Proceedings of ASME Turbo Expo 2011
GT2011
June 6-10, 2011, Vancouver, British Columbia, Canada

GT2011-45832

ON A NUMERICAL METHOD FOR THE SIMULATION OF STEADY JET ARRAY INTERACTION WITH ROTATING COMPONENTS

Valery Kostege, Sergey Chernyshev
Alstom Power
Moscow
Russia

Vladimir Vassiliev
Alstom Power
Baden
Switzerland

ABSTRACT

Experimental and theoretical investigations show that unsteady effects like moving wakes, tip vortices, passing shocks, pulsating injections and other similar structures significantly affect the aerodynamic characteristics of turbines and compressors. They also influence the thermal state and lifetime of components. Therefore it is very important for designers of turbomachines to properly simulate these effects. On the other hand, time-accurate computations are still expensive and require substantial resources in CPU and computer memory. Moreover the elapsed time is high.

However in certain cases the numerical model for unsteady calculations can be simplified, allowing proper capture of the unsteadiness impact, but with much less required computing capacity. This makes the approach acceptable for design applications. Such a simplified method, applicable to a simulation of a steady jets array interaction with rotating components, is described in this paper. The advantages and limitations are discussed, and the validation results and application examples are presented.

INTRODUCTION

1. The importance of unsteady effects in turbomachinery has already been recognized in the earlier days of gas turbine development [1,2]. However only in the last two decades have these effects been extensively investigated, and knowledge of the unsteady flow phenomena has considerably improved.

One quite important unsteady effect inside multistage turbines and compressors is the influence of wakes from upstream rows on the downstream stages. Typical example of this interaction is the penetration of stator wakes inside rotor passages. This phenomenon has been investigated in numerous works, for which a complete review would require a separate paper. It is worthwhile to mention here the work of Hodson [3], who had drawn attention to the tendency of wakes in turbine cascades to migrate towards the suction side of the aerofoils and create the so-called “negative jet effect”. Hodson & Dawes [4] published one of the first unsteady CFD results and provided comparison with experiments. In several works (e.g. in [5-8]) the impact of wakes on transition, separation and reattachment has been investigated, and it was shown that wakes play an important role in the transition to turbulence in the separated shear layer, and can cause a separated boundary layer to reattach.

In a real 3D stage the unsteadiness in the rotor hub region results from both incoming vane wakes (wake interaction) and incoming vane hub secondary flows (vortex interaction). This combined effect has been investigated in [9-11].

In [12] the interaction of unsteady wakes with the shocks in transonic turbine has been numerically studied.

A periodic wake causes not only pressure fluctuation but also total temperature distortions, which becomes apparent when looking at the energy conservation equation for an adiabatic flow of a perfect gas (see e.g. [13])

$$Cp\,\frac{DTtot}{Dt} = \frac{1}{\rho}\frac{\partial p}{\partial t}$$

Therefore the unsteady wake influences the gas temperature distribution inside the turbine and affects the thermal state of components. Additional influence on metal temperatures comes from the wake´s impact on the heat transfer coefficients [14]. However, even more significant impact on the thermal state can be caused by hot streaks, which are formed in the combustor and penetrate inside the turbine.

The presence of combustor hot streaks results in the creation of additional unsteady transport mechanisms within the rotor, the generation of additional secondary flows (streamwise vorticity) and the preferential migration of hot and cold fluid within the rotor passage. These essentially unsteady effects are discussed in [15-16] and several other works.

Inlet flow distortion produces a significant impact on compressor behaviour, and the propagation of distortion is an unsteady process. As far as the compressor flow is decelerating it is prone to instability, which can lead to a surge or rotating stall. Both can be caused by different factors including inlet distortion, and they are essentially unsteady. In recent time the problem was extensively investigated (see for example [17-18]), although several questions are still remain open.

This short overview shows that unsteady effects play an important role in the multistage turbomachine and they should be accounted for in modern design practice. It is worth mentioning that in some special cases the flow, which is unsteady in the absolute coordinate system, can be made steady by a change of reference frame, but in turbomachine with two or more blade rows there is no such coordinate system. The flow here is fundamentally unsteady, and requires appropriate unsteady analysis methods.

2. CFD methods are widely used in current design practice, but in many cases they are applied for analysis of separate GT components in stationary frame of reference. The examples of such applications can be found in [19-22].

For the analysis of multistage turbomachines the mixing plane approach is usually applied. In the mixing plane approximation, the computed flow at the end of each blade row is circumferentially averaged and used as inflow condition for the downstream blade row. Similarly, the circumferentially averaged solution at the beginning of the flow passage is used as an outflow boundary condition for the upstream blade row. This procedure provides the coupling of individual blade rows and allows application of a steady-state calculations, since the boundary conditions for each row are axisymmetric. There are different realisations of mixing plane approach, and some of them can be found in [23-24].

The increased capacities of modern computers allow applications of LES (Large Eddy Simulation) and DES (Detached Eddy Simulation) techniques in real applications [25]. Such calculations basically belong to the area of unsteady simulations, but up to now they could be applied to single GT components only, and do not cover the problem of stator/rotor interaction.

The unsteady calculation of multistage turbomachine can be performed using sliding mesh approach. In this case the information on the interface between stationary and moving rows is transferred without circumferential averaging and all unsteady effects mentioned above can be captured. However, in the case of mixing plane it is sufficient to consider only one passage in each blade row, since the flow here is periodic. In contrary, the unsteady simulations should be performed on a periodic sector of geometry, but this is often impossible, since each blade row can have a different number of blades that do not have a common denominator. This is done to avoid instabilities caused by resonance between two components. Therefore in practical applications it is necessary to calculate the whole wheel, which increases the mesh size by two orders of magnitude, and the computation time more than two orders of magnitude. As the result the accurate unsteady calculations become impractical.

To reduce the requirements of computational resources several simplified procedures have been developed. One approach is referred to as “Chorochronic Periodicity Method” (also known as “Phase lag” method) has first been developed by Erdos and Alzner [26]. It assumes periodicity of the flow in time and stores the flow solution at pitchwise boundaries for application as boundary conditions during the following period. Adaption of this method to storing Fourier coefficients has been performed by He [27]. This method still requires storage of large amount of information, and does not capture several non linear effects.

A second approach, which does not assume temporal periodicity, is the “Time Inclination Method” presented by Giles [28,29], where the computational time is inclined by a transformation of the governing equations. This approach is used in analyzing rotor-stator interaction, but it is not applicable to multi-stage configuration. Both above approaches are not available in commercial CFD packages.

Another simplified approach is to scale adjacent blade rows such that spatial periodicity in the model is achieved. As such, it is rather the model geometry than the numerical treatment of the flow equations that is affected. The advantages are that no special treatment in time is assumed, and it allows performing analysis with any code capable of handling standard periodic boundary conditions. The disadvantage is that the model geometry is modified, and the method has its limitations in terms of maximum tolerable amount of scaling.

