Mechanics and Mechanical Engineering
Vol. 12, No. 2 (2008) 111–124

Quasi–unsteady simulations of the clocking phenomena in the two–stage turbine

Krzysztof Sobczak
Antoni Smolny
Jarosław Błaszczak
Technical University of Łódź,
Institute of Turbomachinery
Wólczańska 219/223, 90-924 Łódź, Poland

Received (25 October 2008)
Revised (30 October 2008)
Accepted (15 December 2008)

The numerical investigations of a flow in the model turbine are presented. The clocking phenomena in the two–stage turbine with low–aspect and highly loaded blade rows are studied. The quasi–unsteady simulations by means of the frozen rotor approach were conducted for several circumferential relative positions of stators. The optimum clocking position was predicted on the basis of the estimated turbine efficiency. The effects of clocking on wake interactions and blade loading variations are presented and discussed.

Keywords: Clocking phenomena, numerical simulations, frozen rotor

1. Introduction

The relative motion of stationary and rotational blade rows in turbines generate inherently unsteady flow fields. Due to aerodynamic interactions, pressure and other flow parameters change considerably in time. As a result, unsteady effects of the successive blade row interaction should be considered during the design process.

At present efficiency levels of turbine components are already much higher than 90% and their further improvement is very limited indeed. Unsteadiness exerts a considerable influence on turbine or compressor efficiency. Its impact can be either beneficial or not. Therefore, a thorough understanding of unsteady flow field interactions has become essential for designers to achieve higher aerodynamics efficiency.

Clocking or indexing is currently an attractive technique and an effective way to modify the interaction intensity and improve the turbine performance. The basic idea of clocking is to improve the overall turbine or compressor efficiency and durability by a variation of relative circumferential positions of stationary or/and rotating blade rows in consecutive stages. This technique has a potential to control unsteady interaction mechanisms, in particular a loss generation. The largest efficiency increase can be achieved in the case when subsequent stators and rotors have equal blade counts.

It is known that the maximum efficiency is attained when the preceding stator/rotor vane wake impinges the leading edges of the next stator/rotor vanes. The first results of the experimental study of clocking phenomena were reported by Huber et al. [1] for a two–stage turbine. They showed a 0.8% efficiency variation at midspan due to clocking of the second vane. From this time the problem has been studied by many researchers, both experimentally and numerically.

Unsteady 3D Navier–Stokes calculations carried out by Arnone et al. [2] in a three–stage low–pressure turbine showed a 0.7% efficiency variation due to clocking. Rainmoller et al. [3] investigated the clocking phenomena in a 1 and 1/2 stage subsonic turbine. The authors found a 1% efficiency variation at midspan. The stator and rotor clocking in a highly loaded high-pressure turbine was studied by Jouini et al. [4]. They concluded that a 4% change in efficiency could be detected. Additionally, an increase in the relative unsteadiness level on the second vane for the optimal clocking position was revealed. The experimental investigations of Krysiński et al. [5] have shown that efficiency changes near the hub are significantly larger than at midspan and this results from an interaction of secondary flows and wake.

The research conducted currently by the authors of this paper focuses on the flow interaction accompanying different relative positions of subsequent stator vanes at different clocking positions, which causes nonlinear behavior of the overall stage. A better understanding of this problem is of key importance to control the interaction mechanism that is responsible for loss generation processes. An accurate prediction of the flow physics in a multistage turbine can deliver the detailed data for this analysis. Together with experimental investigations, numerical simulations can lead to a deeper understanding of the unsteady and time averaged flows in multistage turbomachines.

Computational Fluid Dynamics (CFD) simulations are more and more popular due to a significant progress in the capabilities of numerical algorithms, turbulence modeling and available computer resources. Despite the fact that properly conducted unsteady simulations of flows can reveal the phenomena accompanying clocking in a turbine, they are still limited. It results from the fact that enormous computer resources must be employed and the applied models neglect some effects of real physics.

