Proceedings of the 3rd IASME/WSEAS Int. Conf. on FLUID DYNAMICS & AERODYNAMICS, Corfu, Greece, August 20-22, 2005 (pp206-211)

Simulation of Flow in Gas Turbine Pre-Swirl Systems with Emphasis on Rotor-Stator Interface Treatment

A. C. BENIM
Department of Mechanical and Process Engineering
Duesseldorf University of Applied Sciences
Josef-Gockeln-Str. 9, D-40474 Duesseldorf
GERMANY
alicemal.benim@fh-duesseldorf.de

M. CAGAN
Department of Mechanical and Process Engineering
Duesseldorf University of Applied Sciences
Josef-Gockeln-Str. 9, D-40474 Duesseldorf
GERMANY
markos.cagan@fh-duesseldorf.de

B. BONHOFF
Gas Turbine Design Systems
Siemens Power Generation
Mellinghofer Straße 55, D-45473 Mülheim a. d. Ruhr
GERMANY
bernhard.bonhoff@siemens.com

D. BRILLERT
Gas Turbine and Combined-Cycle Plants
Siemens Westinghouse Power Corporation
4400 Alafaya, MC Q3 039, Orlando, FL 32826-2399
USA
dieter.brillert@siemens.com

Abstract: - Flow in gas turbine pre-swirl systems has been computationally investigated. Emphasis has been placed on rotor-stator interface treatment, where the quasi-steady approach has been assessed by comparisons with the fully transient analysis. In the first part of the study, carried out in two-dimensions, the validity range of the quasi-steady formulation has been investigated. In the second part, to be discussed in the oral presentation, the analysis is extended to three-dimensional problems.

Key-Words: - Pre-swirl systems; Gas turbine cooling; Rotor-stator interaction; CFD.

Nomenclature

A_{PSN}	pre-swirl nozzle area (m^2)
$C_{D,PSN}$	pre-swirl nozzle discharge coefficient (-)
C_t	tangential velocity (m/s)
$\dot{m}$	mass flow rate (kg/s)
p	pressure (Pa)
R	gas constant (J/kgK)
T	temperature (K)
U	rotational speed (m/s)
y^+	non-dimensional wall distance (-)
α	angle betw. pre-sw. nozzle and sw. plate (rad)
κ	isentropic exponent (-)
Π	pressure ratio ($p_{t,0} / p_{s,2}$) (-)

Subscripts

id	ideal
PSN	pre-swirl nozzle
ref	reference
rel	relative
s	static
t	total
0	domain inlet
1	pre-swirl outlet
2	domain outlet

1 Introduction

To extend life whilst minimizing the impact on engine performance, high pressure turbine blades are typically supplied with cooling air at the lowest possible temperature using a feed system known as "pre-swirl". Cooling air is expanded through nozzles located on stationary components and collected by rotating receiver holes in a cover-plate fixed to the turbine disc, or discharged directly to the blade roots. For an efficient design of a pre-swirl system, a detailed understanding of the flow processes is essential.

One of the key issues within this context is the rotor-stator interaction. Principles of internal cooling-air systems of gas turbines have already been investigated [1,2]. However, the previous computational studies [3-5] were mainly considering two-dimensional axis-symmetrical arrangements, where a rotor-stator interaction problem, as such, does not arise. A three-dimensional simulation of a cover-plate receiver flow was presented in [6] and recently in [7-11]. However, in those studies [7-11], a quasi-steady formulation was employed without discussing its suitability for the considered flows. Indeed, to the best of the authors' knowledge, a thorough assessment of the accuracy of the quasi-steady formulation, within the context of pre-swirl systems, has not yet been presented.

Thus, the main emphasis of the present study is the assessment of the predictive capability of the quasi-steady formulation. Quasi-steady and fully transient approaches have been applied for treating the rotor-stator interface of a pre-swirl arrangement. The results of the transient formulation have been taken as reference for assessing the quasi-steady approach. The first part of the study has been performed for a two-dimensional, planar geometry. Subsequently, the analysis has been extended to three-dimensional systems, which will be discussed in the oral presentation.