The sliding mesh calculations and simplified procedures are extensively used in research works, which examples were given above, but there are practically no reports in open literature about application of unsteady analysis in design of real size turbomachines. The latter can be explained by still too high computational efforts required for such analysis. Therefore further development of engineering procedures capturing unsteady effects of interest, and at the same time allowing calculations with moderate efforts is still feasible.

One simplified approach of this kind is discussed in this paper. This approach is limited to interaction of jet array injected from the stator with rotor. In GT this type of flow is often realised in secondary air systems, and in cooled rotating parts. The advantage of this procedure is faster convergence and reduced requirements to the mesh size in comparison with sliding mesh method, and possibility to use standard commercial codes for calculations. The disadvantage is the limited area of applications, although as shown below the procedure fits to the purpose.

NOMENCLATURE

C_f – friction coefficient
C_p – specific heat
D – hole diameter
D_h – hydraulic diameter
k – turbulence kinetic energy
k_s – sand roughness
P – static pressure, Ptot – total pressure
Re – Reynolds number
T – static temperature, Ttot – total temperature
t – time
U – velocity
x, y,z – Cartesian coordinates
x,r,φ – cylindrical coordinates
α - swirl angle
ε – turbulence dissipation rate
ρ – density
τ – radial angle
μ– laminar viscosity
μ_t – turbulent viscosity
Ω- rotational speed

ω - frequency

Subscripts
av –averaged value
J – maximal value in jet
max – maximal value in main flow
norm – normalised value
rel – value in rotating reference frame

NUMERICAL PROCEDURE

Area of application

Inside the turbine one can find different examples of the stator jet array interaction with rotor components, and some of them are presented in Fig.1.

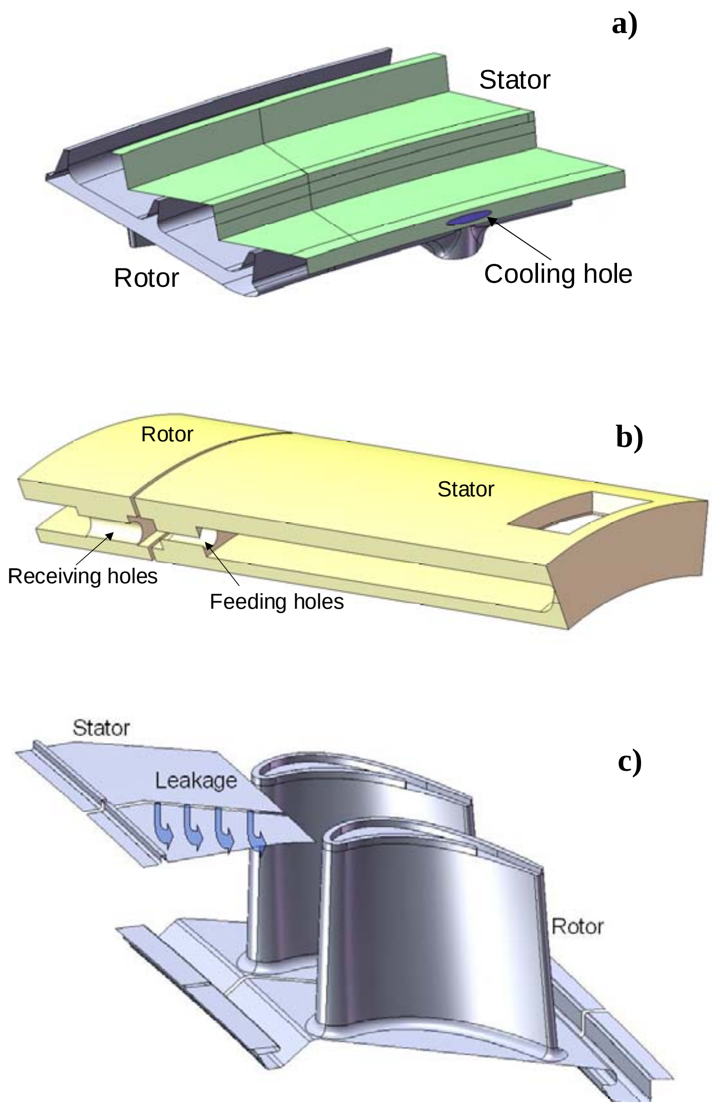


Fig.1 Application examples (a- passive shroud cooling, b – stator-rotor air transfer system, c – leakage in front of rotor

The air injected from the stator components is often used for the passive cooling of the blade tip and/or shroud. In Fig.1a one example of shroud passive cooling is shown. Here the cooling air blown from the orifices onto the stator mixes with the hot gas, reduces its temperature and protects the rotor blade. The cooling effectiveness depends on different factors, among which the unsteady effects play an important role, as will be demonstrated in the last section of this paper.

Another example is the air transfer from the stator to the rotor schematically shown in Fig.1b. The cooling air through the system of orifices is injected into the cavity between stator and rotor and then enters the rotor through another system of orifices. It is necessary to note that the presented scheme is a simplification of realistic air transfer systems, and often they have more complicated feeding and receiving parts. Nevertheless many of them are based on the same design principles, and therefore can be described within the scheme of stator jet array interaction with rotor.

The leakages from the stator (Fig. 1c) and their impact on the rotor can also be modelled by the array of jets injected from the axial slots. There are more examples of flows, which can be reduced to the scheme considered.

One block approach

To account for stator/rotor interaction in above cases the calculations should be performed using moving and non-moving grid blocks with appropriate coupling algorithms. The example of a two block CFD model corresponding to the air transfer system from Fig.1b is presented in Fig.2a. (In Fig.2a one sees the flow path and in Fig. 1b the metal parts, which create this flow path). The picture shows only part of the system and the whole arrangement can be quite complicated and require big size meshes.

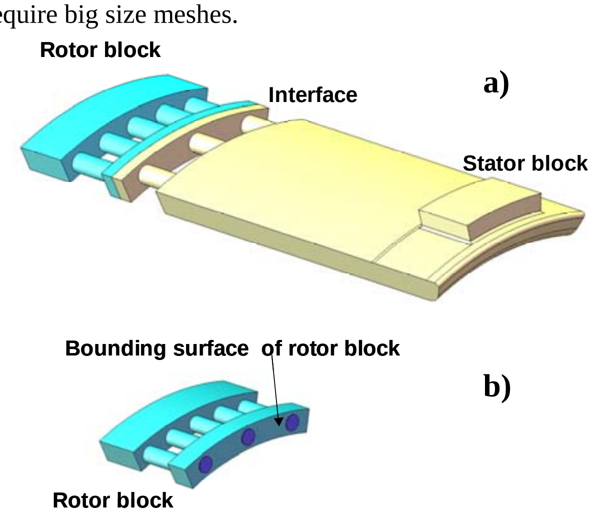


Fig.2 Two block (a) and one block (b) CFD models of air transfer system

On the other hand the flow inside stator holes is often close to fully developed turbulent flow. Its structure is not much affected by the rotor, and only the mass flow is changing according to the variations of pressure inside the rotor/stator cavity. Therefore the 3D calculations of stator part in many cases are not really necessary, and the injection can be modelled by appropriate boundary conditions applied to the rotor block. The one block arrangement is shown in Fig.2b, and this picture illustrates the important feature of simplified calculation procedure – replacement of the stator block by the bounding surface of the rotor block.

