

Editorial

Excellence in Turbomachinery Research: The Best of the 12th European Turbomachinery Conference

Marcello Manna 

Dipartimento di Ingegneria Industriale, Università di Napoli Federico II, via Claudio 21, 80125 Naples, Italy; marcello.manna@unina.it; Tel: +39-081-7683287; Fax: +39-081-2394165

Received: 29 June 2018; Accepted: 29 June 2018; Published: 9 July 2018



Turbomachines are undoubtedly the key components of all power systems, and despite the impressive advances of the past century, there seems to still be room for improvement. While the reduction in the specific fuel consumption directly related to the components efficiency enhancement is the most natural and pursued objective, other aspects progressively acquire importance. Noise, life, reliability, and safety, to quote a few examples, require technological leaps that can only stem from solid, well-conceived, fundamental, and applied research. Those concerns are equally shared by the power generation world, the aerospace community, and the automotive one, all mainly focused on efficiency innovations.

This Special Issue appearing in the *International Journal of Turbomachinery, Propulsion, and Power* [1] (www.mdpi.com/journal/ijtp), collects original research articles addressing advances in computational and measurement techniques as applied to the above mentioned fields for the benefit of the academic and industrial research communities. These manuscripts originate from the 12th European Turbomachinery Conference (ETC12) held in Stockholm (Sweden) in March 2017 under the local coordination of the Swedish Royal Institute of Technology. The conference main objectives are the presentation of the latest developments in the turbomachinery field; the promotion of the transfer of this technology across Europe; and the creation of a forum suitable to disseminate and advertise the results of the research projects funded by the European Commission, as well the resultant benefits from the support by the Commission.

The ETC series, started in 1995, is managed and organized by *Euroturbo* (www.euroturbo.eu), the European Turbomachinery Society, a not-for-profit international scientific organization legally established in February 2012, specifically organized to enhance the dissemination, exchange, and publication of top quality research in the field of turbomachinery. Yet, the policy of the society has always been to seek papers documenting good quality research rather than simply increasing their numbers, so that striving for quality rather than quantity has always been the society's motto.

The research works appearing in this Special Issue were selected from a pool of two hundred full manuscripts submitted for review to the ETC12. They represent the outcome of a rigorous review process completed with the help of three independent referees supervised by a senior scientist, who rated them of journal quality.

Six of the papers appearing in this issue are of computational nature, two deal with experimental work, one is of theoretical imprinting, and one is of the mixed experimental–computational type. Two of them received financial support from the European Union, four from their National Research Agencies, and two from industrial partners. Turbine and combustor–turbine interaction are the most addressed topics, while interests in problems of fundamental nature are also available. Achievements are summarized next for the benefit of the interested readers, who can have free access to the whole Special Issue here: http://www.mdpi.com/journal/ijtp/special_issues/ETC12.

Frey et al. [2] developed the harmonic set technique, which generalizes the harmonic balance method to zero-frequency harmonics to resolve rotational a-periodicities of the time-averaged flow

in turbomachinery, making the simultaneous simulation of the zero-frequency and of the unsteady disturbances possible. While their approach neglects the nonlinear interactions between different harmonic sets, it does capture correctly the nonlinear coupling terms within one harmonic set, allowing harmonic balance simulations of configurations, including two stators or two rotors with different blade counts, to be performed using only one passage per blade row. Computational costs can, therefore, be considerably reduced. In their paper, the authors demonstrated that the approach can also be used to accurately predict the indexing effect of a fan inlet distortion on the mean flow in a fan stage. Comparisons with time-domain and harmonic balance simulations on full annulus configurations have shown that the time-averaged flow in the stator row, as well as the decrease in isentropic efficiency and increase in total pressure ratio, are accurately predicted.

Ciorciari et al. [3] computationally investigated the secondary flow and the loss development in the T106Div-EIZ low-pressure turbine cascade with divergent endwalls utilizing (unsteady) Reynolds-averaged Navier–Stokes ((U)RANS) simulations in cases with and without periodically incoming wakes at $Ma_{2th} = 0.59$ and $Re_{2th} = 2 \times 10^5$. The predictions are compared with experimental data by the same research group. Comparing the T106Div-EIZ against the T106A-EIZ with parallel endwalls under the same exit flow conditions, it turns out that the former produces more intense secondary flows in the passage. This is caused not only by the higher front-loading, but also by the endwall geometry, which influences the roll-up of the incoming endwall boundary layer and the axial loss development in the front part of the cascade. The overall loss development indicates that the increased values in the case of divergent endwalls are mainly related to the endwall flow. A comparison of the steady and unsteady cases shows a redistribution of the loss generation components because of the incoming bar wakes. In the front-loaded T106Div-EIZ, the incoming wakes cause a premature endwall loss production in the front part of the passage, resulting in a lower intensity of the secondary flow downstream of the trailing edge and smaller secondary mixing losses. Nevertheless, in the investigated cases, the profile losses are augmented by an earlier transition/separation and a longer reattached turbulent boundary layer because of the incoming wakes, causing an increased overall entropy generation with respect to the undisturbed case.