2 Modelling

The modelling has been based on the general-purpose CFD code Fluent [12], which employs the finite volume method in conjunction with an unstructured grid definition. The flow has been modelled as compressible (ideal gas) and turbulent. For treating the velocity-pressure coupling, the SIMPLEC procedure [13] has been applied in the quasi-steady calculations, whereas the PISO algorithm [14] has been used for the transient simulations. As turbulence model, a realizable high Reynolds number k-ε model [15] has been used. On solid surfaces, no-slip conditions apply for momentum equations, in conjunction with a non-equilibrium wall-functions approach [16]. An adiabatic wall has been assumed for the energy equation. At the inlet, the total pressure and the total temperature, at the outlet, the static pressure has been prescribed. For the convection terms, a second order upwind discretization scheme [17] has been used. For the time discretization, a second order implicit scheme [12] has been employed.

Alternative quasi-steady formulations for the rotor-stator interface are the "mixing plane" and the "frozen rotor" approaches. In both procedures, the flow in each of the stator and rotor sub-domains is formulated as a steady-state problem with respect to the sub-domain's reference frame. In the initial phase of the work, it was observed that the "mixing plane" approach is not convenient to solve flows exhibiting strong non-uniformities such as recirculation regions at the interface plane. Therefore, this approach was not considered any further.

Thus, the "frozen rotor" approach has been employed as the "quasi- steady" formulation. In this approach, the spatial variations of the variables are transmitted across the interface boundary, without any averaging, for a given relative position of the stator and rotor. Accuracy of the computations can be increased by considering a larger number of relative positions, which, then, approximate the transient flow as a series of steady-state solutions. In the present study, six equidistant positions within a pitch have been considered.

Of course, the most sophisticated and accurate approach is the fully unsteady treatment of the flow, considering the relative motion of the stator and rotor in time, which has been performed, here, by means of a sliding mesh technique.

3 Results

In the first part of the study, the problem has been formulated as a two-dimensional, planar problem, in the sense of a linear cascade. A sketch of the solution domain and the boundaries is shown in Fig. 1. In generating the grid, a structured organization has been used in most parts, especially for resolving the wall boundary layers. For preventing skewed quadrangles, this has been amended by unstructured regions, as transition between structured zones. A detail view of the grid near the pre-swirl nozzle exit is also shown in Fig. 1.

A grid independency study has been carried out, within the framework of the quasi-steady formulation. For all grids, it has been ensured that the near-wall y+ values lie within the range of $30 < y^+ < 100$, in most parts (with slight local exceptions), as this represents [18] an optimal range for the application of the wall-functions approach. Fig. 2 shows the variations of the predicted mass flow rate ($\dot{m}$) through the system, and the relative total temperature (T_{trel}) at the receiver outlet, for an investigated case (displayed values are made dimensionless by the value obtained for the finest grid (reference value)). The grid with about 12,000 cells has been accepted to provide sufficient grid independency for present purposes (Fig.2).

