Numerical Simulation on the Flow Field in a Turbine Stage with Upstream Flow Injection from the Outer Casing: Effects of the Injection Angle*

Ken-ichi FUNAZAKI**, Carlos Felipe Ferreira FAVARETTO**, Masaya KAMATA** and Tadashi TANUMA***

Flow injection is being used in steam turbines for some combined or geothermal plants in order to increase its total thermal efficiency. It can be easily imagined, however, that such a flow injection is accompanied by additional aerodynamic loss due to the flow mixing or the change in local flow angle. Preliminary three-dimensional steady numerical simulations are conducted for the flow field inside the chamber from which the secondary flow is injected. The results showed a small variation of the flow velocity at the injection slot and provided useful information concerning the magnitude of the pitchwise variation of the injection angle. Further numerical simulations in a turbine stage were then performed by prescribing different injection angle distribution at the injection slot. The turbine stage computations revealed the existence of a high loss region generated from the interaction of the injected flow and the passage vortex near the tip of the nozzle vanes. It was found that injection angles orientated to the suction side of the stator provided lower loss values, while cases where flow was injected to the pressure side minimized the flow angle deviation.

Key Words: Flow Injection, Turbine Stage, Inlet Distortion, Tip Clearance, Shrouded Rotor

1. Introduction

Secondary flow injection into a flow field can be found in many mechanical engineering applications, ranging from micro air injection in axial compressors[4] to steam injection in high efficiency combined cycle steam turbines. In the latter, injection flows (from 5 to 15% of main mass flow rate) from steam generators are introduced between turbine stages, resulting in a substantial improvement of the thermodynamic efficiency. As a counterpart of the gain due to the injection of steam at a higher temperature, the loss due to the mixing of the secondary and the main streams may incur additional pressure loss.

The upstream steam injected from the outer casing is usually at a different temperature and velocity than the main flow. Since the injected steam and the main flow are not completely mixed in the vicinity of the stator leading edge, pitchwise and spanwise variations of the mass flow and the unsteadiness of the inlet flow angle occur (Fig.1). Such variation will change the turbine stage operating conditions, affecting the blading flow pattern and efficiency.

The lack of information on the flow field generated by the flow distortion and the increasing demand of the steam turbine manufacturers in designing high performance machines have motivated a significant number of authors to investigate the inlet and outlet flow distortion phenomena.

In the work of Hirai et al.[5], numerical simulation was used to investigate the effect of circumferential positions of inlet hot streaks to a single stage turbine. The pressure loss due to unsteady flow for a transonic rotor with inlet total pressure distortion was also analyzed.

Moser[6] investigated distorted flow conditions caused by asymmetric exhaust and ribs in the radial
and circumferential direction using pneumatic probes. Zeschky and Gallus determined the effects of the outlet distortion provided from the stator exit on the turbine rotor by using three-dimensional hot-wire and pneumatic probes. The formation of the passage vortices in the rotor were found to be strongly influenced by the non-uniform stator outlet flow which caused the accumulation of low-energy fluid in the rotor wake close to midspan.

Biesinger & Gregory-Smith (5) analyzed the effect of upstream tangential blowing in a turbine cascade of rotor blades. According to their experiments, blowing at low velocity served to increase the amount of low energy fluid, resulting in a rise in secondary flow and loss. As the blowing rate was increased, the positive streamwise vorticity started to counteract with the secondary vortex, and a reduction in secondary kinetic energy and loss was achieved. The injection angle was also analyzed by the authors, the low angle being more effective than the high angle, keeping the blowing jet closer to the end wall at inlet to the cascade.

Funazaki et al. (6) analyzed the effect of flow injection from the outer casing on the flow field around nozzle vanes. Several test cases were analyzed by changing the blade lean angle and the injection slot configuration. It was found that the blade which was leaned to the positive (pressure) side provided a reduction in the overall energy loss but increased the outlet flow angle deviation.

Funazaki et al. (7) investigated the effects of flow injection in a turbine stage by performing numerical simulation and analyzing the experimental data measured in the single stage turbine rig. It was found that increasing the flow injection reduced the total pressure ratio and increased the outlet flow angle deviation, which is the difference between the outlet flow angle of the no injection and the injection cases. The authors concluded that such behavior was due to the fact that the flow injection angle was considered as normal to the casing. In such situation, the penetration of the injection flow is higher and so is the mixing loss.

