Validation of a novel mixing-plane method for multistage turbomachinery steady flow analysis

Du Pengcheng\textsuperscript{a}, Ning Fangfei\textsuperscript{b,}\textsuperscript{*}

\textsuperscript{a}School of Aeronautic Science and Engineering, Beihang University, Beijing 100083, China
\textsuperscript{b}School of Energy and Power Engineering, Beihang University, Beijing 100083, China

KEYWORDS
Centrifugal compressor; Flux conservation; Mixing-plane method; Reverse flow; Turbomachinery

Abstract The steady calculation based on the mixing-plane method is still the most widely-used three-dimensional flow analysis tool for multistage turbomachines. For modern turbomachines, the trend of design is to reach higher aerodynamic loading but with still further compact size. In such a case, the traditional mixing-plane method has to be revised to give a more physically meaningful prediction. In this paper, a novel mixing-plane method was proposed, and three representative test cases including a transonic compressor, a highly-loaded centrifugal compressor and a high-pressure axial turbine were performed for validation purpose. This novel mixing-plane method can satisfy the flux conservation perfectly. Reverse flow across the mixing-plane interface can be resolved naturally, thus making this method numerically robust. Artificial reflection at the mixing-plane interface is almost eliminated, and then its detrimental impact on the flow field is minimized. Generally, this mixing-plane method is suitable to simulate steady flows in highly-loaded multistage turbomachines.

© 2016 Chinese Society of Aeronautics and Astronautics. Production and hosting by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/4.0/).

1. Introduction

Multistage turbomachinery flow is inherently unsteady due to the relative motion of adjacent rotor and stator blade rows. However, at ordinary operation point, the time-averaged flow field in each blade passage in either rotor or stator blade row is almost the same in the relative frame of reference fixed to the corresponding blade row under consideration. Thus, quasi-steady calculation can be performed based on single blade passage, which can capture the major flow features but with much less computational effort than full unsteady simulation. The simplest way to obtain a steady state solution is the frozen rotor technique. It keeps the relative positions of the rotor and stator fixed, and thus the results are position-dependent.

\textsuperscript{*} Corresponding author. Tel.: +86 10 82338270.
E-mail addresses: dupengch22@163.com (P. Du), fangfei.ning@buaa.edu.cn (F. Ning).

Peer review under responsibility of Editorial Committee of CJA.

http://dx.doi.org/10.1016/j.cja.2016.10.005

This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/4.0/).

Please cite this article in press as: Du P, Ning F Validation of a novel mixing-plane method for multistage turbomachinery steady flow analysis, Chin J Aeronaut (2016), http://dx.doi.org/10.1016/j.cja.2016.10.005
solution can be obtained. Since the mixing-plane method was proposed, it has been the most popular way in the simulations of multistage turbomachinery flows, and it will still be the favorite method for engineers in the next decade.

In general, there are two key steps to realize a mixing-plane method. First, an averaging technique is required to obtain the circumferentially averaged flow state at each side of the interface. Second, the circumferentially averaged flow state must be transferred through the interface. Various detailed treatments of the above procedures are proposed and are still under further improvement in the literature.2–7

For the first key step in mixing-plane method, the straight and simplest way is to circumferentially average the flow variables directly. However, first, there is no consensus on what flow variables should be circumferentially averaged.8–10 The principal flow variables such as density, velocity and pressure are often selected, while the total pressure, total temperature and flow angles can also be circumferentially averaged to model each successive blade row more like individuals. Second, actually there is no suitable weighted averaging method that can deal with the situation when there exist both forward and inverse flows across the interface. Besides, due to the non-linear relationship between the flow variables and the flux, the total flux calculated from circumferentially averaged flow variables may be deviated from the total flux of the non-uniform flow. Thus the conservation of mass, momentum and energy would not be strictly satisfied. To solve this problem, the flux-based circumferential averaging technique11 has been widely used. However, there is a root square operation in solving the mixed-out static pressure from the averaged fluxes, and thus divergence would be encountered during the initial stage of calculations. Another shortcoming of this method is that it cannot allow the reverse flow across the mixing-plane interface directly. Recently, Wang12 improved this method. The primitive variable variations are directly determined from the flux difference between the two sides of the interface, and then the numerical stability is enhanced and reverse flow across the interface can be handled. Later on, Ning13 presented his mixing-plane model which can also ensure the flux conservation but based on a more physical background.