In the investigations presented here, a 3D quasi–unsteady CFD calculation procedure was used to investigate the flow interaction in a modern two–stage turbine for some chosen stator clocking positions. The fully unsteady simulations were not possible due to limitations of available computers. Instead, steady simulations were conducted for a few stator/rotor (frozen rotor) relative positions. The numerical investigations were focused mainly on the flow phenomena related to the stator clocking and its effect upon the 3D flow field in stator and rotor passages of a highly loaded low–aspect two-stage turbine. The numerical investigations, which run in parallel, can significantly support the experimental investigations of the flow field in the turbine.

2. Simulations of the stator–rotor interaction

Flows in gas turbines are usually very complex. They are three–dimensional (3D), viscous and unsteady. Despite the fact that the present–day CFD codes are able to take into account all these factors, their practical application is limited by the sizes of tasks and the capacity of available computer resources. The strongest limitation arises from unsteadiness of the flow field. In this case the computational capability is determined not only by speed of computer processors but also by the data storage capacity of the hard drive.

The most important factor of unsteadiness in turbines comes from the relative blade row motion in multiple frames of reference. This leads to an unsteady interaction of pressure fields, shock waves and wakes between stators and rotors. The CFD provides different approaches to take into account changes in the reference frames. The ANSYS CFX code offers the following methods:

- stage method,
- frozen rotor method,
- transient rotor–stator method.

They differ significantly as regards the way the data are transferred between stationary and rotational frames. In the stage method (known also as the mixing plane method), steady state computations are carried out. At the interface between different frames of reference, parameter fields are circumferentially averaged and then exchanged in the form of boundary conditions. Such an approach removes the transfer of all unsteady structures from one vane to another one. An influence of wakes, shock waves and separations is taken into account only through their effect on circumferentially averaged parameter profiles. However, the stage method requires moderate computational resources. Consequently, it is widely used in numerical simulations of multistage machines.

The frozen rotor method maintains the velocity field in the absolute frame of reference during the transfer between stationary and rotational wheels. As a result, velocities are recalculated from one frame of reference to another one, which leads to preservation of unsteady structures in the transition between stators and rotors. However, a drawback of the method follows from the fact that rotors maintain their fix positions in respect to stators in the solution. Consequently, wakes from the upstream vane are directed always into the same angular position of the next vane.

In both the above–described approaches, flow fields are solved as stationary and the unsteady interaction is only approximated. The transient rotor–stator method (the so–called sliding mesh) predicts the true transient interaction of the flow between stator–rotor passages. In this approach, the transient relative motion between components on each side of the connection is simulated. It ultimately accounts for all interaction effects between components that are in a relative motion to each other. The interface position is updated each time step, as the relative position of grids on each side of the interface changes. Thus, the proper application of this method provides the results that are the closest to reality. However, as has been mentioned above, unsteady computations are very demanding as far as the time to obtain solutions and the computer storage capacity are concerned. Therefore, an application of this method is still limited in multi–stage turbomachinery simulations.

In cases of the frozen rotor method and the transient rotor–stator method application, an additional problem appears when the number of blades of stators and rotors is different. The circumferential averaging in the stage method requires calculation of the flow in only one passage of stators and rotors. In the remaining methods, the pitches of stators and rotors should be the same. If there are different pitches, then codes like ANSYS CFX usually provide a possibility to simulate a single passage by the circumferential rescaling of fields at the interface, but when the pitches are significantly different, it is a source of additional calculation errors.

In the case of numerical investigations of clocking phenomena, the stage method by its nature cannot be applied. The best solution is the transient rotor–stator method. However, the size of the task in the case of multiple–stage turbines makes its application still limited. Thus, the frozen rotor method is a reasonable alternative. The simulations in a few stator–rotor (the frozen rotor method) relative positions bring the results even closer to reality.

