match the up-to-date blade. In consequence, large database with abundant airfoil geometry and flow condition should be used for design and analysis in the throughflow methods. A novel method based the neural network to get the empirical model of deviation angles and losses will be discussed in this section.

A novel approach to get a more compete empirical model is by neural network (Schmitz et al. 2011; Li et al. 2021). The basic idea of this method can be summarized in 3 steps. Firstly, set up a database, the database includes the geometry of airfoil and flow field paraments, both geometry and flow field data form experimental and numerical simulation can be used. Secondly, surrogate models are trained by neural network to obtain the correlation of between loss or deviation angle with geometry and flow field data to get new empirical model. Thirdly, the new empirical models are combined with throughflow simulation models to get the prediction of the flow paraments.

Schmitz et al. (2011) use a database includes 106 randomly created airfoils, and for each airfoil both geometry paraments like the metal angel, maximum profile thickness, leading edge paraments, curvature parameters, chord length, and flow field paraments like relative Mach number, relative inflow angle and Reynolds number are included. Figure 7, show the distribution of the airfoil losses over the Mach number and the inflow angle. They develop a novel method to get loss and deviation angel prediction model for compressor. They pointed out that, the accuracy can be increased globally or just for small areas by adding more sample points to the training. They also confirm that secondary effect flow such as the end-wall boundary layer and spanwise mixing should be further investigated.

Li et al. (2021) use genetic algorithm-back propagation neural network (GA-BGNN). Loss prediction surrogate model at design and off-design condition of the compressor blade span was developed by GA-BGNN. They established a database, which contains several sets of blade element geometry and blade performance data. Considering different working condition of rotor and stator, different database and surrogate model are used. They also take into consideration of the mechanism of different losses generated between the rotor and stator. The prediction results from the surrogate model are compared with the result from traditional empirical results and experimental data, shows the surrogate model matches much better than the traditional model.

With the help of the neural network, a database contains a hug amounts data of the airfoil can be applied for the construction of the empirical model of loss and deviation angle with higher accuracy, which will better reflection of the flow in the blade region and improve the performance prediction accuracy.

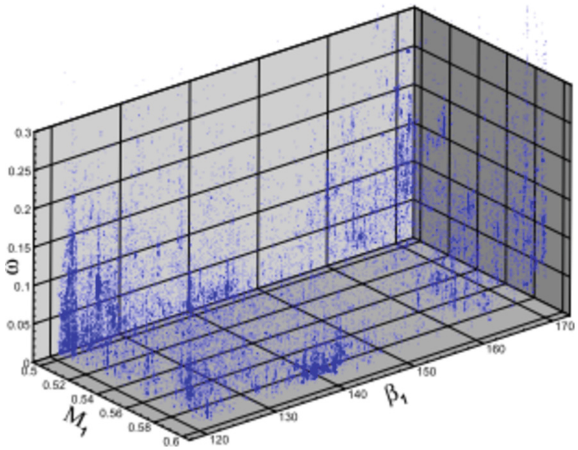

Fig. 7. Distribution of the airfoil losses over the Mach number and the inflow angle (Schmitz et al. 2011)

5 Conclusion

This paper detailed reviewed the development and several issues of the CFD-based throughflow simulation method. The discontinuity at the leading and trailing edge, modeling of loss and deviation are major concerns of the CFD-based throughflow. Several conclusions have been made.

1. Time-marching throughflow method has advantages over the streamline curvature and stream function method in including capable of predicting mass flow rate at the choke state, capturing shock wave, able to perform transonic simulation and inherent unsteady.
2. Treatment of blockage factor, shock waves capture, discontinuity at the leading and trailing edge of the blade rows are some key issues of the accuracy of the time-marching throughflow simulation. These issues should be further investigated and common sense may be discussed among researchers.
3. The discontinuity models at the leading and trailing edge of the blade rows have long been concerned in the throughflow simulation. Automatic incidence correlation model is limited by the incidence, which cannot be too lager, and limited in the subsonic condition. Momentum flux discontinuity model have problem in transonic flow and require manual input of additional total pressure loss parameters. The actuator disk model cannot be applied to blocking conditions and describe the shock wave. Arbitrary discontinuity breakdown model is an analytical solution to the leading and trailing edge discontinuity problem under the physical flow picture, and can be applied to the calculation of various working conditions.
4. Modeling the losses and deviation angle has long been concerned in throughflow simulation. Previous researchers use the cascade experimental data to get the empirical model of Loss and deviation angle. However, with the development of the airfoil and compressor design methodology, those empirical models are not suitable for the modern throughflow simulation. Neural network method with a large airfoil and

flow parameter are maybe a novel approach to get empirical models apply for model throughflow simulation.