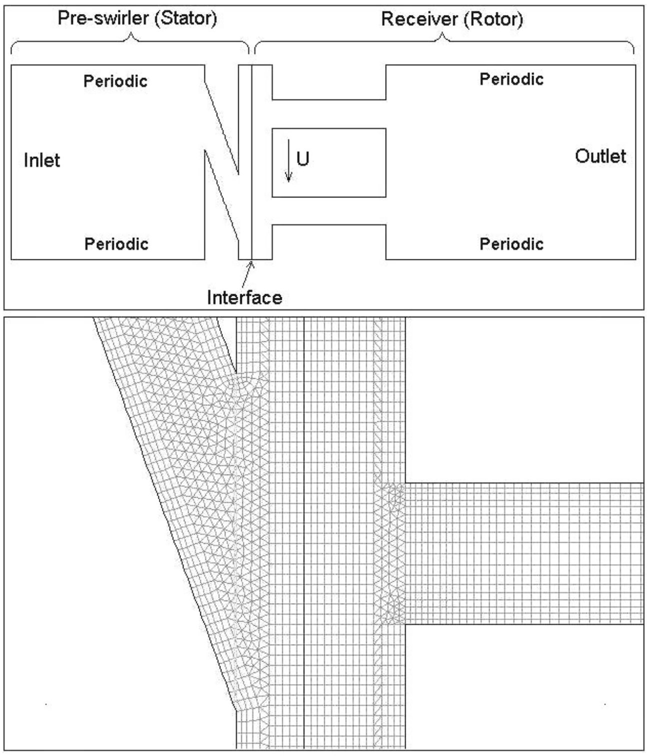

Fig. 1. Solution domain (above) and detail grid (below).

In effort of applying a sufficiently accurate temporal resolution, the time step size has been chosen in such a way that the maximum cell Courant numbers have remained well below unity in all transient computations. In the transient simulations, attention has been paid to the establishment of a fully periodic flow structure in time, by computing the necessary amount of cycles

For the case with Π =1.2 and $U/C_{t,id}$ = 1.5, the streamlines computed by the quasi-steady formulation are compared with the instantaneous streamlines predicted by the unsteady approach for the same relative stator-rotor position, in Fig. 3. Hereby, the ideal absolute tangential velocity $C_{t,id}$ is obtained by the following expression

$$C_{t,id} = \cos\alpha \cdot \sqrt{2\frac{\kappa}{\kappa-1} R\, T_{t,0}\left(1-\left(\frac{p_{s,1}}{p_{t,0}}\right)^{\frac{\kappa-1}{\kappa}}\right)} \qquad (1)$$

As one can see in Fig. 3, the instantaneous streamlines by the unsteady approach show substantial differences from those obtained by the quasi-steady approach.

Fig. 4 compares, for the same case, the isotherms predicted by the quasi-steady approach, with the instantaneous isotherms predicted by the unsteady approach, for the same relative stator-rotor position. One can see that the predicted isotherms by both approaches show large differences.

The variation of the normalized mass flow rate by the relative stator-rotor positioning in the quasi-steady and unsteady approaches are shown in Fig. 5, for the mass flow rate through a single receiver hole and for the total mass flow rate, for the same case. The mass flow rate through a single receiver hole shows significant variations around its mean value, whereas such variations turn out to be less predominant for the total mass flow rate, meaning that the mass flow rate through a single receiver hole is "compensated" by the mass flow rate through the other one, to a certain extent. This trend has been predicted by the both approaches, whereas this "compensation" mechanism seems to be more effective for the quasi-steady formulation. For the total mass flow rate, the quasi-steady formulation shows a slight phase shift and slightly smaller amplitudes compared to the unsteady formulation, and a smaller mean value (Fig. 5).