According to Bohn et al. [6], steady state numerical simulations of clocking phenomena with the frozen rotor method allow one to determine properly the optimal clocking position in a multistage turbine. However, as far as the absolute values of efficiency changes are concerned, some significant discrepancies appear. As a result, this approach seems to be a convenient tool for the initial analysis of the optimal clocking position preceding the fully unsteady simulations.

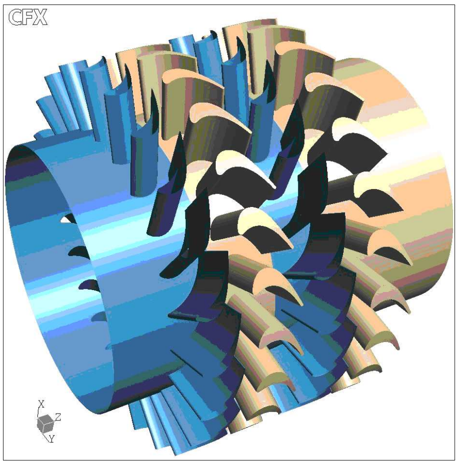

Figure 1 Two-stage model turbine

3. Numerical procedure

Numerical simulations give a deep insight in the flow structure and can be very helpful to adjust properly the instrumentation to achieve high quality experimental results. Thus, they are usually the first step in the investigation process. It is beneficial if rapid numerical CFD data are available for a research group planning experiments and conducting measurements. On the other hand, detailed experimental results can be of a great importance for very detailed numerical investigations concluding the research activity.

This approach should be beneficial in the present investigations of clocking effects. It is supposed to facilitate the determination of flow phenomena interactions in the turbine, in particular wake and secondary flows and related loss producing mechanisms.

In this paper the numerical investigations of a flow, which are the preliminary stage of the new turbine studies, are presented.

4. Turbine geometry

A series of experimental investigations will be conducted on a newly designed two–stage model turbine for a low–aspect and highly loaded blade row version. The turbine, presented in Fig. 1, consists of two geometrically identical turbine stages. Each stator is composed of 22 blades, whereas the rotors consist of 23 blades. All the blades are prismatic. The trailing edges of stators and the mass centers of rotor blade cross–sections are located along the radial direction.

The experimental facility is a large–scale one. The shroud diameter is equal to 0.514 m. The low speed Mach number attains $Ma = 0.35$. The stator Reynolds number based on the chord is $5 \cdot 10^5$ and for the rotor it is equal to $3 \cdot 10^5$. The size of the test turbine allows a good spatial resolution to be obtained in the steady and unsteady flow at traverse planes located downstream of blade rows.

The size of blades results from the requirement to have a sufficiently large chord and span blades, which facilitate detailed measurements for chosen clocking positions. It is of a great importance to avoid merging endwall (hub and shroud) structures into a single loss structure. Literature studies, e.g. [7], show that the applied blade height equal to 82 mm will be sufficient to prevent the loss merging flow effect. In the experimental investigations, steady data will be acquired with five–hole probes with a sphere diameter of 2.5 mm, whereas unsteady measurements will be conducted with 3–sensor hotwire probes and fast response pressure probes located on the stator wall to investigate the vane unsteady pressure.

5. Clocking positioning

In the numerical simulations, the clocking position of the first stator is changed. Four stator–to–stator relative positions, evenly distributed along the stator pitch, are considered (every 4.09^o). They are presented in Fig. 2. The initial clocking position (CP 0) corresponds to the same angular location of both stator blades. The even numbering of the clocking position comes from the plans of broader numerical simulations, i.e., for 8 clocking positions, which are to be carried out. Eight stator–rotor relative positions (the frozen rotor method) are taken into account (every 1.96^o).

6. Boundary conditions

The computational hexahedral mesh, presented in Fig. 3, consists of ∼1250000 nodes. It has been generated by means of ANSYS TurboGrid. In order to achieve a proper mesh element quality in the blade boundary layer region, an O-grid topology is introduced around the blade surface. In the boundary layer zones, an automatic wall function is applied. The size of the task has not allowed one to refine the mesh enough for a direct solution of boundary layers.