For the second step of a mixing-plane method, i.e., the information exchange across the interface, the most important issue is to prevent wave reflections from the interface. Giles9 proposed a 1D characteristic-based nonreflective boundary condition (NRBC) in which the circumferential and radial variations are neglected. Then, Saxer and Giles10 extended this NRBC to quasi-3D flow based on the assumption that the radial variations are small compared to circumferential variations. Anker et al.,11 extended this theory to fully 3D NRBC. The 1D NRBC is most widely used due to its simplicity and robustness.1–6 However, it does not always work well when the circumferential nonuniformity is strong and/or the adjacent blade rows are closely spaced, and thus the non-physical reflections from the interface would be notable. To remit this problem, Ning13 proposed a method that a constant-radius buffer layer was added at each side of the interface to damp the outgoing waves.

In this paper, the basic theory of this novel mixing-plane method is first stated and then three typical test cases are given to demonstrate the superiority of this mixing-plane method.

2. Mixing-plane governing equations

From the authors’ point of view, the mixing-plane method should not just be a pure numerical procedure to transfer the circumferentially averaged flow variables across the interface. A physical correspondence for this pitchwise mixing can be found, i.e., we can just make the gap between the two adjacent blade rows long enough so as to mix out all the non-uniformities (as shown from Fig. 1(a) and (b)), while the spanwise mixing is assumed to be suspended in the “extended mixing region”. Therefore, for a fully converged flow field in the case as shown in Fig. 1(b), we can find such an intermediate position where the flows are pitchwise uniform. At this position, if we cut out an infinitely thin slice as denoted by two lines “ml” and “mr” in Fig. 1(b), the following governing equations expressed in cylindrical coordinate system hold

$$\frac{\partial \mathbf{Q} }{\partial t} + \mathbf{F}_{\text{int}} - \mathbf{F}_{\text{ml}} = 0$$  \hspace{1cm} (1)

where the mixed-out conservative variables $\mathbf{Q}$ in the slice and the advective flux $\mathbf{F}$ are expressed as,

$$\mathbf{Q} = [ \rho, \, \rho v_r, \, \rho v_{\phi}, \, \rho v_z, \, \rho e ]^T,$$

$$\mathbf{F} = [ \rho U, \, \rho U v_r + n_p, \, \rho U v_{\phi} + n_p, \, \rho U v_z + n_p, \, \rho U H ]^T$$

with $r$ being the pseudo time, $\rho$ the density, $(v_r, v_\phi, v_z)$ the absolute velocity components expressed in cylindrical coordinate $(x, \theta, r)$, $e$ the total energy, $U = n_r v_r + n_\phi v_\phi + n_z v_z$ the advective velocity normal to the blade row interface, $p$ the static pressure and $H$ the total enthalpy. The unit vector $\mathbf{n} = (n_r, n_\phi, n_z)^T$ denotes the normal direction of blade row interface, and $n_\phi$ is actually equal to zero because the interface is a revolution surface.