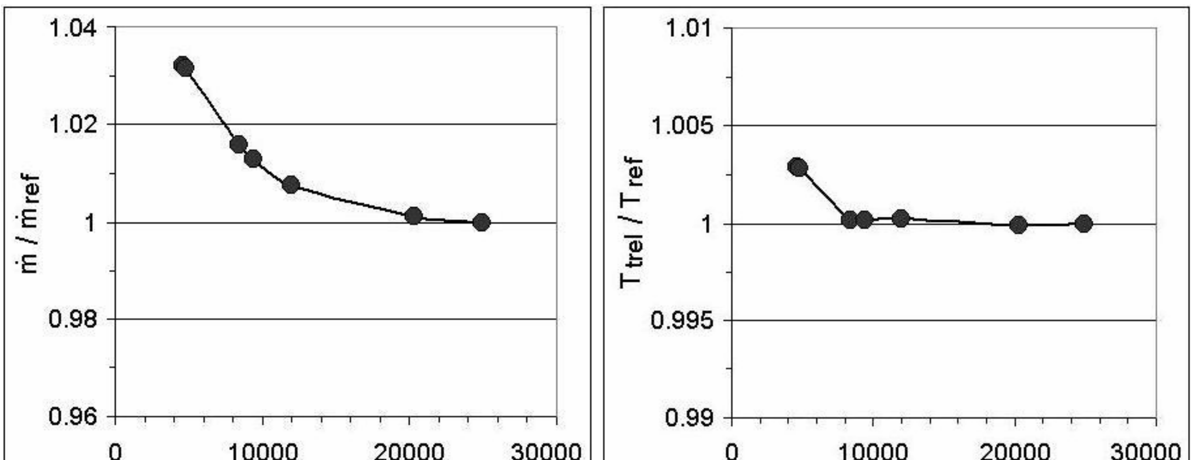

Fig. 2. Variations of mass flow rate (left) and receiver outlet relative total temperature (right) with grid size.

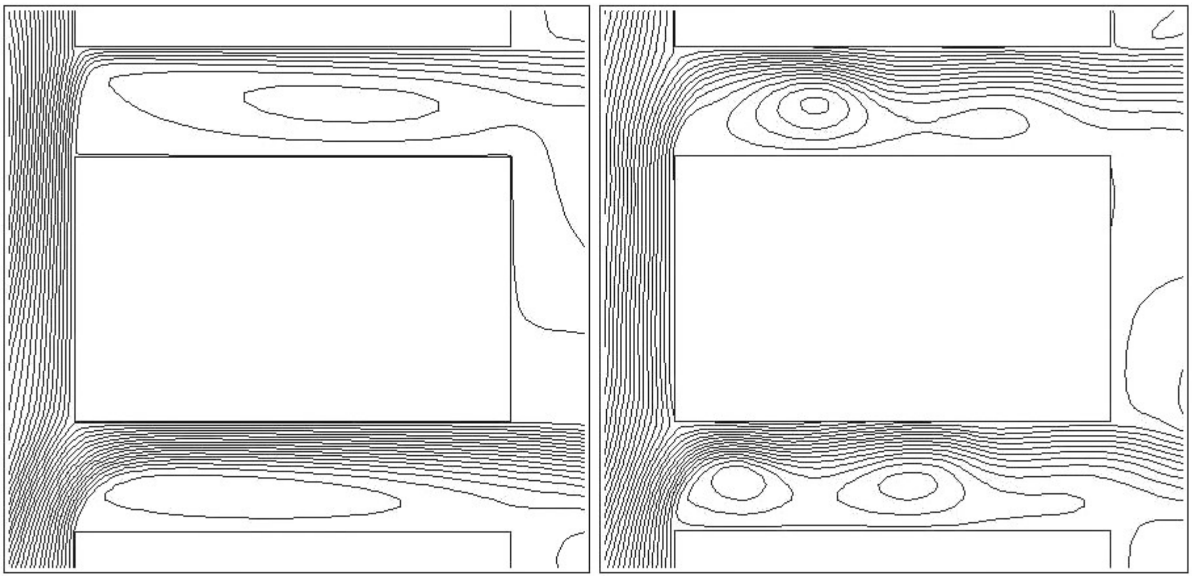

Fig. 3. Predicted streamlines for Π =1.2 and $U/C_{t,id}$ = 1.5; Left: Quasi-steady, Right: Unsteady.

