## APPLICATION OF RANS AND LES TO THE PREDICTION OF FLOWS IN HIGH PRESSURE TURBINE COMPONENTS

Nicolas Gourdain\* Laurent Y.M. Gicquel CERFACS Computational Fluid Dynamics Team 31057 Toulouse Cedex 1 France Email: Nicolas.Gourdain@cerfacs.fr Remy Fransen Elena Collado TURBOMECA DT/MD/MO 64511 Bordes Cedex 1 France

## **Tony Arts**

von Karman Institute Turbomachinery Dpt. Rhodes St Genese, 1640 Belgium

## ABSTRACT

This paper investigates the capability of numerical simulations to estimate unsteady flows and wall heat fluxes in turbine components with both structured and unstructured flow solvers. Different numerical approaches are assessed, from steady-state methods based on the Reynolds Averaged Navier-Stokes (RANS) equations to more sophisticated methods such as the Large Eddy Simulation (LES) technique. Three test cases are investigated: the vortex shedding induced by a turbine guide vane, the wall heat transfer in another turbine guide vane and a separated flow phenomenon in an internal turbine cooling channel. Steady flow simulations usually fail to predict the mean effects of unsteady flows (such as vortex shedding) and wall heat transfer, mainly because laminar-to turbulent transition and the inlet turbulent intensity are not correctly taken into account. Actually, only the LES (partially) succeeds to accurately estimate unsteady flows and wall heat fluxes in complex configurations. The results presented in this paper indicate that this method considerably improves the level of physical description (including boundary layer transition). However, the LES still requires developments and validations for such complex flows. This study also points out the dependency of results to parameters such as the freestream turbulence intensity. When feasible solutions obtained with both structured and unstructured flow solvers are compared to experimental data.

- a Sound speed,
- C Blade chord,
- *H* Heat transfer coefficient (=  $\frac{q_{wall}}{T_0 T_{wall}}$ ),
- M Mach number,
- P Pressure,
- q Heat flux,
- *r* Mixture gas constant ( $r = 287 J.kg^{-1}.K^{-1}$ ),
- Re Reynolds number,
- S Curvilinear abscissa,
- T Temperature,
- Tu Turbulence level,
- $u_i$  Velocity component in direction i,
- x Axial coordinate,
- ρ Density,
- $\Delta t^+$  Normalized time step (=  $\Delta t \times a_0/C$ ),
- .*i* Total value,
- .is Isentropic value,
- .s Static value,
- .0 Inlet value,
- .2 Outlet value,
- .+ Normalized value,
- LES Large Eddy Simulation,
- RANS Reynolds Averaged Navier-Stokes,
- SGS Sub-Grid-Scale,
- SMB Structured multi-block (flow solver),
- UNS Unstructured (flow solver).

NOMENCLATURE

<sup>\*</sup>Address all correspondence to this author.

## INTRODUCTION

Unsteady flows as well as the transfer of thermal energy between a flow and a wall occur in a lot of industrial applications (electronic circuit boards, gas turbines, etc.). The prediction of such flows remain complex due the interaction between different kind of physics such as dynamic and thermal boundary layers, wall thermal properties, etc. In the context of gas turbine applications, the high pressure turbine experiences high temperature gradients at the walls and its life duration directly relies on the capacity of designers to correctly estimate the impact of unsteady flows and wall heat transfer [1]. Unfortunately, these flow phenomena are difficult to predict in such complex environments (high temperature, complex geometry including technological devices such as cooling holes, tip gap, etc.). Turbulence also plays a major role on heat transfer and a laminar to turbulent transition is often observed on the turbine blade walls. This complex phenomenon depends on many parameters such as the Reynolds number, turbulence intensity, wall roughness, shocks, etc. An accurate estimation of the wall heat transfer is thus out of range with classical steady state numerical simulations in most gas turbine configurations.