The part of bounding surface outside the ring containing feeding orifices is the real stator wall. The remaining part with the orifices is treated as the effective inlet, i.e. on this part of the surface the inflow conditions are prescribed. These two parts are indicated in Fig. 3.

One simplified version of such a model is already known in engineering practice. In that model the orifices are replaced by slot to ensure axisymmetric inflow conditions, and therefore the problem is reduced to a steady one in the rotating frame of reference. However in the oversimplified model not only unsteady effects are excluded, but it also cannot simultaneously maintain the correct injected mass flow and jet penetration, which significantly limit the usefulness of these calculations.

In our case the discrete structure of injection is preserved, and the only missing part is the simulation of the wall boundary layer on the stator wall between the orifices.

Inflow conditions

The definition of parameters on the effective inlet depends on type of injection and shape of injection holes. As an example the injection from elliptic holes is described here. This case is used below for verification purposes and for demonstration of method capabilities. Similar relationships can be derived for any other injections, e.g. for rectangular orifices shown in Fig. 1c.

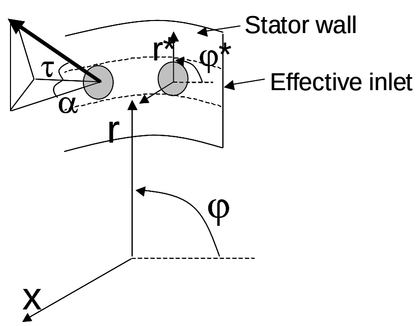


Fig.3 Definition of unsteady inflow conditions

In the stationary frame of reference the inflow parameters of each jet are defined based on correlations [30] for the round pipes. Basically these correlations are derived for incompressible flow, and therefore define the distribution of velocities. In many practical applications the flow is compressible, and to account for compressibility the mass fluxes should be prescribed instead of velocities, but as far as the flow is subsonic the shape of these distributions can be preserved the same as in [30]. Therefore for the distribution of mass flux at the outlet of round hole one has the relationship

$$(\rho U) = (\rho U)_J \left(\frac{2r^* - D}{D} \right)^{\frac{1}{n}} \tag{1}$$

where r* – local radial coordinate indicated in Fig.3. The parameter n in many practical cases can be defined as n=7, and for more accurate definition the following formula can be used

$$\frac{1}{n} = \sqrt{C_f} \tag{2}$$

Here Cf is friction coefficient, which can be calculated using relationships (6). For elliptical orifices the mass flux distribution is defined as

$$(\rho U) = (\rho U)_J \left(\frac{2r^* - D(\varphi^*)}{D(\varphi^*)} \right)^{\frac{1}{n}} \tag{3}$$

where D(φ∗) is the local width of the orifice. The mass flux components in cylindrical coordinates (x,r,φ) (Fig. 3) are calculated according to

$$(\rho U)_x = \frac{(\rho U)}{\sqrt{1 + \tan^2\alpha + \tan^2\tau}} \quad (\rho U)_r = \frac{(\rho U)\cdot\tan\tau}{\sqrt{1 + \tan^2\alpha + \tan^2\tau}}$$

$$(\rho U)_\varphi = \frac{(\rho U)\cdot\tan\alpha}{\sqrt{1 + \tan^2\alpha + \tan^2\tau}} \tag{4}$$

where swirl angle α and slope angle τ are defined based on the holes geometry.

The turbulence characteristic are varying across the pipe, but taking into account that the model is simplified it is sufficient to prescribe at the jet inlet the maximal values taken from the correlations [30]

$$\frac{\mu_t}{\mu} = 0.035 \cdot Re_D (\rho U)_J \sqrt{\frac{C_f}{8}} \ , \quad k = U^2{}_J \frac{C_f}{8} \tag{5}$$

where Re_D is Reynolds number based on pipe diameter. and friction coefficient C_f is calculated from

$$\sqrt{\frac{8}{C_f}} = 2.44 \cdot \ln(\sqrt{\frac{C_f}{32}}\, Re_D) + 2 \, , \sqrt{\frac{8}{C_f}} = 2.44 \cdot \ln(\frac{D}{2k_s}) + 4.9 \tag{6}$$

The first relationship is for the smooth and second for the rough pipes, k_s – is sand roughness.

The total temperature can be assumed constant in many practical cases. However if the heat transfer in stator should be accounted, it should be analysed separately and appropriate values of temperature can be derived from this separate analysis. In all cases considered in this work we used

Ttot=constant (7)

On the part of the effective inlet between jets the mass flux is assumed zero (ρU)=0, and the turbulence characteristics and temperature are assumed equal to the values in the neighbouring jets. The latter simplifies the calculations, but provides inaccuracy in the near wall region.

The relationships (1-7) define all required parameters at the mass flow inlet in the stationary frame of reference as functions $f(x, r, \varphi)$. To account the relative movement of rotor and stator the following transformation should be applied

$$f(x,\ r, \varphi\cdot) \rightarrow f(x,\ r,\ \ \varphi - \omega \cdot t) \tag{8}$$

If rotor and stator have the same periodicity then ω=NΩ , where N is number of periodic pieces. In this case one periodic piece for rotor should be calculated and the boundary conditions (8) are derived from one stator piece. If the rotor and stator have the common denominator and *m* stator pieces correspond to one rotor piece than again the actual computational domain includes only one rotor passage, ω=NΩ, but the boundary condition (8) should include *m* periodic stator pieces. If *m* rotor pieces correspond to one stator piece than the computational domain should include *m* rotor passage, and ω=(N/m)Ω.

If no common denominator is available then the calculations should be done for the whole rotating wheel, and in this case parameter ω in (8) equals $\omega=\Omega$. Chorochronic periodicity method can also be applied for reduction of computational resources, and a big advantage here is that it does not require large memory as far as the rotor/stator interface is defined analytically. However realisation of the last approach within commercial code requires significant effort, and in present work we have considered only cases with common periodicity, where the application of commercial code is straightforward.

The velocity components and total temperature on the effective inlet in stationary and rotating frames of reference are related by equations

$$U_{x\ rel} = U_x\ ,\ U_{r\ rel} = U_r\ ,\ U_{\varphi rel} = U_\varphi - \Omega r,\ T_{tot\ rel} = T_{tot} + \frac{(\Omega r)^2 - 2U_\varphi \Omega r}{2C_p} \quad (9)$$

where Cp is specific heat, which can be assumed constant. However using commercial solver it is not necessary to include this transformation explicitly in relationships (8), because it is done automatically inside the code. The same remark is valid for turbulent kinetic energy, which can be prescribe as turbulence intensity (k/U^2_J) (relationship (5)) and then is automatically recalculated inside the code.