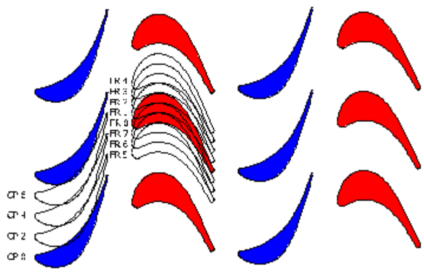

Figure 2 Clocking and frozen rotor positions under analysis

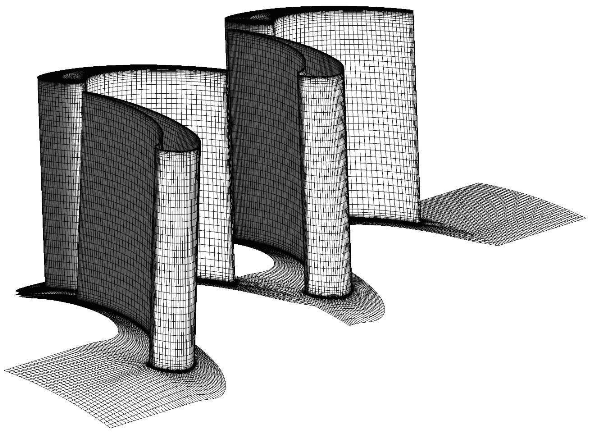

Figure 3 Computational mesh on blades and hubs

Total pressure, total temperature, axial flow direction and low turbulence intensity were applied as boundary conditions at the inlet. At the outlet, static pressure was imposed. The boundary conditions are presented in Tab. 1 in detail. The walls were treated as the adiabatic and smooth ones. The rotational periodicity interface was applied on mesh sides along the circumferential direction. All the boundary conditions presented were estimated on the basis of the capabilities of the experimental test stand.

Table 1 Boundary conditions

Inlet total pressure	125 000 Pa
Inlet total temperature	325 K
Inlet flow angle	0^o
Inlet turbulence intensity	1%
Outlet static pressure	100 000 Pa
Rotor speed	3200 rpm (53.3 Hz)

7. Numerical scheme

The ANSYS CFX code, which is based on the finite volume method, was used in the simulations of the ideal gas (air) flows through the two–stage axial turbine. Three–dimensional, steady, compressible, turbulent simulations were carried out. The SST (Shear Stress Transport) turbulence model with an automatic wall function was applied. The second order space discretization was used. Due to the limitations of computational resources, single passages of turbine blade rows were considered. Therefore, because the stator/rotor pitch ratio was equal to 22/23, a circumferential field rescaling was used at the interfaces.

A good level of convergence was obtained in all the cases analyzed. Root Mean Square (RMS) residuals in all cases under analysis did not exceed $2.0 \cdot 10^{-6}$. Mass, momentum and energy flows were properly balanced in the computational domain. Unbalanced flows were well below of 0.01% of the mean flows.

8. Discussion of the results

8.1. *Efficiency distributions*

In the investigations under consideration, the following total–to–total efficiency definition is applied:

$$\eta = \frac{\gamma}{\gamma - 1} \cdot \frac{\ln (T_2/T_1)}{\ln (p_2/p_1)} \tag{1}$$

where:

T_1, p_1 – mass–averaged temperature and pressure at the cascade inlet
T_2, p_2 – mass–averaged temperature and pressure at the cascade outlet
γ – specific heat ratio

In Fig. 4 the overall efficiency of the turbine is presented. In order to show the tendency more clearly, two periods of efficiency are shown. In the first step the parameters (pressure and temperature) were mass averaged in the outlet and inlet cross–sections and then the efficiency was calculated for particular frozen rotor positions. In the second step, the values of efficiency from all clocking positions were averaged. Consequently, a rise in the overall efficiency is observed for the clocking position CP 2. On the other hand, the lowest efficiency is observed for the clocking position CP 6 (half a pitch displacement). Despite the fact that maximal and minimal efficiency positions are easily distinguishable, the efficiency variability is very low – 0.1 %.