The solution of Eq. (1) is the intermediate mixed-out state. As shown in Fig. 1(b), the advective fluxes from surface “L” to “ml” and from “R” to “mr” are conservative if the flow is inviscid and adiabatic, and we can thus obtain the following equations:

$$\mathbf{F}_{\text{int}} = \frac{1}{S_L} \int_L \mathbf{F} \cdot d\mathbf{S}$$  \hspace{1cm} (2)

$$\mathbf{F}_{\text{ml}} = \frac{1}{S_R} \int_R \mathbf{F} \cdot d\mathbf{S}$$  \hspace{1cm} (3)

where $S$ is the surface area at the interface, and the subscripts “L” and “R” denote the exit of upstream blade row and the inlet of downstream blade row. Therefore, at each time-marching step for the passage flows, suppose that the boundary conditions at both boundaries “L” and “R” have been defined, and the advective flux across these two boundaries can be evaluated, and thus the flux terms in Eq. (1) are known. Subsequently, the intermediate fully mixed-out state can be updated by solving Eq. (1) through an individual time-marching loop (the time stepping can be synchronized with that for the passage flows). As Eq. (1) converges, the advective flux at the blade row interface is conserved. The solution of Eq.
The method is almost identical to the mixing-plane method proposed by Wang. It is applicable for steady simulation of multistage turbomachinery flow. However, in many cases, especially when the blade loading is relatively high and/or the blade rows are closely spaced, the perturbation waves propagating toward either side of the blade row interface are far from linear, and thus the adopted 1D characteristic boundary condition methods on the basis of local linearization of flow equations will lead to wave reflections which are non-physical and will contaminate the solution. The convergence of the mixing-plane governing equation would be influenced by the artificial reflections, and thus the solution of Eq. (1) would be paradoxical. It may produce a solution that the flow is not fully mixed-out or the flow is over mixed. The entropy generation at the mixing-plane interface maybe deviates from that generated during the corresponding physical mixing process. And it depends on the degree of artificial reflections at the interface and may have different trends from case to case.

To alleviate this problem, constant-radius buffer layers are added at each side of the interface (Fig. 1(c)). Then the following governing equations are solved in each buffer layer:

$$\frac{\partial Q_{bl}}{\partial t} + \frac{\partial F_{bl}}{\partial x} + \frac{\partial G_{bl}}{\partial \theta} - \alpha \frac{\partial Q_{bl}}{\partial \theta} = 0$$

where

$$Q_{bl} = \left[ \rho, \rho U, \rho v_\theta, \rho V, \rho e \right]^T,$$

$$F_{bl} = \left[ \rho U, \rho U^2 + p, \rho U v_\theta, \rho UV, \rho U H \right]^T,$$

$$G_{bl} = \left[ \rho v_\theta, \rho v_\theta U, \rho v_\theta^2 + p, \rho v_\theta V, \rho v_\theta H \right]^T.$$

In the above formulations, the subscript “bl” denotes “buffer layer”, $V = l_x v_x + l_\theta v_\theta + l_y v_y$ with the unit vector $I = (l_x, l_\theta, l_y)$ being orthogonal to both the normal and tangential direction of the interface, and $\omega$ is the pitch speed of the blade row connected to the corresponding buffer layer. The above equations can be derived from traditional 3D flow equations but with the coordinate “x” in Eq. (4) interpreted as that normal to the blade row interface. This method is similar to the so-called perfect matching layer (PML) or absorbing layer, i.e., by adding a PML in which the outgoing waves can be quickly damped so that the non-reflecting boundary conditions are easier to be specified at the outward boundary. It is also noted that the dynamic process described by Eq. (4) is essentially isentropic unless there is shock wave propagating into the buffer layer. Therefore, the buffer layer would not induce extra loss, while the additional loss due to the outgoing shock wave (if any) can be fittingly taken into account.

To ensure the conservative property, the integral form of Eq. (4) is actually solved. The solution process is also synchronized with that for the passage flow. The mesh of the buffer layer can be constructed so that the mesh lines are along the axial and tangential directions. And the mesh size of each buffer layer is equivalent to the size of the block mesh which it connects to. The length of the extended buffer layer influences the effectiveness of the damping of the outgoing waves and thus the degree of reflections at the interface. The longer the buffer layer is, the weaker the reflection will be. Many numerical experiments have shown that when the length of each buffer layer is larger than 0.5 times the pitch of the connected blade row, the isentropic flow solution would not be influenced by the length of the buffer layer. Thus, the length of the buffer layer is set as 0.5 times the pitch of the connected blade row as default in our CFD code. In the buffer layer, the spatial discretization is simple and the solution process is less expensive. The interface between the two adjacent buffer layers (Fig. 1(c)) is now the “new” interface on which the 1D NRBC is applied.