The inflow conditions (1-8) define the effective inlet as mass flow inlet. In many practical situations this is acceptable. However the mass flux is related to pressure and temperature as (see e.g. [31])

$$(\rho U) = F(P_{tot}, P_{tot}/P, T_{tot}) \quad (10)$$

If the time independent value of mass flux is prescribed and the calculated static pressures at the inlet is also time independent the resulting total pressure will be constant as well. However if the injection plane is affected by passing bow waves, the static pressure is time dependent and according to (10) the feeding total pressure will also oscillate, which is not physical. This happens when the distance between injection plane and moving object is below 2.5d (d – characteristic size of the obstacle). Using commercial code the problem cannot be solved by defining the effective inlet as a pressure inlet, because the pressure should be prescribed explicitly, but it is not known beforehand. Therefore to get the physically feasible total pressure behaviour the correction procedure should be applied. After calculation of flow with constant mass flow the time dependent static pressure (P(t)) and time averaged total pressure (Ptot av) are extracted from the analysis. The time dependent mass flow is calculated from formula (10) using P(t), and Ptot av , and applied as the new inflow condition (i.e. $(\rho U)_J$ in (1-8) is replaced by the time dependent value). In the corrected solution the variation of Ptot is reduced to a minimum, and if more accuracy is required the correction can be repeated.

Solver

The approach outlined above allows analysis with any code aimed at calculations of unsteady periodic flows. In this work we have used commercial code Fluent. The pressure based solver, and k-ε model were activated. The boundary conditions in form (1-8) were programmed using internal compiler (so called user defined functions). As far as Fluent does not allow zero (ρU) value at the mass flow inlet, the small value $(10^{-4} \cdot (\rho U)_J)$ was prescribed on the effective inlet in the area between the jets.

Verification

To verify the unsteady inflow procedure the simplified geometry of passively cooled blade shroud has been considered. The shroud geometry shown in Fig.1a is similar although not fully identical to one investigated in [32-33]. In the simplified model used for verification only the airfoil is not included and the shroud cavity is axisymmetric. The cooling air is injected into the cavity from the stator through the set of axial elliptical holes, where in the numerical model a sector with one hole is simulated. Due to an axisymmetric rotor part the flow in the simplified geometry is steady-state in the stationary frame of reference, and therefore the steady state calculations provide the reference case.

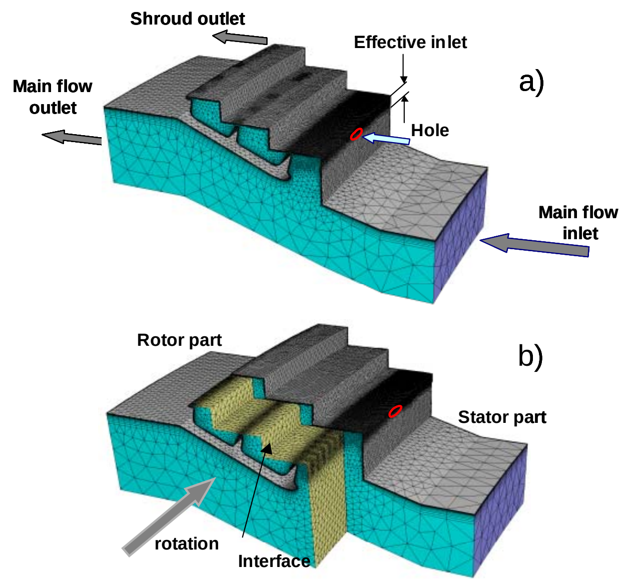


Fig.4 Reduced CFD model - blade shroud without airfoil (a - one block mesh, b - two block mesh with interface)

Three calculations have been performed: the reference case (i.e. steady), unsteady inflow, and sliding mesh calculations. The geometry in all cases is the same and the boundary conditions in the reference case and in the sliding mesh calculations are the same as well (in all 3 cases the mass flow of cooling air is prescribed). The reference case and unsteady inflow are calculated on the same mesh shown in Fig.4a, which allows comparison of the simplified approach with the accurate numerical solution in comparable conditions.

The sliding mesh simulations (the two block sliding mesh model shown in Fig. 4b) did not include the flow inside the stator block and therefore should provide the same results as the reference case. These calculations were done just for comparison of the computational efforts.

The comparison of gas temperature on the radial surface (radius equals to position of the injection hole centre), calculated with different methods (stationary reference frame, sliding mesh and simplified procedure), is presented in Fig.5

(Here normalised values of temperature Ttot $_{rel\ norm}$=(Ttot $_{rel}$ – (T tot $_{rel\ J}$)/(Ttot $_{rel\ max}$ –Ttot $_{rel\ J}$) are plotted.)

There is practically no difference between calculations in stationary frame and with sliding mesh. The solution of simplified method deviates from the accurate one near the effective inlet.

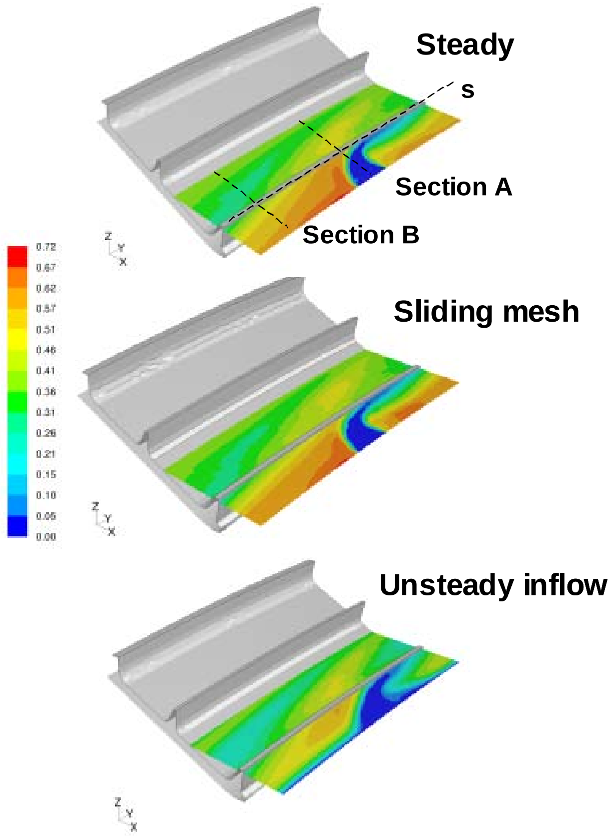


Fig.5 Normalised relative total temperature distribution on radial surface

As mentioned above the meshes in all cases have the same size, but the unsteady calculations require more time. In simplified method the calculation time by factor 5 higher than in steady case, and sliding mesh calculations last 20 times longer than the steady. This example clearly shows the computational effectiveness of the simplified method in comparison with standard sliding mesh approach. This difference is caused by 4 times more iterations on each time step in the sliding mesh calculations, and the latter is the result of the interface location inside the strong shear flow. The difference would be even more pronounced if the stator block would be included into the sliding mesh analysis.