In the actual turbine stage configuration, however, the flow injection angle is not necessarily normal to the wall. The injection flow is driven to a chamber from which the fluid exits through a thin slot (Fig. 2). Since the chamber has a circumferential curvature and the injection flow enters the domain at two inlet pipes, the secondary flow is expected to have a variation in its direction and magnitude. The goal of the present paper is to describe the effects of the flow injection angle on the turbine stage in terms of the total pressure ratio and outlet flow angle deviation.

**Nomenclature**

- **AMG**: algebraic multigrid
- **CV**: corner vortex
- **HV**: high velocity
- **LE**: leading edge
- **LF**: low-energy fluid
- **m**: mass flow
- **MLP**: modified linear profile
- **P**: pressure
- **PAC**: physical advection correction
- **PS**: pressure side
- **PV**: passage vortex
- **RP**: rotor potential
- **SS**: suction side
- **SV**: shroud vortex
- **TE**: trailing edge
- **Tu**: turbulence intensity
- **a**: flow angle
- **β**: flow injection angle
- **ν**: kinematic viscosity
- **νT**: eddy viscosity
3. Numerical Simulation

3.1 Grid generation

A structured multi-block grid system was generated for both simulation cases by using the CFX-TurboGrid 1.6 and the CFX-Build 4.4 software (AEA Technology Ltd.). The computational domain was generated with H-type, O-type and C-type grid blocks. Concerning the chamber-stator case, 26,544 grid points were used for each of the stator passages, 258,504 points for the chamber and 33,631 for the domain extension at the outlet (Fig. 4). For the stator-rotor case, the grid system around the stator consisted of 548,050 grid points, around the rotor 465,710 points and in the cavity 71,875 points (Fig. 5). In the tip clearance region of the rotor (approximately 0.02% of the span) 5 grid points in the spanwise direction were used.

![Fig. 4 Chamber-stator grid system (Wall boundaries represented only)](image)

![Fig. 5 Stator-rotor grid system (Additional blade passages are represented)](image)
Considering the fact that a wall function was employed, the mesh seeding was performed so that the dimensionless wall distance $y^+$ of the first near wall node was approximately 30. Grid independency analysis was performed by changing the grid density in the streamwise, pitchwise and spanwise directions.

### 3.2 Computational code

The three-dimensional, steady-state, Reynolds-averaged, compressible Navier-Stokes equations were solved with the CFX-TASCflow 2.11.1 computational code (AEA Technology Ltd.). Several numerical analyses of the flow field inside turbine stage using this code were reported by von Hoyningen-Huene & Hermeler(9), Peters et al.(10) and Funazaki et al.(9).

Concerning the domain discretization, the code uses a finite volume method based on the finite element method in order to enable a more accurate modeling of the geometry. The diffusive terms are calculated by the conventional finite element method using shape functions to calculate the derivatives. In the same fashion, the pressure gradient terms in the momentum equations are also determined.

A second-order differencing scheme, the Modified Linear Profile (MLP) with the Physical Advection Correction (PAC) scheme (CFX-TASCflow Theory Documentation(11)), was employed. These schemes are based on the conservative finite volume approach with special care taken to minimize the errors normally associated with upstream differencing schemes. The underlying principle of PAC schemes is that the resulting algebraic equations should correctly mimic the transport properties of the physical processes. For instance, considering an advection dominated transport equation, the correction for the $z$-derivative would be a function of the source term and the components of advection along the $y$ and $z$ coordinate directions(11). Therefore, this discretization method produced the most accurate solution among the methods available in the code.

The solution algorithm uses the AMG (Algebraic Multigrid(12)) method, which is based on the ACM (Additive Correction Method, Hutchinson et al.(12)) correction strategy. The algorithm is fully coupled, i.e., the momentum and continuity equations are solved simultaneously. The unstructured data structure allows connectivities in periodicity and grid interface regions to be solved in a fully implicit fashion. Detailed information on the theoretical basis of the software can be found in the CFX-TASCflow Theory Documentation(11).