Acknowledgements. This work is partially supported by National Nature Science Foundation of China (No. 51976116), Natural Science Fund of Shanghai (No. 19ZR1425900), and the Open Research Subject of Key Laboratory (Fluid Machinery and Engineering Research Base) of Shanghai Province (No. Szjj2019-022), which are greatly acknowledged. The authors also gratefully acknowledge the helpful comments and suggestions of the reviewers, which have improved the presentation.

References

Wu, C.: A General Theory of Three-Dimensional Flow in Subsonic and Supersonic Turbomachines of Axial-, Radial-, and Mixed-Flow Types. NACA TN 2604, pp. 1–90 (1952)

Denton, J.D., Dawes, W.N.: Computational fluid dynamics for turbomachinery design. Proc. Inst. Mech. Eng. C J. Mech. Eng. Sci. **213**(2), 107–124 (1999). https://doi.org/10.1243/0954406991522211

Jin, H.: Application of circumferential average method in multistage axial fan/compressor design and analysis. Dissertation, Beihang University (2011)

Harry, M.: A Digital Computer Program for the Through-Flow Fluid Mechanics in an Arbitrary Turbomachine Using a Matrix Method. London (1966)

Novak, R.A.: Streamline curvature computing procedures for fluid-flow problems. J. Eng. Power **89**(4), 478–490 (1967)

Spurr, A.: The prediction of 3D transonic flow in turbomachinery using a combined throughflow and blade-to-blade time marching method. Int. J. Heat Fluid Flow **2**(4), 189–199 (1980)

Dawes, W.N.: Toward improved throughflow capability: the use of three-dimensional viscous flow solvers in a multistage environment. J. Turbomach. **114**(1), 8–17 (1992)

Yao, Z., Hirsch, C.: Throughflow model using 3D Euler or Navier-Stokes solver. In: Turbomachinery - Fluid Dynamic and Thermodynamic Aspects, Germany (1995)

Damle, S.V.: Throughflow method for turbomachines using Euler solvers. In: 34th Aerospace Sciences Meeting and Exhibit. (1996)

Damle, S.V., Dang, T.Q., Reddy, D.R.: Throughflow method for turbomachines applicable for all flow regimes. J. Turbomach. **119**(2), 256–262 (1997)

Baralon, S., Eriksson, L.E., Hall, U.: Validation of a throughflow time-marching finite-volume solver for transonic compressors. In: ASME 1998 International Gas Turbine and Aeroengine Congress and Exhibition (1998)

Baralon, S., Eriksson, L.E., Hall, U.: Evaluation of higher-order terms in the throughflow approximation using 3D Navier-Stokes computations of a transonic compressor rotor. In: ASME 1999 International Gas Turbine and Aeroengine Congress and Exhibition (1999)

Gallimore, S.J.: Spanwise mixing in multistage axial flow compressors: Part II—throughflow calculations including mixing. J. Turbomach. **108**(1), 10–16 (1986)

Gallimore, S.J., Cumpsty, N.A.: Spanwise mixing in multistage axial flow compressors: Part I—experimental investigation. J. Turbomach. **108**(1), 2–9 (1986)

Pacciani, R., Filippo, R., Michele, M., Andrea, A., Strfano, C., Federico, D.: A CFD-based throughflow method with an explicit body force model and an adaptive formulation for the s2 stream surface. Proc. Inst. Mech. Eng. A J. Power Energy **230**(1), 16–28 (2016). https://doi.org/10.1177/0957650915607091

Fay, G., Lawerenz, M., Przewozny, H.: Calculation of pitch averaged viscous flow in an annular compressor cascade. Comput. Methods Exp. Meas. IX **21**, 185–194 (1999)

Sturmayr, A., Hirsch, C.: Throughflow model for design and analysis integrated in a three-dimensional Navier-Stokes solver. Proc. Inst. Mech. Eng. A J. Power Energy **213**(A4), 263–273 (1999)

Simon, J.F., Thomas, J.P., Leonard, O.: On the role of the deterministic and circumferential stresses in throughflow calculations. J. Turbomach. Trans. ASME **131**(3), 1–12 (2009)

Simon, J.F., Leonard, O.: Modeling of 3-D losses and deviations in a throughflow analysis tool. J. Therm. Sci. **16**(3), 208–214 (2007)

Simon, J.F.: Contribution to throughflow modelling for axial flow turbomachines. Dissertation. Ingenieur Civil Electro-Mecanicien (2007)

Taddei, S.R., Larocca, F.: Axisymmetric design of axial turbomachines: an inverse method introducing profile losses. Proc. Inst. Mech. Eng. A J. Power Energy **222**(A6), 613–621 (2008)