For 3D problems, the application of this model is straightforward, i.e., we can apply the model individually on each revolution surface stacked along spanwise direction. In our in-house CFD code which is named Multiblock Aerodynamic Prediction (MAP) code, the whole procedure is contained in a module which includes the construction of the buffer layer and the solution of the governing equations. It usually needs less than 10% of total computational time for the mixing-plane model.
3. Validation test cases

The description and validation of the MAP code can be found in Ref. 4. It solves the integral form of the governing equations which are discretized in space using a cell-centered finite-volume method. The advective fluxes are evaluated using the low-diffusion flux-splitting scheme\(^{13}\) coupled with Monotone Upstream-Centered Schemes for Conservation Laws (MUSCL) interpolation to obtain high-order spatial accuracy. The diffusive fluxes are solved using traditional central differencing. The one-equation Spalart–Allmaras turbulence model\(^{14}\) is used for turbulent flows, which is discretized and solved in a coupled manner with the mean flow equations. Message passing interface (MPI) is used to parallelize the code. The discretized system is solved using the so-called matrix-free Gauss–Seidel algorithm.

To demonstrate the characteristics of this new mixing-plane method, three typical turbomachinery test cases are calculated.

![Fig. 2](image1.png) Instantaneous static pressure (normalized by inlet total pressure) distributions at 90% span.

![Fig. 3](image2.png) Instantaneous entropy distributions at 90% span.

The widely-used 1D characteristic-based mixing-plane method proposed by Giles\(^{9}\) is also implemented for comparative purpose.

3.1. 1.5 stage transonic axial compressor

A 1.5 stage compressor extracted from a multistage high-pressure compressor is considered first. In order to perform fast unsteady calculation as reference, the blade count number ratio is scaled to be 1:1:2. In the unsteady simulation, the rotor shock travels upstream across the first mixing-plane (MP1) due to the high loading of the rotor and relatively small gap from the upstream inlet guide vane (IGV) (Fig. 2). Meanwhile, the thick rotor wakes propagate through the second mixing-plane (MP2) and interact with the downstream stator (Fig. 3). Thus, this test case is very suitable for demonstrating the effectiveness of the proposed mixing-plane model in dealing with the typical circumferential nonuniform flow field featured by strong shock and thick wakes.

First, the flux differences across the rotor-stator interfaces are quantified in Table 1. The mixing plane without buffer layer gives comparative amount of flux difference as Giles’ method, while the proposed mixing-plane method with buffer layer gives the lowest flux difference among the three methods. The maximum flux difference in terms of percentage of the upstream value is from the tangential momentum flux and still less than 0.04%. In general, it can be concluded that the mixing-plane method with buffer layer satisfies flux conservation more strictly.

The relative Mach number contours at 90% span are shown in Fig. 4. On the whole, Giles’ method and mixing plane without buffer layer give almost the same flow patterns. There are obvious non-physical reflections when the rotor shock propagates across the MP1 thus distorts the rotor flow field, making it obviously different from the time-averaged unsteady solution. The new mixing-plane method with buffer layer can greatly reduce the artificial reflections at the interfaces and the obtained flow patterns are very close to the time-averaged results. Due to much smaller circumferential nonuniformity of the flow field in the stator inlet region, the reflections at MP2 are very weak for all the three methods and the flow fields in the stator row are quite similar.