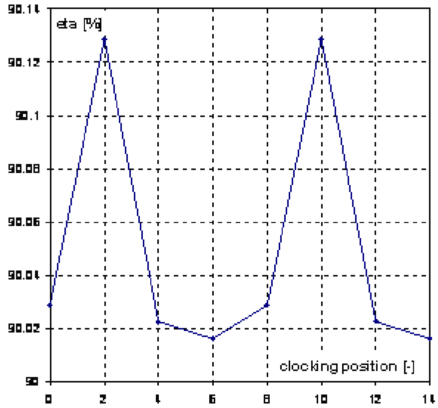

Figure 4 Averaged efficiency for different clocking positions

The reason of such a low efficiency rise for the CP 2 position is a result of significant differences in the optimal clocking position along the height of the blade. Efficiency distributions in a few spanwise positions are presented in Fig. 5. In this case the circumferential averaging of parameters was conducted for chosen spanwise positions and the efficiencies for all frozen positions were determined in the first step. Next, mean values of efficiency for all frozen positions were calculated for the analyzed clocking positions. According to that diagram, the highest efficiency level is obtained in the channel part in the hub vicinity. In the upper part of the span, efficiency drops considerably $\sim$ 4%.

Table 2 Spanwise changes of efficiency and optimal clocking positions

Spanwise position	10%	25%	50%	75%	90%
Efficiency difference	0.51%	1.07%	0.33%	0.61%	0.76%
Best clocking position	CP 0	CP 2	CP 0	CP 2	CP 2
Worst clocking position	CP 4	CP 6	CP 4	CP 6	CP 0

The efficiency difference values, as well as the best and worst clocking positions for different spanwise locations are summarized in Table 2. The highest changes in the efficiency level for different clocking positions are observed for 25% of the blade height, where the efficiency variability exceeds 1%. It is worth noticing that some significant changes take also place in the top section of the channel. The smallest differences are revealed for the midspan of the turbine. Despite the fact that the efficiency differences are considerable for all span levels, their optimal positions change considerably. As a result, the overall efficiency changes described above are very slight. Efficiency changes at the hub (10%) and the shroud (90%) are out of phase with those observed in the channel center. A detailed look at Table 2 indicates that if the first vane wake were properly aligned with the second vane leading edge from the hub to the tip, a nearly 0.7% improvement in the turbine performance would be possible.

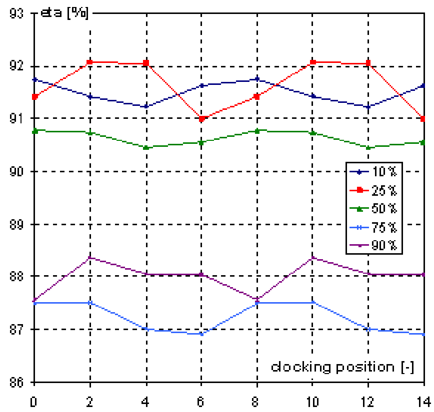

Figure 5 Averaged efficiency along the blade height at different clocking positions

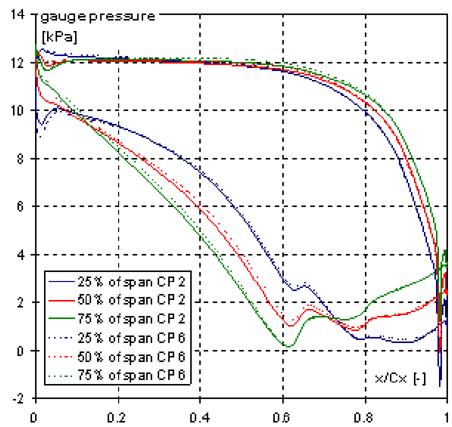

Figure 6 Blade loading for the 2nd stator at 25%, 50% and % of span for the maximal(CP 2) and minimal (CP 6) efficiency clocking positions