The difference between accurate and simplified approaches is presented in details in Fig.6, where the normalised tangential velocity, normalised relative total pressure and temperature distributions along the axial direction are plotted. The positions of sections A&B are indicated in Fig.5, where section A is located at the hole centre, and section B is located between holes at 20% of shroud width measured from its edge. The axial distance is normalised to the hydraulic diameter of the hole (x $_{norm}$=x/D$_h$), tangential velocity to linear rotational speed at this radius (U$_{\varphi\ norm}$=U$_\varphi$/Ωr), total pressure to the jet total pressure (Ptot $_{rel\ norm}$=Ptot $_{rel}$/Ptot $_{rel\ J}$), and temperature is normalised in the same way as in Fig.5. In Fig. 7 the distributions of normalised static pressure and temperature in tangential direction on the front surface of shroud are shown. The normalisation is the same as above, and direction s (s $_{norm}$=s/D$_h$) is indicated in Fig. 5.

These comparisons show that the simplified solution is inaccurate near the effective inlet, where it replaces actual stator wall. However the discrepancy decays fast and at the mid position between injection plane and shroud both solutions are similar. The temperature distribution shows biggest discrepancy near the effective inlet (the prescribed temperature on the effective inlet equals to the coolant temperature), and other parameters are in reasonable agreement in the whole calculation domain.

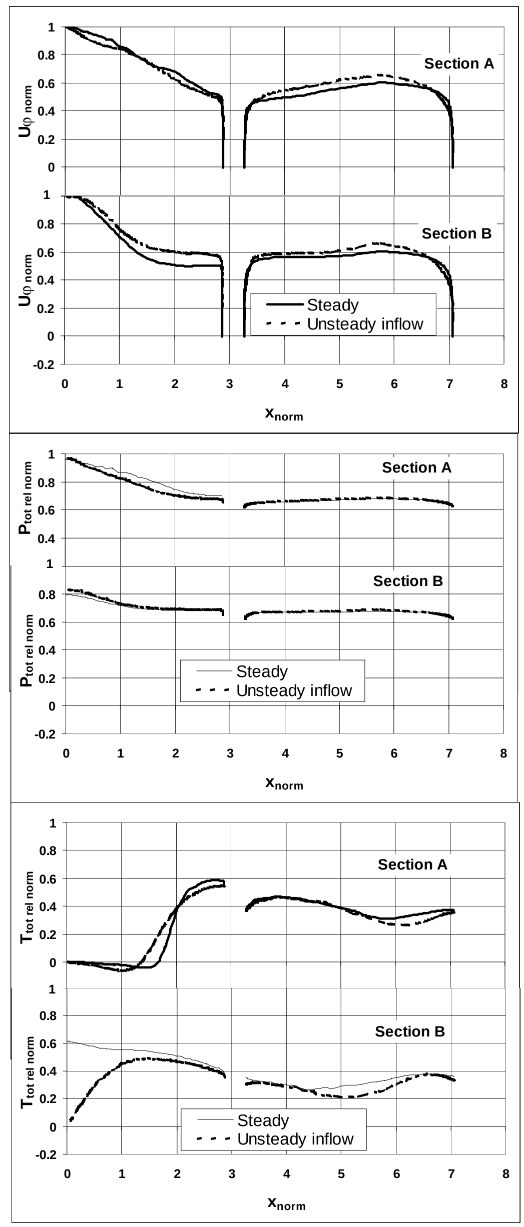


Fig. 6 Comparison of tangential velocity, relative total pressure and temperature in two sections (positions in Fig. 5)

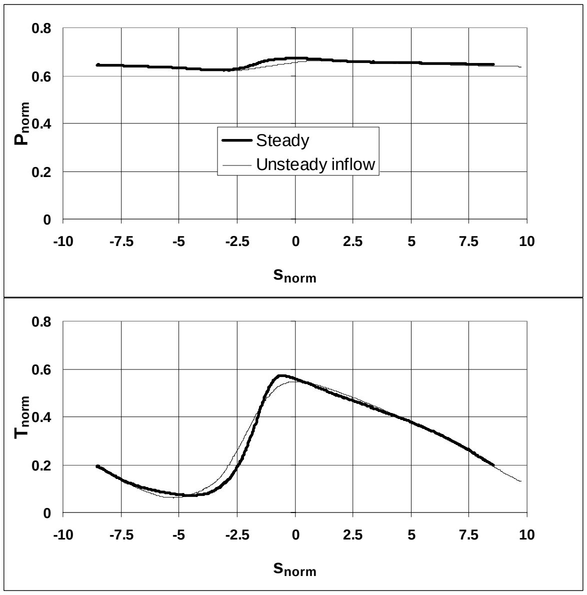
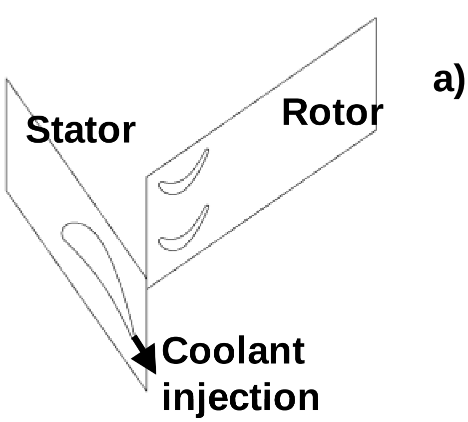


Fig. 7 Comparison of normalised static pressure and temperature along the shroud front surface

Thus, the simplified approach demonstrates reasonable accuracy and its computational efficiency justifies its use at least in engineering applications related to passive cooling of the shroud. In such applications the stator thermal state is less important than rotor temperature and analysis considered allows evaluation of this temperature with the same accuracy as the more complicated approaches.

As another verification example the cooling air injection from the trailing edge of the stator has been considered. The scheme of the flow is shown in Fig. 8a. This is 2D flow in stator and rotor cascades, where the cascade geometry corresponds to geometry discussed in [21]. The cooling air is injected from the trailing edge of the stator. Strictly speaking the procedure for jet array described by relationships (1-7) is not directly applicable here. However in this case the problem can also be reduced to one rotating block with the effective inlet, and therefore this example allows verification of the main feature of the simplified method.

It is necessary to note that similar analysis as in this verification example has been executed by Ameri et.al. (see e.g. [14]). However their approach is limited to the cascade geometry, and in the current work it is expended on the jet array as well.