### 3.3 Boundary conditions

The boundary conditions employed were based on the experimental data. For the inlet region, total pressure, total temperature, inlet turbulence intensity ($Tu$) and the ratio between eddy and molecular viscosity ($\nu_r/\nu$) were prescribed. The values for the inlet turbulence intensity and inlet eddy length scale were investigated by performing several simulations combining difference values. In the present study it was found that $Tu=1\%$ and $\nu_r/\nu=10$ presented the best agreement with the experimental data. For the outlet region, static pressure was prescribed. Since this quantity had not been measured, back-calculations were performed by changing the outlet static pressure values until the resulting outlet total pressure best matched the experimental data.

Concerning the flow injection in the stator-rotor case (Fig. 3), the velocity was prescribed according to the injection angle $\beta$ and its magnitude was calculated as 10% of the main stream flow rate. In the chamber-stator case (Fig. 2), the velocity was calculated in the same fashion and applied normal to the injection flow inlet. In both cases, the same value for the total temperature at the inlet was prescribed in the flow injection region.

The chamber, blade, hub, shroud and disk cavity wall regions were assumed as adiabatic and the non-slip condition was applied. A logarithm wall function was used for the near-wall grid points. For the pitchwise boundaries, periodic boundary condition was applied. The Reynolds number based on the stator axial chord length and inlet velocity was 38 000 for all cases.

### 3.4 Definition of the frame of reference

The stator, the disk cavity and the domain extension downstream of the rotor were assumed to be in the stationary frame of reference whereas the grid blocks surrounding the rotor blade were in the rotating frame of reference. Thus, three sliding grid interfaces were created at the junctions between different frames of reference.

Concerning the software adopted in the calculations, two interface models are available. The "stage" model, usually referred to as mixing-plane approach, performs a pitchwise average of the flow quantities at the interface, assuming a complete mixing of the upstream velocity profile. The "frozen rotor" model does not perform any averaging and assumes the relative position of the two components are fixed for the entire simulation (as shown in Fig. 3), i.e., a quasi-state approximation. The flow field around the stator is calculated and the results at the sliding interface used as inflow conditions for the rotor domain. The flow around the rotor is then solved in the relative frame of reference. Both models were tested and the pitchwisely averaged results did not show significant difference in relation to the type of interface model.
The major drawback found when using the "stage" model is that the important flow patterns were lost due to the averaging whereas for the "frozen rotor" model the wake/core profile was preserved across the interface. For such reason, the "frozen" rotor model was found to be more useful in identifying the flow structures and was chosen as the standard one. For more information on the sliding grid interface models refer to CFX-TASCFlow Theory Documentation.

3.5 Turbulence model

It is widely known that the two-equation models, such as the $k-\varepsilon$, are not accurate for complex fluid flow simulation. For instance, the anisotropy existing in the highly turbulent region near the trailing edge or in the region where the injection flow is mixed with the main flow is not taken into account in that kind of model. In addition to such strong limitation, the Galilean invariance existing in a problem with multiple frames of reference is also neglected, as mentioned by Durbin and Peterson Ref. Considering such limitations, the Reynolds stress second-order closure turbulence model using the SSG (Speziale, Sarkar and Gatsi) quadratic model with scalable log-law wall function was employed.

The major drawback of using the Reynolds stress turbulence model, however, is the additional mathematical formulation involved. Not only the computational time will be substantially increased, if compared to the two-equation models, but also the convergence will be more unstable. As a remedy, the first 20 iterations were solved with the $k-\varepsilon$ model and the remaining with the second-order closure model.

4. Results and Discussion

4.1 Flow field inside the chamber and around the stator

Figure 6 presents the circumferential velocity vector distribution in the center of the injection slot. The secondary flow is injected through two pipes, as shown in Fig. 2, at approximately 120° and 240°. It can be observed that the injection angle is changed with the circumferential position, oriented to the suction side of the stator blade in some locations and to the pressure side in other locations. Since the axial velocity component in the vicinity of the injection slot is negligible comparing to the mean axial velocity of the main flow at the inlet, the injection flow angle can be determined from the projection of the velocity vector to the respective plane normal to the streamwise direction.