Gaetani and Persico [4] presented the results of an experimental study on the evolution of hot streaks generated by gas turbine burners in an uncooled high-pressure turbine stage. The prescribed hot streaks were directed streamwise and characterized by a 20% over-temperature with respect to the main flow at the stage inlet. The hot streaks were injected in different clocking positions with respect to the stator blade and provided a total temperature perturbation representative of aero-engine conditions. Several measurement techniques were applied to investigate the hot streak impact on the thermal and aerodynamic behavior of the stage. It has been shown that throughout the convection within the stator channel, the hot streak undergoes different migration and attenuation depending on the injection position. In particular, the clocking of the hot streak with the stator blade leading edge induces a dramatic deformation of the hot streak, which takes the shape of the blade wake. The temperature attenuation across the stator is severe for all cases, and the maximum temperature ratio drops down from 1.2 to 1.05. With the exception of the injection on the suction side, a portion of the hot streak is entrained in the cross-flow induced by the stator pressure field on the shroud. A negligible increase in the stator total pressure loss is found as a result of the hot streak transport and evolution. When the stator blade thermal stress is of concern, the hot streak injection on the leading edge is the worst possible operating condition, as can be argued by the temperature field downstream of the stator and as was already pointed out by other authors. The lowest interaction with the blade and the highest temperature diffusion all over the channel occur when the hot streak is injected at mid pitch. On the contrary, downstream of the rotor, the highest diffusion is found for the injection at 1/3 of the pitch on the pressure side, with potentially positive implications on the following stage. The expected over-speed induced by the hot streak as a result of its higher enthalpy content at the stator exit is very limited, although expected, as documented by previous theoretical analysis, and has a minor impact on the rotor incidence angle. Conversely, an interesting effect is found on the vorticity field, which shows

additional contributions at the top and bottom of the hot streak (except for in the case of leading edge injection). Such additional vorticity cores also alter the rotor secondary flows as found at the rotor outlet. The temperature disturbance at the rotor outlet is further attenuated, consistently, with a significant spreading of the incoming rotor disturbances. These results are in line with the enhanced migration that the hot streak experiences in the rotating channel and trigger the interest for further experimental and computational studies specifically oriented to the unsteady hot streak migration in the rotor.

Suman et al. [5] investigated, from a numerical and experimental point of view, the performance of a cooling axial fan equipped with morphing blades, which can control itself to reduce or eliminate the need for active control systems. Morphing blades were obtained using shape memory alloys (SMA) strips embedded in a polymeric matrix. Exploiting the results of previous studies related to SMA strips and polymeric matrix characterization, and thermal activation by means of a wind tunnel and computational fluid dynamics (CFD) data of the cooling fan, several structural analyses were carried out in order to discover the effects of the aerodynamic and centrifugal loads on the blade shape. Structural analyses were carried out using a reconstructed blade shape obtained with an innovative three-dimensional blade surface capture system provided by the Kinect sensor. The latter was able to detect the blade shape changes during the activation tests without altering the thermal and flow wind tunnel conditions. The results highlighted that the aerodynamic and centrifugal loads induced by the cooling fan operation do not determine a dramatic variation of the airfoil shape. This demonstrates that the wind tunnel blade shape characterization is suitable for analyzing the SMA blade shape modification, and that the polymeric structure offers a proper resistance against these loads and allows variation of the blade shape according to the action of the SMA strips. A preliminary experimental characterization of the cooling fan was then conducted using a purpose-built test-rig, which allowed for the evaluation of the fan performance and the blade shape modification during thermal activation. The experimental data and numerical results are in good agreement both for the fan performance and the structural analysis.