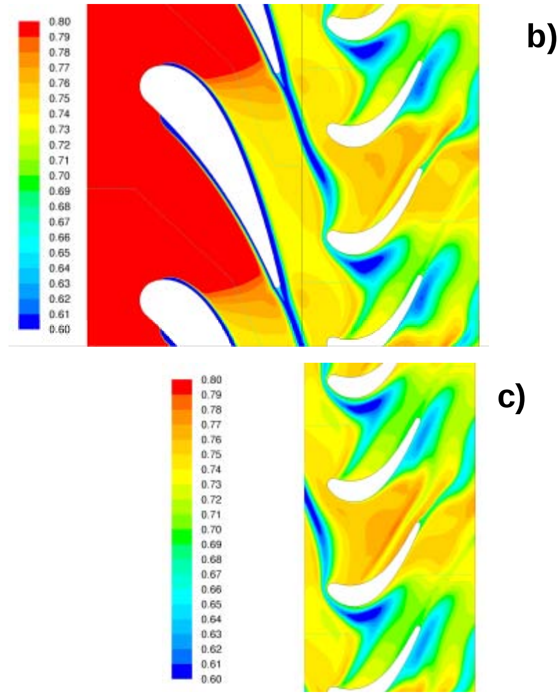


Fig. 8 Coolant injection from stator cascade
a) Flow scheme; b) Normalised total temperature distribution - sliding mesh solution; c) Normalised total temperature distribution –unsteady inflow

The effective inlet was defined here as the velocity inlet and the spatial distribution of parameters in it (velocity components, total temperature, turbulence) were taken from the steady calculation of stand-alone stator cascade. At the inlet of rotating block they were transferred using relationship (8). It is necessary to note that this procedure is working as far as the flow on the interface is subsonic (as in case considered), and it is aimed to evaluate the temperature field created by jets. The vortex shedding cannot be captured in this method. The comparison of temperature calculations using sliding mesh and simplified procedures is presented in Fig.8b,c. The results are in reasonable agreement, but simplified method allowed calculation time reduction by factor 5 (2 times due to the grid reduction and 2.5 times due to the reduction of iterations per step).

APPLICATION EXAMPLE

To demonstrate the opportunities, which the method provides in turbine analysis, the passive cooling of shroud using different holes pattern has been simulated. The blade and shroud geometry as shown in Fig.1a are similar to geometries investigated in [32-33]. In the application example not only shroud cavity, but the whole blade passage was modelled, and therefore this case is an example of 3D unsteady flow, which cannot be reduced to a steady one by any reference frame transformation.

The investigated different hole patterns are shown in Fig.9. In Case 1 there is one axial hole per blade (elliptic hole with axis length ratio 6), in Case 2 - 6 axial round holes, in Case 3 – one radial round hole above the shroud, and in Case 4 – 6 radial round holes. The effective inlets for axial holes and radial holes are indicated in Fig.9 as well (for axial holes it is the same as shown in Fig. 4a).

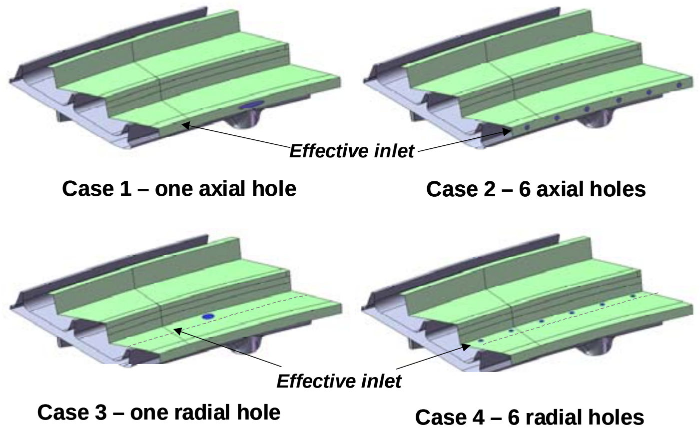


Fig.9 Hole patterns

The results of temperature calculations for cases 1 and 2 are presented in Fig. 10-12. As far as the shroud of blade is not cooled the metal temperature distribution is similar to gas temperature, and therefore the pictures also indicate the thermal state of blade surface. The mass flow of coolant is the same in both cases, but single jet has higher penetration, and therefore provides lower temperature on the shroud, especially on it front part. In addition to this single jet partially prevents hot gas penetration from the main flow to the shroud cavity. In contrary the multi jet injection creates vortex structure in front of shroud, which sucks the hot gas from the main flow. This feature is illustrated in Fig.11, where the flow structures in the cavity between stator and rotor are compared (in case 1 the cut through the maximal vortex intencity is shown). In addition to cases 1 and 2 the injection from the slot was analysed as well, and these results are also shown in Fig.10-12. One can see that 6-hole configuration provides results similar to slot and therefore further increase of number of holes only slightly influences temperature distribution.

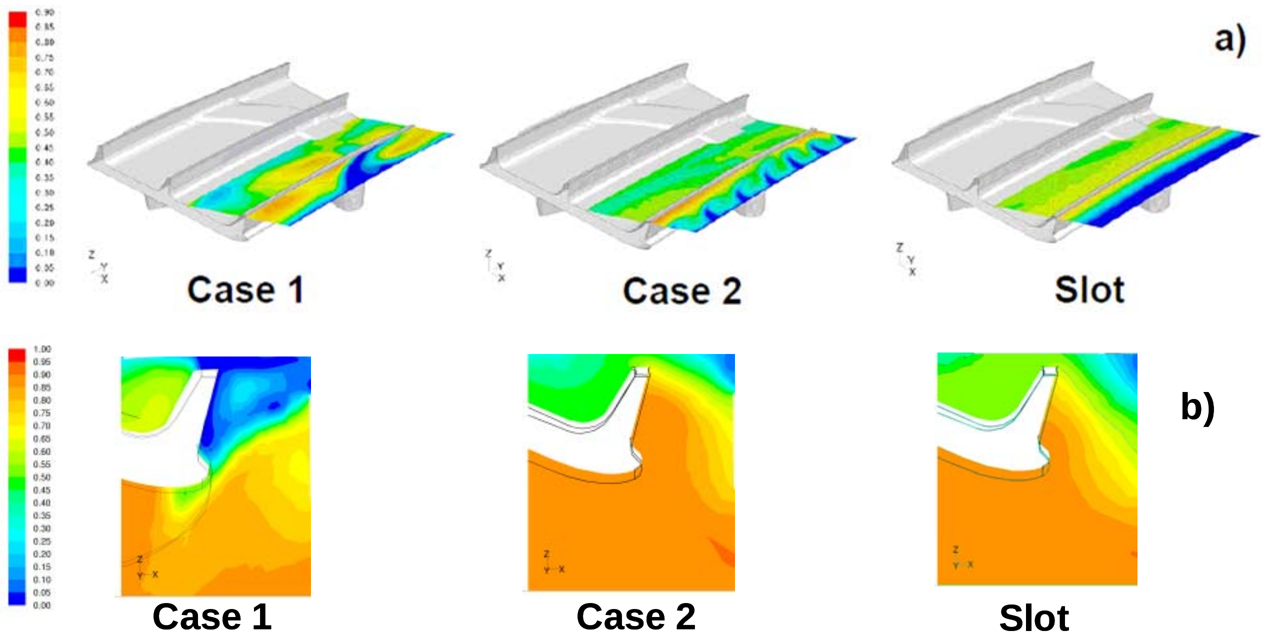


Fig.10 Normalised relative total temperature a) on radial cut through the hole axis; b) axial cut through the hole axis