In Fig. 7, the circumferential distribution of the injection angle ($\beta$) is represented as a function of the circumferential position for three locations: near the upstream edge of the slot (thick grey line, position 1), in the center (thin solid line, position 2) and near the downstream edge (thick solid line, position 3). For the $\beta>90^\circ$ cases, the flow injection angle is orientated to the suction side (following the direction of the passage vortex) and for $\beta<90^\circ$ cases, to the pressure side. It can be noted that the range of the angle variation is $\pm 30^\circ$, relative to the injection angle normal to the slot ($\beta=90^\circ$). The range in which the

---

*Fig. 6 Velocity vector distribution in the center of the injection slot*

*Fig. 7 Flow injection angle distribution at three locations*

*Fig. 8 Normalized injection velocity distribution at three locations*
experiments were performed \((83^{\circ} \leq \beta \leq 97^{\circ})\) is also indicated in Fig. 7 by a transparent box.

Figure 8 shows the injection velocity distribution normalized by the mean inlet axial velocity of the main flow. The results near the upstream edge of the injection slot (position 1) presented the highest magnitude variation, approximately 9%.

From the results shown it can be learned that the injection flow angle variation is substantial compared with the change in the magnitude of the velocity. Therefore, the stator-rotor simulation that follows will only take into account the injection angle variation, keeping the magnitude of the velocity vector constant for all cases.

4.2 Flow field in the turbine stage (no injection)

Initially, the no injection case was solved and validated with the experimental data described by Kamata et al.\(^{(6)}\). Their measurements were taken at two traversing planes ranging from 3.8% to 96% span and approximately three rotor blade pitches. Figure 9 presents the axial velocity distribution downstream of the rotor blades. The results were normalized by the mean inlet axial velocity and depicted for three pitch lengths. Similar flow patterns downstream of the rotor can be observed on both experiments (Fig. 9 (a)) and numerical simulation (Fig. 9(b)) results. The passage vortex generated by the cross-pressure gradient near the root can be clearly identified at approximately 25% span \((PV2)\). The correspondent passage vortex near the blade tip is also recognizable but with a relative lower intensity \((PV2)\). The vortex generated from the interaction of the flow field with the blade tip, the shroud and the clearance can be observed by the contours denoted as \(SV\). The corner vortex generated on the rotor hub surface \((CV)\) can be entirely observed in the numerical results and partially recognizable by the experiments.

Once the flow patterns for the no injection case were recognized, numerical simulations for the 10% injection were conducted.

4.3 Flow field in the turbine stage (10% flow injection)

The flow injection in combined cycle steam turbines can generate a substantial gain in terms of the thermodynamic efficiency. As a counterpart, it is likely that additional profile loss due to the mixing of the two streams may occur, depending on the injection flow rate, the injection angle and the geometry of the downstream edge of the injection slot. However, the flow injection may also be regarded as a technique to reduce the profile loss, as described by Biesinger & Gregory-Smith\(^{(6)}\). According to the authors, the effect of the increased blowing is first to thicken the inlet boundary layer, giving greater secondary flow and more loss, and then as a re-energisation of the inlet boundary layer takes place together with increasing counter streamwise vorticity, the passage vortex is progressively weakened, with a corresponding reduction in loss.

Based on the flow field at the injection slot obtained by the chamber-stator simulation (Fig. 7), the turbine stage calculations were performed for \(\beta=60^{\circ}, 90^{\circ}\) and \(120^{\circ}\). The injection angle was prescribed as constant as well as the injection velocity. Following the findings of Biesinger and Gregory-Smith\(^{(6)}\) that smaller angles may be more beneficial for reducing the profile loss, additional test cases for \(\beta=45^{\circ}\) and \(135^{\circ}\) were also analyzed.