The rotor inlet relative flow angle distributions at 90% span are compared in Fig. 5. As the gap between the blade leading edge and the interface is small, enforcing a uniform inlet flow angle distribution at the rotor inlet would lead to physically unrealistic blade leading edge loading. Therefore, all of the three mixing-plane methods used here allow the inlet flow angle to adjust circumferentially at the inlet, which can be

<table>
<thead>
<tr>
<th>Flux term</th>
<th>No buffer layer</th>
<th>With buffer layer</th>
<th>Giles’ method in Ref.(^{9})</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>MP1</td>
<td>MP2</td>
<td>MP1</td>
</tr>
<tr>
<td>Mass flux</td>
<td>0.048</td>
<td>2.5 × 10(^{-2})</td>
<td>3.87 × 10(^{-5})</td>
</tr>
<tr>
<td>Axial momentum flux</td>
<td>0.015</td>
<td>1.5 × 10(^{-3})</td>
<td>4.44 × 10(^{-5})</td>
</tr>
<tr>
<td>Tangential momentum flux</td>
<td>0.029</td>
<td>6.0 × 10(^{-5})</td>
<td>2.20 × 10(^{-3})</td>
</tr>
<tr>
<td>Radial momentum flux</td>
<td>0.046</td>
<td>2.5 × 10(^{-2})</td>
<td>1.00 × 10(^{-3})</td>
</tr>
<tr>
<td>Total enthalpy flux</td>
<td>0.034</td>
<td>1.6 × 10(^{-2})</td>
<td>4.00 × 10(^{-5})</td>
</tr>
</tbody>
</table>

Please cite this article in press as: Du P, Ning F Validation of a novel mixing-plane method for multistage turbomachinery steady flow analysis, Chin J Aeronaut (2016), http://dx.doi.org/10.1016/j.cja.2016.10.005
realized by setting the boundary conditions at interface to be like a far field boundary. Without the buffer layer, the mixing-plane method gives almost identical flow angle distributions as Giles’ method. The circumferential variation trends are compatible with the time-averaged results, but the magnitude can have a difference of about 2°. Satisfyingly, the difference between the mixing-plane method with buffer layer and the time-averaged results is minor. Fig. 6 shows the stator inlet flow angle distributions at 90% span. Although the flow patterns in the stator row are very similar (Fig. 4), much better predicted result can be achieved by the new mixing-plane method with buffer layer.

The streamwise evolutions of the circumferentially mass-averaged entropy are shown in Fig. 7. Due to the interaction of the rotor shock and the IGV blade, additional loss was generated at the aft part of the IGV passage. All the steady mixing-plane methods cannot predict this unsteady effect as expected. At the IGV and rotor interface (MP1), Giles’ method predicts the smallest entropy increment. The entropy increment is larger for the new mixing-plane method and it has about the same magnitude as the entropy generated by the unsteady interaction. Thus the entropy evolution in the rotor
passage is closer to the unsteady results. At the rotor-stator interface (MP2), all the mixing-plane methods give about the same amount of entropy increment. Theoretically, when the nonuniformity of the flow field at the mixing-plane interface is not too strong to violate the basic assumption of the 1D NRBC, there is no need to use the buffer layer. Thus, the magnitude of entropy increment across MP2 is basically the same for the three mixing-plane methods considered.

In turbomachinery design phase, the rotor and stator are usually treated as two separate parts with their outlet and inlet conditions coupled. Uniform total pressure and total temperatures are often set at the inlet of each blade row. Thus, a sound mixing-plane method should yield a solution which is independent of the relative position of the rotor and stator, resulting in uniform pitchwise relative total pressure and total temperature distribution at the inlet of the downstream row. Hanimann et al. pointed out that this is one of the most important criteria for a mixing-plane method and is lacking in most commercial codes. Figs. 8 and 9 show the contours of the normalized relative total pressure and total temperature distributions at 90% span, respectively. The nonuniformity at the rotor inlet is clearly visible in the cases using Giles’ method and no buffer layer mixing-plane method. Compared to these results, the mixing plane with buffer layer provides nearly uniform fields at both the rotor and stator inlets. Due to weaker nonuniformity of flow field in the stator inlet region, all the methods give more uniform distributions than those in the rotor inlet region. Generally, the mixing plane with buffer layer performs best among the three methods considered in this paper.