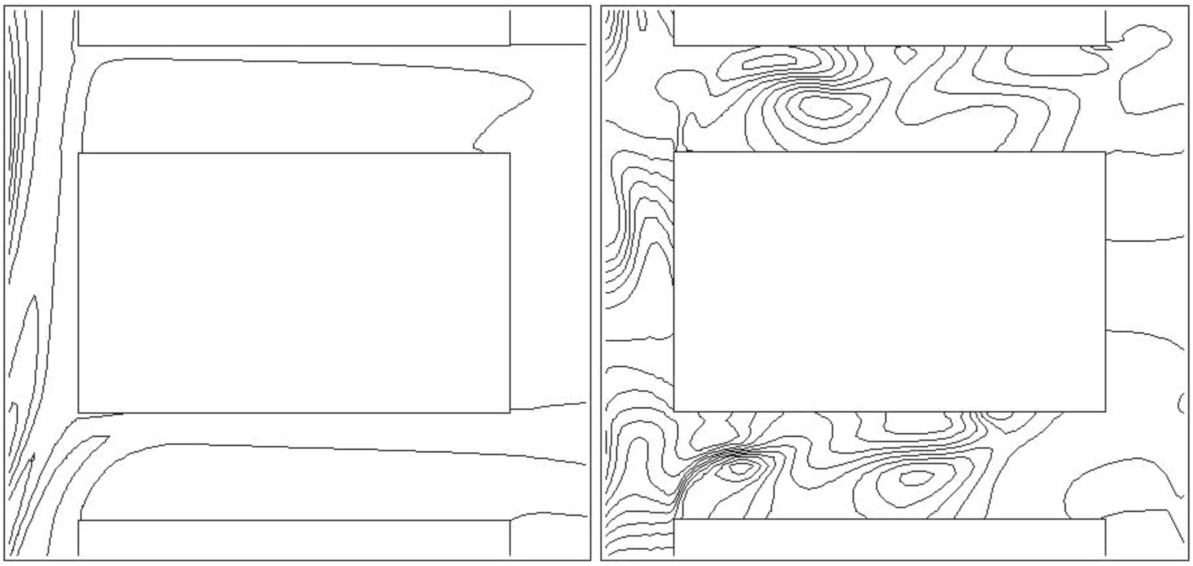

Fig. 4. Predicted isotherms for Π =1.2 and $U/C_{t,id}$ = 1.5; Left: Quasi-steady, Right: Unsteady.

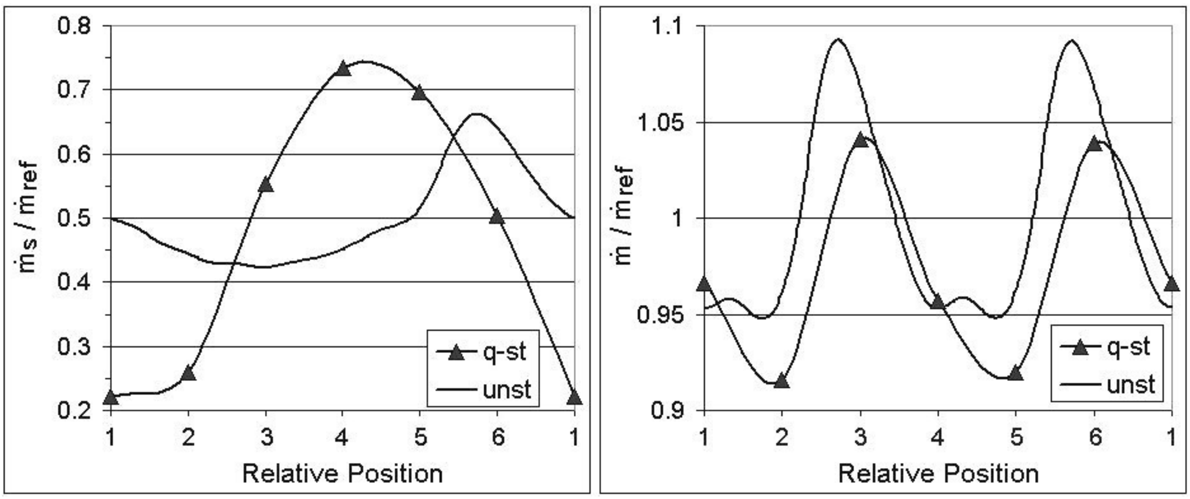

Fig. 5. Variations of mass flow rate with relative stator-rotor positioning for Π =1.2 and $U/C_{t,id}$ = 1.5 as predicted by quasi steady (q-st) and unsteady (unst) approaches;
Left: Mass flow rate through a single receiver hole, Right: Total mass flow rate.