![Normalized axial velocity for the no injection case as seen from downstream of the rotor (secondary vorticity vectors represented on the right side)](image)

Fig. 9 Normalized axial velocity for the no injection case as seen from downstream of the rotor (secondary vorticity vectors represented on the right side)
Figure 10 presents the streakline plots for the no injection and the 10% flow injection ($\beta=90^\circ$) cases. The streaklines were emitted from the most upstream nodes in the vicinity of the end wall. For the sake of clarification, only the stator was depicted and the streaklines were terminated at the frame of reference interface. In Fig. 10(a) one can observe the trajectory of the passage vortex formed by the cross-pressure gradient. In Fig. 10(b), due to the flow injection, the same clear structure cannot be observed. The fluid particles departing from far upstream deviate their trajectories near the upstream of the injection slot. At the bottom part of Fig. 10(b), streaklines emitted from the injection slot are presented. It can be evidenced that approximately one half of the streaklines follows the same behavior as the streaklines emitted from the far upstream and the other half recirculates. From Fig. 10(b), it seems that some streaklines close to the suction side of the stator remained with their initial orientation, maintaining the outlet flow angle unchanged. At different pitch-wise locations, however, the streaklines are gradually deviated. The passage vortex is enlarged and displaced outward from the casing. The original locus where the passage vortex would exist if no flow were injected is filled with the additional vortical structures. These flow patterns are generated from the mixing of the main stream with the injected flow.

Figure 11 shows the normalized axial velocity distribution upstream of the rotor for the 10% injection cases. The contours indicate the increase in the penetration of the injection flow with increasing injection angle. A high velocity region (HV), indicated in Fig. 11 by arrows, can be spotted for all cases. As the injection angle is increased this region is grown in size and displaced outward from the end wall. Such variation of the axial velocity distribution can be attributed to the recirculation region downstream of the injection slot (bottom of Fig. 10(b)), which causes a blockage effect to the main flow. The streaklines emitted from the far upstream, when encountering such region, are compressed from hub to the periphery of the large vortices formed by the mixing of the main flow with the injected flow. Considering the fact that the cross sectional area is reduced in such region, the main stream is decelerated by means of an adverse pressure gradient.

The local increase in axial velocity shown in Fig. 11 incurs in a compensatory deceleration of the flow field. For $\beta=45^\circ$ (Fig. 11(b)) a small low-energy fluid region (LFI) is found between the periphery of the high velocity region and the casing. Below the high velocity region, however, there is a compensatory deceleration of the axial velocity. As $\beta$ is increased from $45^\circ$ to $90^\circ$, this region (LF2) is convected towards the end wall at a similar rate as the displacement of the high velocity region towards midspan (Fig. 11(c)). As the injection angle is increased to the suction side, from $90^\circ$ to $135^\circ$, the low-velocity region near the casing is substantially increased in the same proportion as the high-speed region (Figs. 11(e) and (f)).

![Figure 10](image1.png)  
**Fig. 10** Streaklines for the no injection and 10% injection ($\beta=90^\circ$) cases

![Figure 11](image2.png)  
**Fig. 11** Normalized axial velocity distribution upstream of the rotor for the 10% injection cases (seen from downstream)
In Fig. 12 one can observe the surface flow plots or oil flow plots, which are streamlines restricted to lie in the plane of the boundary surface. The footprint of the passage vortex close to the hub surface can be seen for the no injection and the 10% injection ($\beta = 45^\circ, 90^\circ$ and $135^\circ$) cases. The passage vortices close to the casing ($PV1$) and near the hub ($PV2$) for the no injection case are shown in Fig. 12(a). As the flow is injected to the pressure side for $\beta = 45^\circ$ the size of the passage vortex seems to be reduced. However, as the injection angle is increased the low-energy fluid lump seems to be gradually increased. Observing the plots one may believe that the case represented in Fig. 12(b) is more suitable for reducing the loss than the case in Fig. 12(d). However, these trends describe a local effect happening near the suction side of the stator only, and not the global effects of the injection angle on the flow field.

Figure 13 is a caveat to avoid premature conclusions from Fig. 12. The caption shows the streamwise vorticity distribution in the stator passage at approximately 50% of its axial chord. A magnified view of the plots near the casing for one passage is also represented in the center of the figures. It can be observed that the high vorticity region, which contains the passage vortex and additional vortical structures, is moved towards the pressure side for $\beta = 45^\circ$ and $60^\circ$ and towards the suction side for $\beta = 120^\circ$ and $135^\circ$. Such conclusion was rather expected and agrees well with the trends in Fig. 12. However, what should be learned from the vorticity plots is the actual size of the vortices in the spanwise and pitchwise directions. From Fig. 12 it seemed that the $\beta = 45^\circ$ case reduced the size of the passage vortex, which actually did not happen. Figures 13(d), 13(e) and 13(f) indeed show enlarged vortical structures close the suction side but with rather small length in the pitchwise direction. The opposite behavior was found in Figs. 13(b) and 13(c). In average, the size of the vortical structures were actually enlarged for $\beta = 45^\circ$ and $60^\circ$ and its vorticity significantly increased. The $\beta = 135^\circ$ case, which is equivalent to $\beta = 45^\circ$ but orientated to the suction side of the stator blade, presented the lowest vorticity and smallest vortical structures. Thus, the results shown in Fig. 13(e) are in agreement with the findings of Biesinger and Gregory-Smith[9].