### 3.2. Single-stage NASA centrifugal compressor

The NASA high pressure ratio centrifugal compressor was designed in 1975. Detailed information of stage geometry and test results can be found in Refs.15,16. The main geometry and aerodynamic design parameters of this compressor are listed in Table 2.

Fig. 10 shows the solid view of the centrifugal compressor and the single-passage computational mesh. It contains about $5 \times 10^5$ and $3 \times 10^5$ nodes in an impeller and wedge diffuser blade passage, respectively. The calculated total pressure ratio and adiabatic efficiency characteristics are compared in Fig. 11. The predicted choke mass flow is about 2.4% higher than the experimental data. The calculated total pressure ratio and efficiency have similar variation trends as compared to the experimental data, but the efficiency differences are also apparent. Most importantly, the results using the mixing-plane method with buffer layer are better and almost equal to the reference time-averaged unsteady results.

The flux differences at the mixing-plane interface at the near design working point predicted by the three mixing-plane methods are listed in Table 3. With buffer layer added, the fluxes except the axial momentum flux (the flux tangential to the interface) can be reduced significantly.

The three mixing-plane methods give almost the same results at large mass flow rates. However, when the working point moves toward the stall point, there will exist reverse flow across the mixing-place interface (Fig. 12). The calculation with Giles’ method will diverge when strong reverse flow emerges, resulting in a much larger stall mass flow rate.

![Fig. 8](image)

**Fig. 8** Relative total pressure (normalized by inlet total pressure) contours at 90% span.
(Fig. 11). On the contrary, the mixing-plane method with buffer layer is very robust. It can handle reverse flow naturally and do not induce any numerical instability problem. All the residuals can still continuously drop by four orders of magnitude (Fig. 13) when reverse flow travels across the mixing-plane interface.

The mixing, occurring at the interface, results in an abrupt entropy increment. Fig. 14 shows the circumferentially mass-averaged entropy evolution along the streamwise direction at near stall working point. First, there is a feature that the entropy evolution along the impeller passage (before the mixing-plane interface) is almost not influenced by the different mixing-plane treatments. The upstream impeller can be regarded as working in an isolation environment. However, the impeller-diffuser interactions in this highly loaded centrifugal compressor would cause additional loss in the impeller passage\(^{17,18}\) and this unsteady effect cannot be captured by steady simulations. The mixing plane without buffer layer induces higher entropy increment across the interface, which leads to the total entropy difference at the diffuser exit. By coincidence,

![Fig. 9](image)

Table 2  Compressor’s main design parameters.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design mass flow rate (kg/s)</td>
<td>4.54</td>
</tr>
<tr>
<td>Total pressure ratio</td>
<td>4.0</td>
</tr>
<tr>
<td>Adiabatic efficiency</td>
<td>0.833</td>
</tr>
<tr>
<td>Impeller rotating speed (r/min)</td>
<td>21,789</td>
</tr>
<tr>
<td>Number of impeller blades</td>
<td>15 full + 15 splitter</td>
</tr>
<tr>
<td>Impeller tip clearance (mm)</td>
<td>Inlet: 0.15; Exit: 0.20</td>
</tr>
<tr>
<td>Number of diffuser blades</td>
<td>24</td>
</tr>
</tbody>
</table>

(Fig. 10) Solid view of centrifugal compressor and computational mesh.
the entropy increment at the interface caused by the mixing plane with buffer layer is almost equal to the additional entropy increment in the impeller passage caused by the impeller-diffuser interactions, and thus the overall entropy at the diffuser exit matches the unsteady time-averaged results.

The total pressure contours at 50% span at near stall point are compared in Fig. 15. The total pressure distributions are not uniform at the diffuser inlet in the cases of Giles’ method and the mixing-plane method without buffer layer. However, by adding the buffer layer, the non-physical reflections at the interface are effectively reduced; as a result, the inlet total pressure distributions are nearly uniform. The same behavior can be found for the total temperature distributions in the diffuser inlet region which is not shown here.