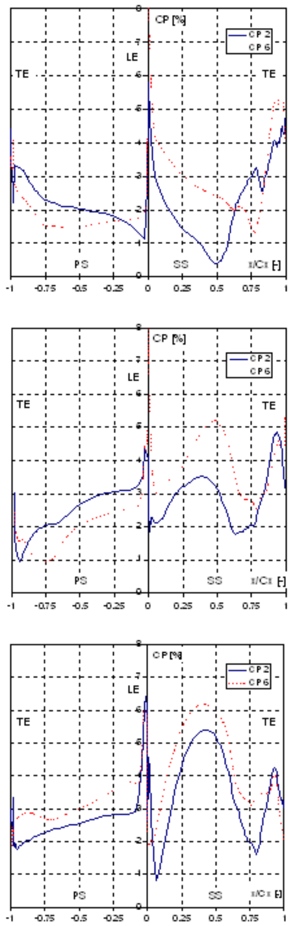

Figure 7 Variable pressure coefficient distributions around the blade for the 2nd stator at 25%, 50% and 75% of span for the maximal (CP 2) and minimal (CP 6) efficiency clocking positions

9. Blade loading

The second stator static pressure distribution close to the turbine hub, at midspan and close to the shroud for the optimum and worst clocking positions, is presented in Fig. 6. The pressure reduction at the suction side, as well as a slight drop at the pressure side is observed up to 70% of the blade chord for the most optimal clocking position. The radial position of the trailing edge of the first stator leads to a lower load of the vane and a reduction of secondary flow effects nearby the turbine hub. For the optimum clocking position, the first stator wake is better aligned with the leading edge of the second stator, which leads to a reduction of velocity at the suction side of the blade from the hub to the tip. Assuming that dissipation coefficients are constant, according to Denton [8], boundary losses are proportional to the third power of velocity at the boundary layer edge. This estimation indicates that a loss can be reduced. The relative motion of adjacent blade rows, simulated in this case by changes of the frozen rotor positions, induces a variety of unsteady flow phenomena like a potential flow interaction, wake interaction and vortex interaction, which is the most significant in turbines. Clocking affects these mechanisms and different modifications can be observed depending on clocking positions. A variation of pressure amplitudes for the considered clocking position is a superposition of the wake and potential effects. Fig. 7

Unsteady pressure coefficient distributions are presented in Fig. 7. This coefficient is defined as follows:

$$CP = \frac{p_{\max} - p_{\min}}{p_{ref}}$$

where:

p_{max}, p_{min}	–	maximal and minimal values of pressure in the same point on the stator blade for different frozen rotor positions
p_{ref}	–	reference pressure (pressure drop in the turbine)

This parameter was obtained on the basis of the static pressure variation for different frozen rotor positions. The analysis was carried out at the hub, midspan and shroud region of the span height. According to coefficient distributions, the maximum efficiency is associated with the minimum variation of pressure in the leading edge zone and on the suction side of the vane. On the pressure side, an opposite situation occurs with an exception of 75% of span. These distributions show that clocking affects pressure distributions in different ways, depending on the blade surface location. Cizmas and Dorney [9] state that a coupling resonance takes place between the inlet guide vane wake and the second stator at the maximum efficiency. This coupling resonance energizes the boundary layer on the suction side of the stator, leading to a reduction of the vane boundary layer loss. In the results presented, the opposite behavior can be observed. The pressure variation intensification takes place but on the pressure side of the blade. However, one has to remember that this parameter does not represent pressure unsteadiness but only its variation resulting from different frozen rotor positions.