The important question within this context is how the important time averaged flow parameters of the unsteady prediction differ from those obtained by the quasi-steady formulation.

This is demonstrated in Fig. 6, which shows the mass flow rate ($\dot{m}$) through the system and the pre-swirl nozzle discharge coefficient ($C_{D,PSN}$) as functions of the velocity ratio $U/C_{t,id}$, for Π = 1.2 as predicted by both formulations ($\dot{m}$ is made dimensionless by the result of the unsteady approach for the smallest velocity ratio: $\dot{m}_{ref}$).

The pre-swirl nozzle discharge coefficient ($C_{D,PSN}$) is defined to be the ratio of the resulting mass flow rate to the ideal one. This is computed through the following expression

$$C_{D,PSN} = \frac{\dot{m}_{predicted}}{\dot{m}_{id}} = \frac{\dot{m}_{predicted}}{\frac{A_{PSN}\, p_{t,0}}{\sqrt{R\, T_{t,0}}}\sqrt{\frac{2\kappa}{\kappa-1}\left(\left(\frac{p_{s,1}}{p_{t,0}}\right)^{\frac{2}{\kappa}} - \left(\frac{p_{s,1}}{p_{t,0}}\right)^{\frac{\kappa+1}{\kappa}}\right)}} \qquad (2)$$

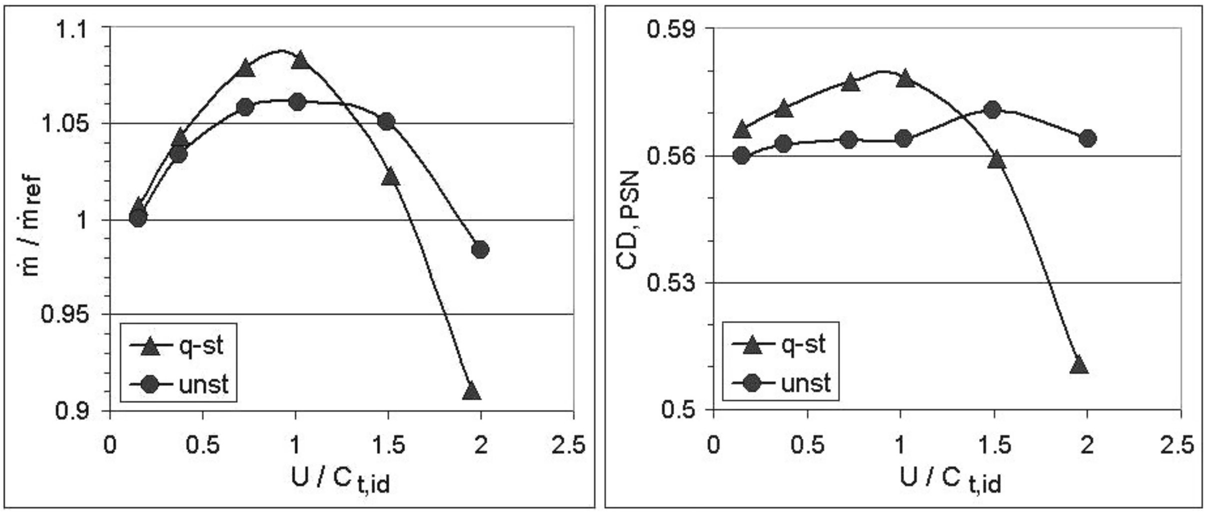

Fig. 6. Mass flow rate (left) and pre-swirl nozzle discharge coefficient (right) as function of velocity ratio $U/C_{t,id}$, as predicted by quasi-steady (q-st) and unsteady (unst) approaches.