Zauner et al. [6] extracted velocity profiles from time- and span-averaged direct numerical simulation data, describing the flow over a high-pressure turbine vane linear cascade near engine-scale conditions with reduced inlet disturbance levels. Based on these velocity profiles, local as well as non-local linear stability analysis of the boundary-layer over the suction side of the vane were carried out in order to characterize a linearly unstable region close to the trailing edge. The temporal linear stability analysis (LST) results are cross-checked with those obtained from a spatial LST approach. Applying the Gaster transform, good agreement is found between the two approaches (the difference between growth rates is about 3.3%), even for highly unstable regions. The linear parabolized stability equation (LPSE) for a representative frequency also shows good agreement with the LST results, indicating small non-parallel effects in the analyzed region. Finally, these unstable linear modes are shown to correspond to time scales found in direct numerical simulations (DNS) data close to the trailing edge (TE). As a result of an interaction between the vortex shedding at the TE, upstream-moving pressure waves and Kelvin Helmholtz (KH) instabilities near the TE, further investigations are needed into the feedback mechanism. The reasonably good correlation between the LPSE results and the growth of disturbance-profiles extracted from time-averaged DNS data supports the assumption of linear instabilities playing a major role in the growth-mechanism of vortex structures close to the trailing edge. A comparison between the growth rates of 2D and oblique modes, obtained by temporal linear stability analysis, suggests that the streamwise wavenumbers, the frequencies, and the locations of the most unstable modes are already well described when only considering 2D disturbances. As the spanwise extent of the DNS is 10% of the axial chord length, the domain is not quite wide enough to capture the mode with spanwise wavenumber $\beta r \approx 40$, which corresponds to the most unstable mode that is suggested by the LST approach. However, the DNS still captures the essential features of 3D breakdown to turbulence within the computational domain and instability growth rates are not very sensitive to βr in this range. The temporal LST approach considering only 2D modes is very stable and reveals reasonable frequencies and growth rates of unstable modes.

The comparably low computational efforts and the possibility for extensive parallelization enable coupling with Reynolds-averaged Navier–Stokes solvers or other applications.

Bontempo and Manna [7] investigated the impact of the standard approximations embodied in the well-known momentum theory (MT) on its performance prediction capabilities. To this aim, the results of the momentum theory, which is still widely used in all blade element/momentum codes for the analysis and/or design of wind turbines, are compared with those obtained with an actuator disk model based on computational fluid dynamics techniques. In this method, the axisymmetric and steady Euler equations are solved with a classical finite volume approach, while the turbine effects are modeled through a set of axial and tangential body forces distributed over a disk-shaped region representing the rotor swept surface. As this method does not rely on the momentum theory simplifying assumptions, it can be suitably employed to verify the momentum theory validity. The analysis is carried out using the well documented experimental data of the National Renewable Energy Laboratory Phase VI wind turbine, for which a large highly accurate database is available. These errors are because of the linearization of the swirl terms and the use of the axial momentum equation in an approximate form. Their quantification has been carried out by comparing the MT results with those of a validated CFD-AD (actuator disk) approach, which does not rely on the same simplifying assumptions of the MT. Contrary to the common belief, these errors are not negligible at all in terms of the axial induction factor. In particular, in the analyzed case, it has been shown that the MT underestimates the axial induction factor near the tip region and overestimates it in the middle span region. On the contrary, no significant errors arise in the tangential induction factor. Finally, the findings reported in their study provide a more meaningful interpretation of the blade element momentum (BEM) validation procedures, which always disregard the impact of the aforementioned errors on the results.

Verstraete et al. [8] introduced a novel parametrization method that uses tri-variate B-spline volumes to deform shape and volume mesh. This parametrization ensures a rich design space while maintaining the geometry description in computer-aided design (CAD) format. Because of its cheap and robust grid generation process, the method can be incorporated efficiently in a one-shot method, which simultaneously converges flow, adjoint, and design. The computational cost of obtaining the flow in the optimized shape is ten times the cost of computing flow and surface sensitivities (adjoint) of the baseline configuration. The method has been applied to minimize the pressure losses of a U-bend passage of a turbine blade serpentine-cooling channel with 540 degrees of freedom. A reduction of 39% in pressure loss was achieved while, at the same time, the mass flow increased by 21%, leading to a non-dimensional reduction in losses of 59%. Finally, the optimized bend shape remains available in CAD format, allowing for a straightforward process to manufacture the optimal shape. The method is a significant step forward in building high-fidelity shape optimization workflows as it removes two major bottlenecks. Firstly, the use of tri-variate B-splines to deform shape and volume consistently provides a robust method of mesh deformation at very low computational cost. Secondly, both the initial and final shapes are defined in a CAD representation that avoids any manual shape approximations in CAD that may lose optimality and are labor intensive. Instead, the optimal shape can be used directly for further analysis or manufacturing.