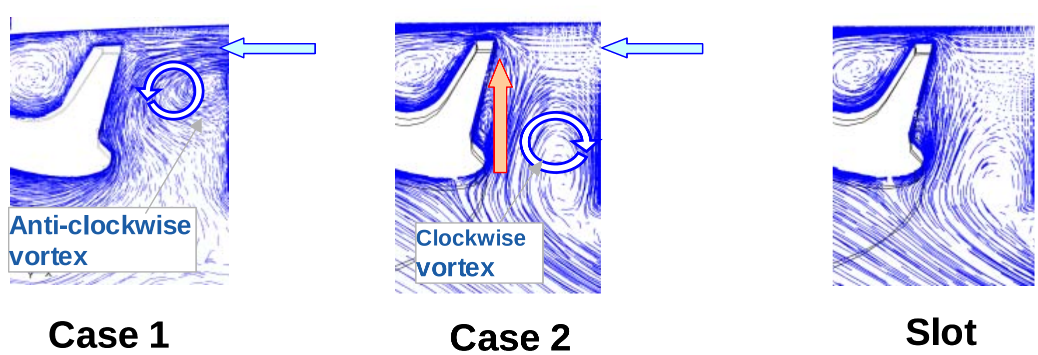


Fig.11 Flow structure in front of shroud (axial cut through the hole axis)

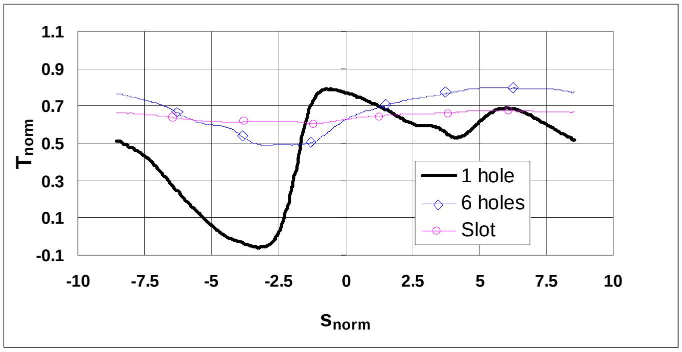


Fig.12 Normalised relative temperature distribution on the shroud front surface (time averaged value)

On the other hand reduction of cooling holes can reduce the averaged shroud temperature. This is not an obvious observation and it is a result of the proper unsteady analysis. Of course in real design it is necessary to take into account different factors, not only average surface temperature, but also its gradients, distribution inside metal, and other features. The proper choice of cooling configuration is multidisciplinary task, which is outside the scope of this paper. Here we just want to emphasise that part of this task should be unsteady calculations accounting stator jet array interaction with rotor.

To demonstrate that it should really be unsteady calculations the above results are compared with the calculations done by so called frozen hole method. In this case the position of stator holes is fixed with respect to the rotor, although the difference in rotational speed is accounted for. This method is physically not correct, but sometimes is used in engineering applications for rough estimations. (More known variant of such approach is so called frozen rotor method).

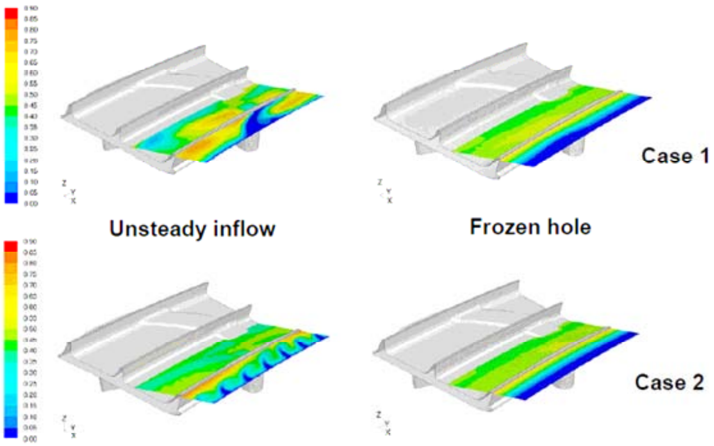


Fig.13 Normalised relative total temperature distribution on radial cut through the hole axis

Comparison of unsteady and steady simulations is presented in Fig.13. This comparison shows that for the 6 holes steady method gives values of temperature in front of the shroud comparable with unsteady data, as far as this configuration is already close to the case of injection from slot. For one hole the frozen hole approach provides wrong result, and even predicts lower jet penetration as in 6-hole configuration. Thus the impact of a cooling holes pitch on the shroud surface thermal state predicted by steady analysis has an opposite trend compared to a realistic unsteady analysis.

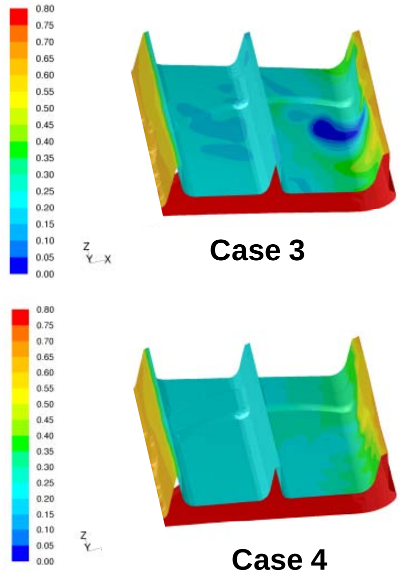


Fig.14 Normalised relative temperature distribution on the shroud surface (time averaged value)

Similar results were obtained for cases 3 and 4, where the radial holes are aimed at temperature reduction in the cavity between two shroud fins. Again the one hole configuration allows lower surface temperature than the 6 holes, as it is illustrated in Fig. 14.

CONCLUDING REMARKS

For simulation of stator jet array interaction with rotating components the simplified numerical procedure is proposed. The calculations are performed using single block mesh for the rotor, and the injection from stator is simulated by effective inlet, where the boundary conditions are unsteady and reproduce the movement of stator holes in the relative frame of reference.

The simplified calculations are in reasonable agreement with the accurate numerical solution, and at the same time significantly reduce the calculation effort. Due to faster convergence simplified procedure reduces the computational time by factor 4 in comparison with sliding mesh approach on the same size mesh, and provides additional advantage in cases where the calculation domain can be reduced from two blocks to single block model. The approach is compatible with any commercial solver capable to handle periodic unsteady flows.

Although it was developed primarily for simulation of the jet array injecting from the stator, there is a possibility to extend it to some other types of flows as demonstrated by example of cooling air injection from the trailing edge.

The limitation on the periodicity of geometry is the same as in sliding mesh approach, and the same procedures allowing overcoming these limitations can be applied (e,g, phase lag, etc.)

The application example of blade shroud passive cooling demonstrates that unsteady procedure allows identification of interesting trends, which can not be captured by a simplified procedure like frozen hole. Therefore the procedure considered provides a useful extension to the state of the art turbine design and analysis tools.