10. Entropy distributions

The entropy distributions were prepared in order to show and understand the phenomena responsible for a modification of the turbine efficiency at different clocking positions. The entropy generation takes place at the end walls and blades. It is a result of energy dissipations in the boundary layer. An additional growth of entropy is caused by secondary flows. However, the most important source of the entropy generation is the wake propagation downstream of each blade. Therefore, entropy distributions are usually used to illustrate wake trajectories in flow passages. On the basis of the flow field parameters, entropy changes in the passage are determined according the formula:

$$\Delta s = c_p \ln\left(\frac{T}{T_0}\right) - R \ln\left(\frac{p}{p_0}\right)$$

where:

T, p - local values of static temperature and pressure,

T_0, p_0 - reference temperature and pressure.

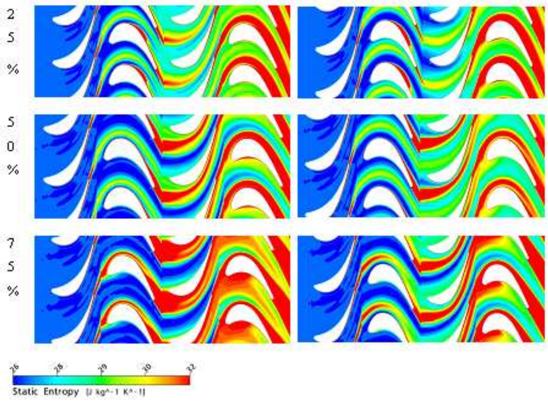

Figure 8 Entropy distributions at 25%, 50% and 75% of span for the clocking position CP 0 and 2

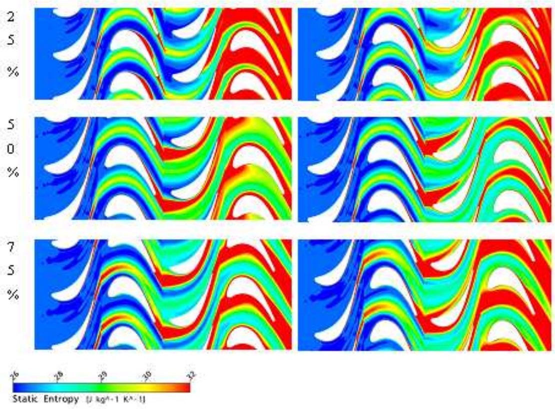

Figure 9 Entropy distributions at 25%, 50% and 75% of span for the clocking position CP 4 and 6

In the real case, the flow leaving the first stator vane row consists of wakes and vortex structures that extend downstream. Rotor blades intercept the continuously shaped wake from the upstream stator. As a result of the stator-rotor relative movement, wake segments are deformed (i.e., twisted, bowed, stretched, etc.) in the rotor passage. When the wake of the first stator enters the rotor passage, it is chopped by the passing rotor blades into discrete elements, which are then accelerated along the axial direction in the middle of the passage between the blades. Due to its interaction with the rotor blade boundary layer, a typical bowed contour develops. This interaction causes velocity disturbances at the edge of the boundary layer, which in turn results in losses in the boundary layer region. The entropy of the whole configuration shows intensities of the interaction and its influence on the loss generation.

In this paper, quasi–unsteady simulations are presented. Consequently, the unsteady behavior of the first stator wake described above cannot be properly reproduced. Nevertheless, some interesting observations can be made. The entropy distributions for 25%, 50% and 75% of the blade height for all the clocking positions are presented in Figs. 8 and 9. The most suitable frozen rotor positions were chosen to visualize the first stator wake trajectory in the first rotor and the inlet region of the second stator. The scale of the entropy contours was adjusted to show clearly the trajectory of the wake structure downstream of the first stator. In quasi-unsteady simulations, wake trajectories appear as continuous low energy/high entropy strips.

According to Arnone et al. [2], the clocking effects are directly connected to the wake trajectories in the turbine passages. Results obtained by many authors by means of the experimental investigations as well as numerical simulations with different methods and turbulence models suggest that the maximum efficiency is reached when the wake downstream the preceding stage stator or rotor reaches the leading edge of the following stage stator or rotor. Lower efficiency is obtained when the wake enters the passage between the next stator or rotor blades.