Taddei, S.R., Larocca, F.: CFD-based analysis of multistage throughflow surfaces with incidence. Mech. Res. Commun. **47**, 6–10 (2013)

Taddei, S.R., Larocca, F.: An actuator disk model of incidence and deviation for RANS-based throughflow analysis. J. Turbomach. **136**(2), 1–9 (2014)

Ji, L., Meng, Q., Zhou, S.: Time-marching method for throughflow computation of turbomachinery. J. Aerosp. Power **14**, 23–26 (1999)

Wan, K., Jin, H., Jin, D., Gui, X.: Influence of Non-Axisymmetric Terms on Circumferentially Averaged Method in Fan/Compressor (2013). https://doi.org/10.1007/s11630-013-0586-1

Wu, D.: Throughflow moldelling and validation of multistage axial flow compressor. Dissertation, Shanghai Jiao Tong University (2019)

Yang, C., Wu, H., Yang, J., Ferlauto, M.: Time-marching throughflow analysis of multistage axial compressors based on a novel inviscid blade force model. Proc. Inst. Mech. Eng. G J. Aerosp. Eng. **233**(14), 5239–5252 (2019). https://doi.org/10.1177/0954410019840588

Liu, X., Wan, K., Jin, D., Gui, X.: Development of a Throughflow-Based Simulation Tool for Preliminary Compressor Design Considering Blade Geometry in Gas Turbine Engine (2021). https://doi.org/10.3390/app11010422

Yang, J., Wang, C., Wang, D., Shao, F., Yang, C., Wu, H.: Time marching based throughflow method: current status and future development. Chin. J. Aeronaut. (**9**), 520996–520996 (2017). https://doi.org/10.7527/s10000-6893.2017.620996

Tao, Q.: An Aerodynamic performance calculation method of fan and booster with inlet distortion. Dissertation, Shanghai Jiao Tong University (2021)

Tao, Q., Jin, H., Liu, X., Zhu, Z.: A fast computational method of the aerodynamic performance of fan and booster with inlet distortion (2022). https://doi.org/10.1177/09544062221115346

Barlon, S., Eriksson, L.E., Hall, U.: Validation of a throughflow time-marching finite-volume solver for transonic compressor. ASME. 98-GT-47 (1998)

Simon, J.F.: Contribution to throughflow modelling for axial flow turbomachines. Dissertation. University of Liege (2007)

Persico, G., Rebays: A penalty formulation for the throughflow modeling of turbomachinery. Comput. Fluids **60**(10), 86–98 (2012). https://doi.org/10.1016/j.compfluid.2012.03.001

Taddei, S.R., Larocca, F.: CFD-based analysis of multistage throughflow surfaces with incidence. Mech. Res. Commun. **47**(47), 6 (2013). https://doi.org/10.1016/j.mechrescom.2012.10.005

Taddei, S.R., Larocca, F.: An actuator disk model of incidence and deviation for RANS-based throughflow analysis. J. Turbomach. **136**(2), 021001 (2014). https://doi.org/10.1115/1.4025155

Nigmatullin, R.Z., Ivanov, M.J.: The mathematical models of flow passage for gas turbine engines and their components (1994)

Denton, J.D.: Loss mechanisms in turbomachines. J. Turbomach. **115**(4), 621–656 (1993)

Lieblein, S.: Analysis of experimental low speed loss and stall characteristics of two dimensional compressor blade cascades (1957)

Lieblein, S., Schwenk, F.C., Broderick, R.L.: Diffusion factor for estimating losses and limiting blade loadings in axial-flow-compressor blade elements (1953)

Koch, C.C., Smith, L.H.: Loss sources and magnitudes in axial-flow compressors. J. Eng. Power **98**(3), 411–424 (1976)

Cater, A.D.S.: The low-speed performance of related aerofoils in cascade, London (1950)

Petrovic, M.V., Wiedermann, A., Manjac, M.B.: Development and validation of a new universal through flow method for axial compressors. Proc. Inst. Mech. Eng. A J. Power Energy **224**(6), 869–880 (2010). https://doi.org/10.1243/09576509JPE991

Schmitz, A., Aulich, M., Nicke, E.: Novel approach for loss and flow-turning prediction using optimized surrogate models in two-dimensional compressor design. In: ASME 2011 Turbo Expo: Turbine Technical Conference and Exposition (2011). https://doi.org/10.1115/GT2011-45086

Li, J., Teng, J., Zhu, M., Qiang, X.: Loss prediction of axial compressors using genetic algorithm–back propagation neural network in throughflow method. Proc. Inst. Mech. Eng. G J. Aerosp. Eng. **0**(0), 1–13 (2021). https://doi.org/10.1177/09544100211041490