Bauinger et al. [9] carried out measurements in a one-and-a-half stage test turbine. In order to characterize the flow field and to obtain steady flow quantities, five-hole probes were used in several measurement planes. Although a very high resolution in circumferential and radial direction is possible with five-hole probes, only certain sectors of the flow field can be measured as a result of quite long measurement times. For this reason, total pressure rakes, which can be traversed 360 degrees in a circumferential direction, were built for this test configuration. During foregoing measurements, differences between the total pressure measured with the rakes and the total pressure measured with a five-hole probes were observed, especially in the measurement plane downstream of the rotor where the flow is highly unsteady. It was found that the measured value for total pressure can deviate significantly from the “real” value. Several parameters that can have an influence on total pressure readings of a probe were therefore analyzed. It was found that the most important parameters are stochastic pressure

fluctuations and turbulence, respectively, as they have a significant influence on the total pressure, especially the one measured by Kiel-head probes. It is, therefore, recommended by the authors that, especially in test rigs with high turbulence intensities, unsteady measurements should be carried out in addition to steady measurements with five-hole probes and rakes. The determination of the stochastic fluctuations is inevitable for the evaluation of steady data from pneumatic probes. Still, five-hole probes and rakes are an appropriate tool for aerodynamic measurements in test rigs, but, in connection with high turbulence, absolute values should be handled with care. In highly unsteady flows, an unsteady calibration of the used probes could also help to solve some of the previously discussed issues.

Barigozzi et al. [10] investigated, both experimentally and numerically, the hot streak migration in a linear vane cascade with shower-head film cooling at an isentropic exit Mach number of $Ma_{2is} = 0.40$, with an inlet turbulence intensity level of $Tu_1 = 9\%$. Two tangential positions of the hot streak center were taken into account: 0% of pitch (hot streak is aligned with the vane leading edge) and 45% of pitch. After demonstrating that computations correctly predict the hot streak attenuation through the vane passage without shower-head blowing, the numerical method was used to investigate the hot streak interaction with shower-head film cooling, at a blowing ratio of $BR = 3.0$, corresponding to a coolant-to-mainstream mass flow ratio (MFR) = 1.15%. The effects of mixing and coolant interaction on the hot streak reduction were interpreted under the light of the superposition principle, whose accuracy was within 12% on the leading edge region, in the central section of the vane span. Clocking effects of the inlet temperature non-uniformity on the hot streak attenuation were found to be negligible for the uncooled vane. In fact, positioning the hot streak core at the leading edge or at the mid passage did not change the peak temperature reduction, from $T_{max,HS}/T_{t\infty} = 1.04$, at the inlet, to $T_{max,HS}/T_{t\infty} = 1.005$, at the outlet. Then, the leading edge aligned configuration was numerically investigated subject to shower-head blowing. On the one hand, the additional attenuation of the hot streak due to shower-head film cooling appeared moderate at the vane exit. On the other hand, vane surface temperature in the leading edge region was strongly affected by coolant jet interaction with the approaching head-on hot spot. The stagnation line was exposed to the highest (normalized) temperature, whereas significant hot streak reduction was found outside the shower-head, especially on the pressure side. Finally, the superposition method proved to be fairly capable of predicting hot streak mixing with coolant on the simulated leading edge of a gas turbine vane.

Schneider et al. [11] presented and validated an algorithm generating a complete set of inlet boundary conditions for Reynolds-averaged Navier–Stokes computational fluid dynamics analysis of high-pressure turbines, to investigate their interaction with lean and rich burn combustors. All flow quantities necessary for consistent RANS CFD boundary conditions are generated in a physically closed, iterative procedure from a set of at least 35 parameters determining velocity, temperature, and turbulence. The pressure profile is not open for parameterization, as this would result in an over-constrained problem. The flow profiles are generated to be periodic and to match a specified operating point. It is shown that the method can reproduce highly complex boundary conditions (BCs) derived from a rich burn combustor simulation. The results were compared in terms of stage efficiency, wall temperatures, and Nusselt number in a high-pressure turbine (HPT) located downstream at different clocking positions of the BC relative to the nozzle guide vane (NGV). The algorithm was capable of maintaining the averaged inlet conditions within tight bounds about the operating point. The distribution of the axial velocity at the interface has a strong influence on the mass flow averaged inflow conditions. The potential for improvement of the accuracy of the method lies in improving the modeling assumptions for the distributions of axial velocity $V_{ax}(r,\theta)$ and turbulent quantities, and in a parameterized representation of combustor wall film cooling. The presented method does not account for physical dependencies of the parameters, that is, the derivation of hot spot location and shape from the velocity field in the traverse is not possible. In order to improve the method, these correlations must be determined by statistical comparison with combustor CFD data in order to reduce parameter space and capture physical dependencies to avoid the generation of unrealistic traverses. Also, the parameterization of the quantities at turbine inlet is decoupled from the upstream combustor