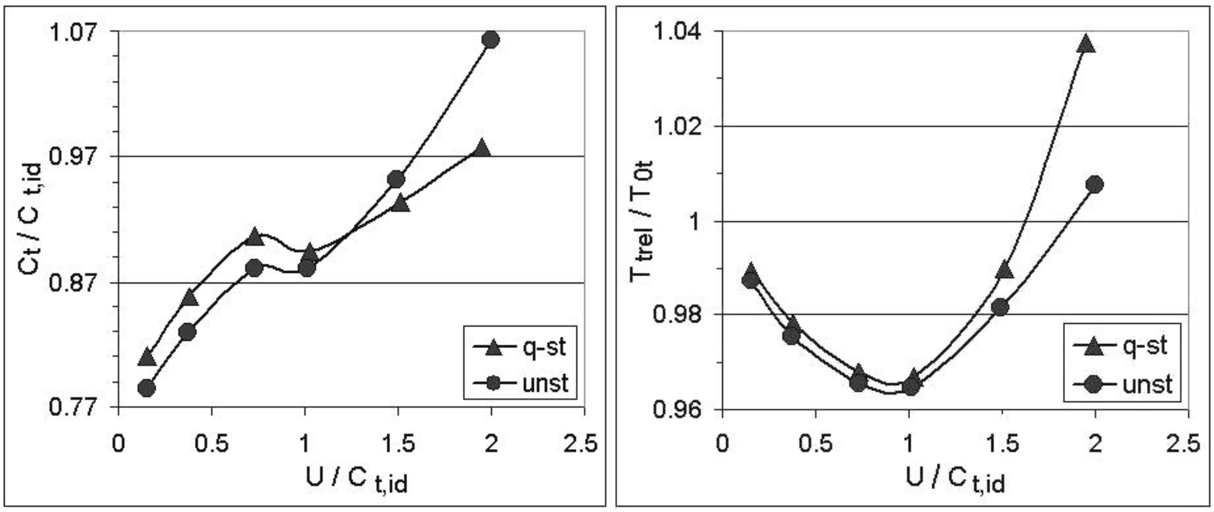

Fig. 7. Non-dimensionalized average tangential velocity at inlet of receiver holes (left) non-dimensionalized average total relative temperature at outlet of receiver holes (right) as function of velocity ratio $U/C_{t,id}$, as predicted by quasi-steady (q-st) and unsteady (unst) approaches

Fig. 7 shows the variation of the non-dimensionalized average absolute tangential velocity at the inlet of the receiver holes and the non-dimensionalized average total relative temperature at the outlet of the receiver holes.

In Figs. 6 and 7, one can observe that the quasi-steady and unsteady approaches agree quite well for lower values of the velocity ratio, such as $U/C_{t,id} < 0.5$. For the medium values, i.e. within the range about $0.5 < U/ C_{t,id} < 1.5$, the predictions show certain deviations, which, however, remain within the range of 3% for the mass flow rate, the pre-swirl nozzle discharge coefficient and the average absolute tangential velocity at receiver nozzle inlet, and well below 1% for the average relative total temperature at receiver nozzle outlet. For the larger values of the velocity ratio $U/C_{t,id}$, such as for $U/C_{t,id} > 1.5$, the discrepancy between the both predictions starts to steadily increase. For the investigated case, this can be considered to mark the validity limit of the quasi-steady approach.

4 Conclusions

Rotor-stator systems as encountered in pre-swirl arrangements of gas turbines have been investigated using quasi-steady and fully transient approaches. It has been demonstrated that the quasi-steady formulation can deliver similar results to the unsteady analysis, for rather low values of the velocity ratio $U/C_{t,id}$, whereas the discrepancy between the two formulations starts to grow for higher values. This implies that such limits needs to be checked for an accurate application of the quasi-steady formulation. The discussion will be extended to further cases including three-dimensional problems in the oral presentation.

References:

[1] Owen, J. M. and Rogers, R. H., *Flow and Heat Transfer in Rotating-Disc Systems*, *Vol. I: Rotor-Stator Systems*, Research Studies Press, Taunton; Wiley, New York, 1989.