The physical understanding of such complex flows and the capability to efficiently predict heat transfer at the design stage is mandatory to improve the efficiency of industrial systems such as gas turbines. In this regard a reliable Computational Fluid Dynamics (CFD) code represents a very attractive approach since it induces a relatively shorter response time in comparison to experimental campaigns. But the validation of a CFD code requires important features like well documented test cases, accurate numerical schemes, grid flexibility and validated turbulence and transition modelling/simulation. A large range of numerical methods is available in the literature to simulate near wall flows [2], from steady-state simulations where all the turbulent scales of the flow are modelled to full unsteady flow methods (all turbulent scales are solved). To complete the flow description when turbulence is modelled, criteria can be used to predict boundary layer transition [3-6]). However there are already clear evidences in the literature that numerical methods that solve a part of the turbulent spectrum provide the most promising results regarding the prediction of heat transfer [7–9]. Another aspect is the grid design that directly impacts both the accuracy and the efficiency of the flow solver. Many applications of CFD considering structured grids for turbine investigations are reported in the literature [10]. While effective in terms of accuray/cost ratio, this method also suffers from major drawbacks such as the difficulty to mesh technological devices (cooling holes, etc.) and the possibility to refine localized regions. A potential answer is the use of unstructured grids that represent a promising way for local mesh refinements and take into account very complex geometries [7, 11].

This paper investigates the ability of existing flow solvers to predict unsteady flows and wall heat transfer in turbine test cases.

Three configurations are considered: the vortex shedding in a nozzle guide vane (high Reynolds number), the wall heat transfer in a highly loaded guide vane (high Reynolds number) and a stalled flow in an internal turbine cooling channel (moderate Reynolds number). For each of the three test cases, RANS and LES predictions are compared to experimental data. When feasible, the LES predictions obtained with a structured flow solver and an unstructured one are compared.

#### **GOVERNING EQUATIONS AND TOOLS**

The governing equations are the unsteady compressible Navier-Stokes equations that describe the conservation of mass, momentum and energy. In conservative form, it can be expressed in three-dimensional coordinates as:

$$\frac{dW}{dt} + divF = 0 \tag{1}$$

where *W* is the vector of primary variables,  $F = (f - f_v, g - g_v, h - h_v)$  is the flux tensor; f, g, h are the inviscid fluxes and  $f_v, g_v, h_v$  are the viscous fluxes (including the potential contribution of models for turbulence). In the mathematical description of compressible turbulent flows the primary variables are the density  $\rho(\mathbf{x}, t)$ , the velocity vector  $u_i(\mathbf{x}, t)$  and the total energy  $E(\mathbf{x}, t)$ . The fluid follows the ideal gas law  $P = \rho r T$ , where *r* is the mixture gas constant. The viscous stress tensor and the heat diffusion vector use classical gradient approaches. The fluid viscosity follows Sutherland's law and the heat flux follows Fourier's law.

Direct Numerical Simulation (DNS) of industrial turbine flows is still out of range. The main reason is that the high Reynolds number related to these flows implies that all the flow scales can not be efficiently represented with current grid sizes (the mesh size for DNS scales as  $Re^{\frac{9}{4}}$ ). Turbulence modelling is thus necessary to represent the cascade of energy and different formalisms exist over a wide range of applications, including flows at high Reynolds numbers [2, 12]. The most common approach for complex configurations is still the Reynolds-Averaged Navier Stokes methods (RANS) that model all the turbulent scales. This approach can be used to obtain either a steady-state solution (RANS) or an unsteady solution that contains the deterministic scales of the flow (unsteady RANS), such as rotor-stator interactions or vortex shedding. In these cases, the simulation of the boundary layer transition requires to use transition criteria [3, 6, 10]. A promising method for unsteady turbulent flows is the Large-Eddy Simulation (LES) that introduces a separation between the resolved (large) turbulent scales and the modelled (small) scales [13]. This separation of scales is obtained by filtering out the small flow scales that can not be properly represented by the mesh, their effects on the filtered field being modelled by

the so-called Sub-Grid-Scale (SGS) model [2, 14]. However the capability of LES to describe flows in turbine configurations (including boundary layers) is not yet well established and the need for LES still requires to be demonstrated, mainly because the computational effort is largely increased with respect to classical (U)RANS method.

Three finite-volume flow solvers are used for this study and compared when feasible. The flow solvers AVBP [15, 16] and *elsA* [17] are used for most applications, except for the RANS simulation of the internal cooling channel (third test case) for which the flow solver is the open source software OpenFoam<sup>1</sup>. The *elsA* code considers structured multi-block meshes and is used both for (U)RANS and LES approaches. The AVBP code considers an unstructured formalism adapted to hybrid volumes and is used only for LES. In this paper, OpenFoam and *elsA* will be designated as structured multi-block (SMB) flow solvers and AVBP as an unstructured (UNS) one. More information regarding the High Performance Computing capabilities of *elsA* and AVBP can be found in [18, 19].