The interaction or counteraction of the vortical structure caused by the flow injection with the passage

**Fig. 12** Restricted surface flow plots (oil flow) on the suction side of the stator

**Fig. 13** Streamwise vorticity distribution for the no injection and 10% injection cases (seen from upstream, 50% axial chord in the stator passage)
vortex is the main mechanism which will increase or decrease the loss. Following the trends shown in Fig. 13, the total pressure ratio contours in the stationary frame of reference (Fig. 14) also illustrate the described loss mechanism. For $\beta=45^\circ$, the high loss region (indicated by the arrow) is concentrated between the casing and 95% span. This region seems small in size but the average of the total pressure ratio is the lowest among all cases. As the injection angle is increased to 60°, the size of the low-energy fluid region is substantially increased (90% span) with a small increase in the total pressure ratio. In both cases ($\beta=45^\circ$ and $\beta=60^\circ$) the flow is injected to the pressure side of the stator blade, opposing to the direction of the passage vortex. As the flow is injected it encounters the concave wall curvature on the pressure side of the stator blade, which causes a relative diffusion, raising the static pressure. For $\beta=90^\circ$, the flow is injected normal to the end wall with no initial favorable direction to follow. The high loss region is completely detached from the endwall and its size reduced. As the flow is injected to the direction of the suction side of the stator ($\beta=120^\circ$ and $\beta=135^\circ$) the high loss region continues to reduce its size and approaches the midspan. In this case, the flow is buffeted on the suction side of the blade, locally accelerating the flow and thus reducing the static pressure.

It can be observed that the loss core for $\beta=120^\circ$ (Fig. 14 (e)) is located at the same spanwise position (63%) as found in the experiments (Fig. 14 (a)). The measurements of Kamata et al. were taken within the circumferential range of $83^\circ<\beta<97^\circ$. According to the results from the numerical simulation of the chamber-stator model (Fig. 7), the respective flow injection angle for the circumferential position of 97° should be around $\beta=117^\circ$. This means that the numerical simulation for $\beta=120^\circ$ is within the range of the measurements and a partial similarity between this particular case and the experimental data was rather expected.

Figure 15 presents the pitchwisely averaged quantities upstream of the rotor for the 10% injection cases. The total span length shown in the graph was assumed as extending from root to rotor casing. The experimental data is presented with a thick curve from root to 90% span. Measurements were not performed in the remaining 10% span due to uncertainty introduced by the highly unsteady flow. The high loss region at 63% span previously shown in Fig. 14 for $\beta=120^\circ$ and for the experiments can also be observed in this caption. Care must be taken, however, when comparing the experimental data with the numerical results. As mentioned before, the measurements were performed at a certain measuring range, covering approximately 14° of pitch length. In this region, the flow angle exiting the injection slot changes substantially with the circumferential direction. This statement was verified by the chamber-stator calculations and shown in Fig. 7. The goal of the stage computations, however, was finding the relationship among injection angle, flow angle variation and loss. Thus, the injection angle for the simula-
tions was prescribed as constant along the circumferential direction. In reality, according to Fig. 7, the curves for \( \beta = 90^\circ \), \( \beta = 120^\circ \) and \( \beta = 135^\circ \) may match the experimental data curve at some spanwise locations. The experimental data contains the average effect of combining all three cases into one. Indeed, the tendency shown for the normal injection and the suction side orientated injection is also observed for the experimental data curve. In the latter, two minimum locations are found (65% and 80% span). In the numerical analysis, cases \( \beta = 120^\circ \) and \( \beta = 135^\circ \) have an injection point almost at the same location (63% and 60% span) while for case \( \beta = 90^\circ \) the minimum is located at 72% span.