geometry and operating condition, because of the highly complex mixing processes in the combustor flow. The method can thus not be used to improve a given combustor design, but only provides information about the downstream propagation of the flow from different HPT inlet conditions. A targeted practical application is matching a reference flow field at combustor exit and investigating how small variations of that field affect the turbine downstream. These variations may represent the deviations of different combustor sectors along the circumference, as well as uncertainties in the prediction of the reference traverse, which may be expressed as differences between simulation and measurement of the same setup, for instance. Obstacles in combining the approach with statistical methods are the large parameter count of the model and the required information on parameter scatter distribution and interrelations. With the current state of research, these gaps can only be filled with empirical knowledge. That is, for a given combustor family, test rig, and so on, one must determine from simulations and/or experiments which of the proposed model parameters vary significantly, how these parameters are distributed, and thus how the complexity of the model can be reduced by neglecting the parameters with minor variation. The quality of the results is thus determined by the quality of available data and experience with the given setup. Hence, application on realistic scales is confined to the qualitative identification of worst-case scenarios at early design phases, leading to problems such as potentially under-cooled regions in the HPT. However, since these critical states cannot be detected by conventional design procedures analyzing only a single traverse, the proposed method can help to increase confidence in the robustness of a design.

Conflicts of Interest: The author declares no conflicts of interest.

References

1. Manna, M. The International Journal of Turbomachinery Propulsion and Power: The Open Access Dissemination Tool for Research on Turbomachinery. *Int. J. Turbomach. Propuls. Power* **2016**, *1*, 1. [[CrossRef](#)]
2. Frey, C.; Ashcroft, G.; Kersken, H.-P.; Schönweitz, D.; Mennicken, M. Simulation of Indexing and Clocking with Harmonic Balance. *Int. J. Turbomach. Propuls. Power* **2017**, *3*, 1. [[CrossRef](#)]
3. Ciorciari, R.; Schubert, T.; Niehuis, R. Numerical Investigation of Secondary Flow and Loss Development in a Low-Pressure Turbine Cascade with Divergent Walls. *Int. J. Turbomach. Propuls. Power* **2018**, *3*, 5. [[CrossRef](#)]
4. Gaetani, P.; Persico, G. Hot Streak Evolution in an Axial HP Turbine Stage. *Int. J. Turbomach. Propuls. Power* **2017**, *2*, 6. [[CrossRef](#)]
5. Suman, A.; Fortini, A.; Aldi, N.; Pinelli, M.; Merlin, M. Analysis of the Aerodynamic and Structural Performance of a Cooling Fan with Morphing Blade. *Int. J. Turbomach. Propuls. Power* **2017**, *2*, 7. [[CrossRef](#)]
6. Zauner, M.; Sandham, N.D.; Wheeler, A.P.S.; Sandberg, R.D. Linear Stability Prediction of Vortex Structures on High Pressure Turbine Blades. *Int. J. Turbomach. Propuls. Power* **2017**, *2*, 8. [[CrossRef](#)]
7. Bontempo, R.; Manna, M. Effects of the Approximations Embodied in the Momentum Theory as Applied to the NREL PHASE VI Wind Turbine. *Int. J. Turbomach. Propuls. Power* **2017**, *2*, 9. [[CrossRef](#)]
8. Verstraete, T.; Müller, L.; Müller, J.-D. Adjoint-Based Design Optimization of an Internal Cooling Channel U-Bend for Minimized Pressure Losses. *Int. J. Turbomach. Propuls. Power* **2017**, *2*, 10. [[CrossRef](#)]
9. Bauinger, S.; Marn, A.; Göttlich, E.; Heitmeir, F. Influence of Pressure Fluctuations on the Mean Value of Different Pneumatic Probes. *Int. J. Turbomach. Propuls. Power* **2017**, *2*, 13. [[CrossRef](#)]
10. Barigozzi, G.; Mosconi, S.; Perdichizzi, A.; Ravelli, S. The Effect of Hot Streaks on a High Pressure Turbine Vane Cascade with Showerhead Film Cooling. *Int. J. Turbomach. Propuls. Power* **2017**, *2*, 15. [[CrossRef](#)]
11. Schneider, M.; Schiffer, H.-P.; Lehmann, K. Parameterised Model of 2D Combustor Exit Flow Conditions for High-Pressure Turbine Simulations. *Int. J. Turbomach. Propuls. Power* **2017**, *2*, 20. [[CrossRef](#)]



© 2018 by the author. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution NonCommercial NoDerivatives (CC BY-NC-ND) license (<https://creativecommons.org/licenses/by-nc-nd/4.0/>).