The entropy distributions presented in Fig. 8 and 9 confirm the above–mentioned observations. However, some discrepancies appear at 25% of the blade height, where the highest efficiency is observed when the wake enters the channel not exactly at the blade leading edge but at the pressure side of the blade.

11. Conclusions

A quasi–unsteady three–dimensional Navier–Stokes analysis was used to investigate the flow phenomena related to the stator clocking in a modern highly loaded two–stage turbine. Instead of full unsteady simulations, the steady simulations for a few relative stator–rotor (frozen rotor) positions were conducted. Despite the fact that not all phenomena can be properly reproduced, some interesting observations can be made.

Some variations in efficiency for different clocking positions of turbine stators are obtained. The efficiency changes are considerable for all span levels analyzed (up to 1%), however their optimal positions change as well. Consequently, the overall efficiency alternations between the best and worst clocking positions are very low (0.1%). In the highest efficiency positions, the wakes of the first stator impinge the leading edge of the second stator.

The clocking position of stators modifies pressure distributions as well as pressure variations on the second stator blade. The simulations reveal that the best clocking position is accompanied by a pressure reduction on the blade as well as by a lower variation of pressure on the suction side of the blade. Some rise of the variable pressure coefficient (CP) is observed on the blade pressure side.

It can be concluded that the quasi–unsteady method related to the frozen rotor approach has a potential to predict clocking effect phenomena and can be used in the preliminary stage of the design process to optimize clocking of the whole turbine. However, it cannot represent the turbine clocking phenomena as well as full unsteady calculations, therefore a more advanced method should be used in the final stage of the turbine design.

In further investigations, numerous comparisons of the simulation results with the experimental data are to be conducted. The investigations of the model turbine will be completed with fully unsteady simulations for some chosen clocking positions.

References

[1] **Huber, F.W., Johnson, P.D., Sharma, O.P., Staubach, J.B. and Gaddies, S.W.**: Performance Improvement through Indexing of Turbine Airfoils, Part I – Experimental Investigation, *Journal of Turbomachinery*, 118, 630-635, **1996**.

[2] **Arnone, A., Merconcini, M., Del Greco, A.S. and Spano, E.**: Numerical Investigation of Three-Dimensional Clocking Effects in a Low Pressure Turbine", Journal of Turbomachinery, 126, 375-384, **2004**.

[3] **Rainmoller, U., Stephan, B., Schmidt, S. and Niehuis**: Clocking effects in a 1.5 stage axial turbine – steady and unsteady experimental investigations supported by numerical simulations, ASME 2001-GT-0304

[4] **Jouini, D.B.M., Little, D., Bancalari, E., Dunn, M., Haldeman C. and Johnson**, P.D. Experimental Investigation of Airfoil Wake Clocking Impact on Aerodynamic Performance in a Two Stage Turbine Test Rig, ASME GT2003-38872.

[5] **Krysiński, J., Błaszczak, J. and Smolny, A.**: Stator Clocking Effect on 3D Flow in a Two-Stage Low Pressure Turbine, ASME Paper GT2005 –68811.

[6] **Bohn, D., Ausmeier, S. and Ren J.**: Investigation of the Optimum Clocking Position in a Two-Stage Axial Turbine, *Int. Journal of Rotating Machinery*, 3, 202-210, **2005**.

[7] **Traupel, W.**: Termische Turbomachinan, Springer Verlag, **1966**.

[8] **Denton, J.D.**: Loss Mechanisms in Turbomachines, *Journal of Turbomachinery*, 115, 621-656, **1993**.

[9] **Cizmas, P.G.A. and Dorney, D.J.**: The influence of clocking of unsteady forces of compressor and turbine blades, *International Journal of Turbo & Jet-Engines*, 17, (2), **2000**.