In order to obtain qualitative information on the effects of the flow injection in terms of total pressure ratio, the values for the analyzed cases were averaged on the surface located in the same the position of the traversing planes (Table 1). Following a similar mixing-plane approach described by Denton[10], the total pressure was weighted-averaged with the values at the injection slot by the respective mass flow (Eq. (1)).

\[
\bar{P}_{\text{t}} = \frac{\bar{P}_{\text{t}} \cdot \bar{m}_t + \bar{P}_{\text{rs}} \cdot \bar{m}_s}{\bar{m}_t + \bar{m}_s}
\]

where: \( \bar{P}_{\text{t}} \) the weighted averaged total pressure, \( \bar{P}_{\text{t}} \)

Table 1 Totaally averaged total pressure ratio upstream and downstream of the rotor for the 10% injection cases

<table>
<thead>
<tr>
<th></th>
<th>CFD (( \beta = 45^\circ ))</th>
<th>0.9947</th>
</tr>
</thead>
<tbody>
<tr>
<td>Upstream</td>
<td>CFD (( \beta = 60^\circ ))</td>
<td>0.9948</td>
</tr>
<tr>
<td></td>
<td>CFD (( \beta = 90^\circ ))</td>
<td>0.9957</td>
</tr>
<tr>
<td></td>
<td>CFD (( \beta = 120^\circ ))</td>
<td>0.9957</td>
</tr>
<tr>
<td></td>
<td>CFD (( \beta = 135^\circ ))</td>
<td>0.9958</td>
</tr>
<tr>
<td>Downstream</td>
<td>CFD (( \beta = 45^\circ ))</td>
<td>0.9614</td>
</tr>
<tr>
<td></td>
<td>CFD (( \beta = 60^\circ ))</td>
<td>0.9613</td>
</tr>
<tr>
<td></td>
<td>CFD (( \beta = 90^\circ ))</td>
<td>0.9616</td>
</tr>
<tr>
<td></td>
<td>CFD (( \beta = 120^\circ ))</td>
<td>0.9617</td>
</tr>
<tr>
<td></td>
<td>CFD (( \beta = 135^\circ ))</td>
<td>0.9618</td>
</tr>
</tbody>
</table>

Extensive numerical calculations were performed in order to investigate the effects of the flow injection angle on the stage performance in terms of the outlet flow angle deviation and the total pressure ratio. The numerical results were validated with the experiments upstream and downstream of the rotor, presenting a satisfactory level of agreement.

The chamber–stator simulation case provided the authors important information concerning the pitchwise variation of the injection angle and the injection velocity at the slot exit. It was found that the maximum range of the injection angle variation was \( \pm 30^\circ \) whereas for the injection velocity it was no
greater than 9%.

The simulation of the flow field in the turbine stage revealed the following trends:

- The loss due to the flow injection can be minimized if small values for the injection angle are considered and the injection flow orientated to the suction side of the stator ($\beta > 90^\circ$).
- For cases where flow is injected to the pressure side ($\beta < 90^\circ$) the flow field counteracts with the passage vortex, reducing its size in the spanwise direction but increasing it in the pitchwise direction.
- For the cases where the flow injection was orientated to the suction side ($\beta > 90^\circ$), the lump of low-energy fluid formed due to the mixing with the main flow enlarged the passage vortex in the spanwise direction but reduced in the pitchwise direction, with a substantial reduction in the streamwise vorticity of its structure.
- The outlet flow angle was reduced for the $\beta < 90^\circ$ case and increased for $\beta > 90^\circ$. The reason for such trend was found to be related to the fact that the region where the flow is mostly disturbed by the injection effects is smaller than for the pressure side orientated angle.

The present study enabled the authors to understand the important parameters acting on a turbine stage with upstream flow injection from the outer casing. The upper and lower limits of the variables of the problem were detected, leading the current research to the development of an optimization tool in the near future.

Acknowledgements

The second author acknowledges the financial support of the Ministry of Education, Culture, Sports, Science and Technology of Japan.

References

(5) Blesinger, T.E. and Gregory-Smith, D.G., Reduc-