[2] Owen, J. M. and Rogers, R. H., *Flow and Heat Transfer in Rotating-Disc Systems*, *Vol. II: Rotating Cavities*, Research Studies Press, Taunton; Wiley, New York, 1995.

[3] Mirzae, I., Quinn, P., Wilson, M. and Owen, J. M., Heat Transfer in a Rotating Cavity with a Stationary Stepped Casing, *ASME Journal of Turbomachinery*, Vol. 121, 1999, pp. 281-287.

[4] Karabay, H., Chen, J. -X., Pilbrow, R., Wilson, M. and Owen, J. M., Flow in a "Cover-Plate" Pre-Swirl Rotor-Stator System, *ASME Journal of Turbomachinery*, Vol. 121, 1999, pp. 160-166.

[5] Pilbrow, R., Karabay, H., Wilson, M. and Owen, J. M., Heat Transfer in a "Cover-Plate" Pre-Swirl Rotating-Disk System, *ASME Journal of Turbomachinery*, Vol. 121, 1999, pp. 249-256.

[6] Popp O., Zimmermann, H. and Kutz, J., CFD Analysis of Coverplate Receiver Flow, *ASME Journal of Turbomachinery*, Vol. 120, 1998, pp. 43-49.

[7] Yan, Y., Gord, M. F., Lock, G. D., Wilson, M. and Owen, J. M., Fluid Dynamics of a Pre-Swirl Rotor-Stator System, *ASME Journal of Turbomachinery*, Vol. 125, 2003, pp. 641-647.

[8] Schneider, O., Dohmen, H. J., Benra, F.-K. and Brillert, D., Investigation of Dust Separation in the Internal Cooling Air System of Gas Turbines, *ASME Paper No GT2003-38293*, 2003.

[9] Schneider, O., Dohmen, H. J., Benra, F.-K., Analysis of Dust Separation Inside Pre-Swirl Cooling Air System, *ISROMAC10-2004-029*, 2004.

[10] Benim, A. C., Brillert, D. and Cagan, M., Investigation into the Computational Analysis of Direct-Transfer Pre-Swirl Systems for Gas Turbine Blade Cooling, *ASME Paper No GT2004-54151*, 2004.

[11] Benim, A. C., Cagan, M., Bricaud, C., Bonhoff, B., Brillert, D., Computational Analysis of Flow and Heat Transfer in a Direct-Transfer Pre-Swirl System, *Proceedings of the Sixth European Conference on Turbomachinery – Fluid Dynamics and Thermodynamics,* 7-11 March 2005, Lille France, Vol. II, 2005, pp. 864-875.

[12] Fluent 6 User's Guide, Fluent Inc. Lebanon, 2004.

[13] Vandoormal, J. P. and Raithby, G. D., Enhancements of the SIMPLE Method for Predicting Incompressible Fluid Flows, *Numerical Heat Transfer,* 7, 1984, pp. 147-163.

[14] Issa, R. I., Solution of the Implicitly Discretised Fluid Flow Equations by Operator-Splitting, *Journal of Computational Physics,* 62, 1986, pp. 40-65.

[15] Shih, T.-H., Liou, W. W., Shabbir, A. and Zhu, J., A New k-ε Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation, *Computers and Fluids,* 24, 1995, pp. 227-238.

[16] Kim, S.-E., and Choudhury, D., A Near Wall Treatment Using Wall Functions Sensitized to Pressure Gradient, *ASME FED Vol. 217 Separated and Complex Flows,* 1995.

[17] Barth, T. J. and Jespersen, D., The Design and Application of Upwind Schemes on Unstructured Meshes, *Technical Report AIAA-89-0366,* 1989.

[18] Benim, A. C. and Arnal. M., A Numerical Analysis of the Labyrinth Seal Flow, in: S. Wagner, E. H. Hirschel, J. Périaux and R. Piva (Eds.), *Computational Fluid Dynamics'94,* Vol. 2, John Wiley & Sons Ltd., New York, 1994, pp. 839-846.