### 3.3. LISA turbine

The last test case is the research turbine “LISA”, which was designed by Behr et al. at the Turbomachinery Laboratory in Swiss Federal Institute of Technology. This turbine is featured by low aspect ratio, high loading, and representing typical profile as well as flow condition in modern high pressure turbine. Some main parameters of this turbine are given in Table 4. Detailed blade geometry and experimental data can be found in Refs. 19, 20.

![](image1.png)  
**Fig. 11** Comparison of centrifugal compressor performance.

<table>
<thead>
<tr>
<th>Table 3 Flux difference at interface (percent of upstream value) at near design working point.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flux term</td>
</tr>
<tr>
<td>-----------------------------------------------------</td>
</tr>
<tr>
<td>Mass flux</td>
</tr>
<tr>
<td>Axial momentum flux</td>
</tr>
<tr>
<td>Tangential momentum flux</td>
</tr>
<tr>
<td>Radial momentum flux</td>
</tr>
<tr>
<td>Total enthalpy flux</td>
</tr>
</tbody>
</table>

![](image2.png)  
**Fig. 12** Circumferentially averaged streamlines.

The entropy increment at the interface caused by the mixing plane with buffer layer is almost equal to the additional entropy increment in the impeller passage caused by the impeller-diffuser interactions, and thus the overall entropy at the diffuser exit matches the unsteady time-averaged results.

The total pressure contours at 50% span at near stall point are compared in Fig. 15. The total pressure distributions are not uniform at the diffuser inlet in the cases of Giles’ method and the mixing-plane method without buffer layer. However, by adding the buffer layer, the non-physical reflections at the interface are effectively reduced; as a result, the inlet total pressure distributions are nearly uniform. The same behavior can be found for the total temperature distributions in the diffuser inlet region which is not shown here.

### 3.3. LISA turbine

The last test case is the research turbine “LISA”, which was designed by Behr et al. at the Turbomachinery Laboratory in Swiss Federal Institute of Technology. This turbine is featured by low aspect ratio, high loading, and representing typical profile as well as flow condition in modern high pressure turbine. Some main parameters of this turbine are given in Table 4. Detailed blade geometry and experimental data can be found in Refs. 19, 20.

![](image3.png)  
**Fig. 13** Residual at near stall working point.

Please cite this article in press as: Du P, Ning F. Validation of a novel mixing-plane method for multistage turbomachinery steady flow analysis, *Chin J Aeronaut* (2016), http://dx.doi.org/10.1016/j.cja.2016.10.005
The blade fillets are considered and the computational mesh is shown in Fig. 16. The grid nodes are about $0.5 \times 10^6$ in each blade passage. First, the calculated flux differences at the mixing-plane interfaces are compared in Table 5. With the aid of buffer layer, all the flux differences except the tangential momentum flux are negligible.

The mass flow rates calculated by the three steady methods are about 11.8 kg/s, slightly lower than the experimental data. The efficiency calculated by the mixing-plane methods with and without buffer layer is 0.875 and 0.873, respectively. The efficiency calculated by Giles’ method is about the same as the results with buffer layer. Fig. 17 shows the streamwise evolution of the circumferentially mass-averaged entropy. The mixing plane without buffer layer gives the largest entropy increment across the interface. The mixing-plane method with buffer layer predicts slightly larger entropy increments at the interfaces but with smaller increments in the blade passages, and thus the overall loss is almost the same as Giles’ method. The slightly larger entropy increment at the interfaces with buffer layer may be due to the numerical dissipation induced by solving the buffer layer governing equations.