REFERENCES

[1] Dean, 1959, “On the Necessity of Unsteady Flow in Fluid Machines” ASME Journal of Basic Engineering, Vol. 81, pp. 24-28.

[2] Meyer, R. X., 1958, "The Effect of Wakes on the Transient Pressure and Velocity Distributions in Turbomachines, " ASME J. Basic Engineering, 80, pp 1544-1552.

[3] Hodson, H.P., 1985, “Measurements of Wake-Generated Unsteadiness in the Rotor Passages of Axial Flow Turbines”, ASME Journal of Engineering for Gas Turbines and Power, Vol.107 April 1985, pp467- 476.

[4] Hodson, H.P.; Dawes, W.N., 1998, ”On the interpretation of measured profile losses in unsteady wake-turbine blade interaction studies”, Journal of Turbomachinery, Vol.120, April 1998, pp276-284

[5] Wu, X., Jacobs, R.G., Hunt, J.C.R., Durbin, P.A., 1999, “Simulation of boundary layer transition induced by periodically passing wakes”, J. Fluid Mech., Vol. 398, pp.109-153

[6] Hodson, H.P., and Howell, R.J., 2005, “Bladerow Interactions, Transition, and High-Lift Aerofoils in Low-Pressure Turbine,” Annual Review of Fluid Mechanics, 37, pp. 71-98

[7] Bons, J.P., Pluim, J., Gompertz, K., Bloxham, M., and Clark, J.P., 2008, “The Application of Flow Control to an Aft-Loaded Low Pressure Turbine Cascade with Unsteady Wakes”, ASME Paper, GT2008-50864.

[8] Volino, R. J., 2010, “Effect of Unsteady Wakes on Boundary Layer Separation on a Very High Lift Low Pressure Turbine Airfoil”, ASME- paper, GT2010-23573,

[9] Sharma, O. P., Renaud, T. L., Butler, T. L., and Milsaps, K., 1988, "Rotor–Stator Interaction in Multi-Stage Axial-Flow Turbines," AIAA Paper No. 88-3013.

[10] Chaluvadi, V.S.P., Khalfas, A.I., Hodson, H.P., Ohyama, H., and Watanabe, E., 2003, “Blade Row Interaction in a High-Pressure Steam Turbine,” ASME Jnl. of Turbomachinery, Vol. 125, pp. 14-24.

[11] Pullan, G., and Denton, J.D., 2003, “Numerical Simulations of Vortex-Turbine Blade Interaction,” 5th European Turbomachinery Conference, pp. 1049-1059.

[12] Hummel, F., 2001, “Wake-Wake Interactions and its Potential for Clocking in a Transonic High Pressure Turbine”, ASME –paper, 2001-GT-0302

[13] Hodson, H. P., Hynes, T. P., Greitzer, E. M., Tan, C. S., 2009, “A Physical Interpretation of Stagnation Pressure and Enthalpy Changes in Unsteady Flow”, ASME-paper GT2009-59374

[14] Ameri, A.A., Rigby ,D.L., Steinthorsson, E., Heidmann, J., and Fabian, J.C., 2008,“Unsteady Analysis of Blade and Tip Heat Transfer as Influenced by the Upstream Momentum and Thermal Wakes,” ASME Paper, GT2008-51242

[15] Prasad, D., Hendricks, G., 2000, “A Numerical Study of Secondary Flow in Axial Turbines with Application to Radial Transport of Hot Streaks,” ASME Journal of Turbomachinery, Vol. 122(4), pp. 667–673.

[16] Ong, J., Miller, R.J., 2008, “Hot Streak and Vane Coolant Migration in a Downstream Rotor”, ASME-paper, GT2008-50971

[17] Day, I. J., and Freeman, C., 1994, “The Unstable Behavior of Low and High-Speed Compressors”, ASME J. Turbomach., 116, pp. 194-201.

[18] Vahdati, M., Simpson, G., and Imregun, M., 2008, “Unsteady Flow and Aeroelasticity Behavior of Aeroengine Core Compressors During Rotating Stall and Surge,” ASME J. Turbomach., 130, 031017.

[19] Jennions, I., Sommer, T., Vassiliev, V., 1998, “A Numerical Study of Air Transfer from a Stationary to a Rotating System through an Unsealed Cavity”, ASME Paper, 98-GT-139

[20] Vassiliev, V., Pirscher, A., Kostege, V., Chernyshev, S., 2005, “CFD application in design of GT structural components”, ASME Paper, GT2005- 68950

[21] Vassiliev, V., Mooslechner, N., Kostege, M., Granovskiy, A., 2007, “Aerodynamic characteristics of a redesigned turbine”, ASME Paper, GT2007- 27472

[22] Vassiliev, V., Rothbrust, M., Irmisch, S., 2008, “Refitting of exhaust diffuser of industrial gas turbine”, ASME paper, GT-2008- 50165

[23] Denton, J.D., 1990, ”The Calculation of Three Dimensional Viscous Flow Through Multistage Turbomachines”, ASME Paper, 90-GT-19

[24] Irmisch, S., et al., 1999, "Efficient Use of Hybrid Grids in Modern Turbomachinery Applications", AIAA-99-0783, 37th Aerospace Sciences Meeting and Exhibit, January 11-14, 1999/Reno, Nv, USA

[25] Tucker, P., Eastwood, P.,S., Klostermeier, C., Xia, H., Ray, P., Tyacke, J., and Dawes, W., 2010, “Hybrid Les Approach for Practical Turbomachinery Flows: Part 2—Further Applications” ASME-paper, GT2010-23807

[26] Erdos, J. I., Alzner, E., 1977, “Numerical Solution of Periodic Transonic Flow through a Fan Stage”, AIAA Journal, 15, 1977, pp. 1559-1568

[27] He, L., 1992, “Method of Simulating Unsteady Turbomachinery Flows with Multiple Perturbations”, AIAA Journal, 30, 1992, pp. 2730-2735

28] Giles, M., 1988, “Calculation of Unsteady Wake/Rotor Interaction”, Journal of Propulsion and Power, 4(4), pp. 356–362

[20] Giles, M. B., 1990, “Stator/Rotor Interaction in a Transonic Turbine”, AIAA Journal of Propulsion and Power, 6(5), 1990, pp. 621-627

[30] Hinze, I.O., 1975, “Turbulence”, McGraw-Hill Inc., New York, 790 p.

[31] Abramovitch G.N., 1976, “Applied Gas dynamics”, Nauka Press, Moscow, 888 p.

[32] Lehmann, K., Thomas, R., Hodson, H., Stefanis, V., 2009, “Heat Transfer and Aerodynamics of over Shroud Leakage Flows in a High Pressure Turbine”, ASME-paper, GT2009-59531

[33] Dawes; W. N., Kellar, W. P., Harvey, S. A., “Towards Cooled Turbine Life Prediction via Concurrent Aerodynamic, Thermal & Material Stress Simulations on Conjugate Meshes”, ASME-paper, GT2010-22482