Fig. 18 shows the spanwise distribution of the relative total pressure loss. The calculated total pressure loss of the first stator is a little larger than the experimental data at most of span. The calculated total pressure loss of the rotor is higher than the experimental data near the shroud, and the flow angle extrema are also over-predicted (Fig. 19). This higher discrepancy may be due to the over-predicted loss caused by passage vortices and the tip leakage vortex near the rotor shroud. The calculated total pressure losses of the second stator match well with the experimental data. Fig. 19 shows the flow angles at each blade row exit. Generally, the flow angles are well predicted by all simulations. Differences appearing in the prediction near the rotor should have also been identified with a higher total pressure loss in that region. Fig. 20 shows the spanwise distribution of the Mach number at the exit of each blade row. Slightly better results are achieved by the new mixing-plane method. Generally speaking, all the three mixing-plane methods perform almost equally in predicting the total performance characteristics and spanwise distributions of the flow quantities in this test case.

![Fig. 15](image-url) Total pressure (normalized by inlet total pressure) contours at 50% span.

### Table 4 Main parameters of “LISA” 1.5-stage axial turbine research facility at design operating point.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Blade number</td>
<td>36:54:36</td>
</tr>
<tr>
<td>Rotor speed (r/min)</td>
<td>2700</td>
</tr>
<tr>
<td>Rotor tip clearance (mm)</td>
<td>0.68</td>
</tr>
<tr>
<td>Inlet total pressure (Pa)</td>
<td>140,000</td>
</tr>
<tr>
<td>Inlet total temperature (K)</td>
<td>328.15</td>
</tr>
<tr>
<td>1.5-stage, total to static pressure ratio</td>
<td>1.6</td>
</tr>
<tr>
<td>Mass flow (kg/s)</td>
<td>12.13</td>
</tr>
<tr>
<td>Total pressure ratio of the first stage</td>
<td>1.35</td>
</tr>
<tr>
<td>Estimated total-to-total efficiency of the first stage</td>
<td>0.878</td>
</tr>
</tbody>
</table>
The relative Mach number contours at 50% span are shown in Fig. 21. The non-physical reflections near the mixing-plane interfaces are all small because the circumferential disturbance is not intensive so that the 1D NRBC can handle it well. Even so, with the aid of buffer layer, the drawn contours near the inlet region of the rotor and the second stator pass through the mixing-plane interfaces more smoothly, which is closer to the reference time-averaged unsteady results. The relative total pressures at the inlets of the rotor and the second stator are not uniform for Giles’ method and the mixing-plane method without buffer layer (Fig. 22). With buffer layer added, nearly uniform relative total pressures are achieved, which is more compatible with the concept of mixing-plane method.
Fig. 21  Relative Mach number contours at 50% span.

Fig. 22  Relative total pressure (normalized by inlet total pressure) contours at 50% span.
4. Conclusions

For steady simulation of flow in multistage turbomachines, a novel mixing-plane method is presented which is based on the concept that pitchwise non-uniformities are mixed out in the hypothetical extended mixing region. The mixed-out state is obtained by solving a set of mixing-plane governing equations which enforces the conservation of the flux at the interface. Then, a constant-radius buffer layer was constructed to alleviate the non-physical reflection from the mixing-plane interface, and one-dimensional characteristic boundary condition is then applied to further eliminate linear disturbances.

Three typical turbomachinery test cases have been carried out for validation purpose. The solutions of this new mixing-plane method satisfy the flux conservation perfectly. It can handle reverse flow across the mixing plane naturally, thus making the method very robust. With the aid of the buffer layer, the non-physical reflection at the mixing-plane interface is almost eliminated. It can give pitchwise uniform relative total pressure and total temperature distributions downstream of the interface. This uniformity respects that it is compatible with the physical concept of mixing-plane method. Generally, this new mixing-plane method is suitable to simulate steady flow in highly loaded multistage turbomachines.

References


Du Pengcheng received the Ph.D. degree in fluid machinery from Beihang University in 2015. He now is a post-doctor at School of Aeronautic Science and Engineering in the same university. His research interests are computational fluid dynamics, fluid-structure interactions and turbomachinery design.

Ning Fangfei received the Ph.D. degree in fluid machinery from Beihang University in 2002. He now is an associate professor at School of Energy and Power Engineering in the same university. His research interests are computational fluid dynamics, high performance parallel computing, turbomachinery aerodynamics and turbulence modeling.