# PREDICTION OF THE VORTEX SHEDDING IN A NOZ-ZLE GUIDE VANE

## Experimental configuration

The test configuration comes from the work of Sieverding et al. [20, 21] which is the outcome of the European Research Project BRITE/EURAM CT96-0143 on Turbulence Modeling of Unsteady Flows in Axial Turbines. The design of the blade is targeted to allow the diagnostic of the trailing edge vortex shedding on the steady and unsteady trailing blade pressure distribution of a laboratory turbine blade at high subsonic Mach number ( $M_{is,2} = 0.79$ ) and high Reynolds number ( $Re_2 = 2.8 \times 10^6$ , based on the chord and outlet velocity). The configuration is adapted to preserve the 2D flow as much as possible. A cascade of 4 flow passages composes the experimental setup (only the central vane is investigated to ensure periodicity). Many unsteady flow features are identified as critical in determining the mean flow eld of this experiment. The main flow structure is the vortex shedding issued by the vane trailing edge boundary layer separation. Associated to this separation point is the generation of pressure waves traveling upstream and downstream then eventually interacting with the adjacent vane walls to produce skin vortices which then travel in the downstream direction along the blade wall. The vortex formation and shedding process is visualized using high-speed schlieren camera and a holographic interferometric density measuring technique.

## **Numerical parameters**

Numerical data shown for this nozzle guide vane are only provided by the SMB flow solver (however a comparison with

## 3

Table 1. BLADE CASCADE CHARACTERISTIC DIMENSIONS.

Chord length C	140 mm	
Axial chord length $C_{ax}$	0.656C	
Pitch to chord ratio	0.696C	
Blade height	100 mm	
Trailing edge thickness to chord ratio	0.0531	
Flow inlet angle	$0^{o}$	
Stagger angle	49.83°	
Number of vanes	4	

results obtained from the UNS flow solver has been presented in [22]). In order to proceed with the computation of the Sieverding's configuration, a 3D computational domain corresponding to a single vane passage is chosen. The characteristic dimensions of the domain are provided in Fig. 1 along with the typical blade dimensions given in Table 1. Note that top, bottom and side boundaries of the computational domain are assumed periodic in agreement with the experimental findings. Inlet and outlet flow boundaries are positioned far enough from the profile to limit their impact on the predictions: *i.e.* respectively located 0.33C upstream the leading edge and 1.0C downstream the trailing edge<sup>2</sup>. A no-slip adiabatic wall condition is applied at the blade surface. To reduce the simulation cost, only 5% of the experimental vane span is considered. This assumption is reasonable since the flow is mainly 2D [20,21]. To confirm this point, both RANS simulation and LES have been performed on a grid considering 20% of the experimental span (the number of grid points was increased by the same factor). Negligible differences were observed with respect to the configuration considering only 5% of the vane span.

The initial set of numerical predictions is obtained by use of the same structured multiblock mesh. The mesh is composed of 500,000 hexahedra distributed around the vane. The extrapolation of the slice to the whole vane span would lead to a 10M cells grid. While this is rather a coarse grid for LES, this mesh provides grid-independent results for RANS simulations. The structured blocks are connected with coincident interfaces except at the periodic boundaries which are non-coincident and are treated through a no-match condition. Turbulent closure for (U)RANS relies on the  $k - \omega$  model of Wilcox [23] with no specific treatment at the wall other than the limiting procedure proposed by Zheng *et al.* [24]. Indeed, with the grid generated here, typical

<sup>&</sup>lt;sup>1</sup>http://www.openfoam.com/

<sup>&</sup>lt;sup>2</sup>Sensitivity to the exit boundary condition treatment and relative position from the blade trailing edge has been specifically studied in a dedicated article under review.



Figure 1. TYPICAL MESH TOPOLOGY USED FOR RANS, URANS AND LES.

mean  $y^+$  of the first flow cells of the wall are estimated at 5 guaranteeing reasonable quality boundary layer estimates provided that the turbulent model behaves adequately in these regions of the flow <sup>3</sup>. For LES, the SGS closure is obtained by use of the Wall-Adapting Local Eddy-Viscosity (WALE) model [25], specially built to compute the turbulence effects near walls.

Convective fluxes are computed with a third-order scheme, based on the Advection Upstream Splitting Method (AUSM) and considering a minimal artificial dissipation [26]. Diffusive fluxes are computed with a second-order centered scheme. For steadystate RANS simulations, the pseudo time-marching is performed by using an implicit time integration scheme, based on the backward Euler scheme and a scalar Lower-Upper (LU) Symmetric Successive Over-Relaxation (SSOR) method [27]. LES and URANS computations use a fixed time-step ( $\Delta t = 1.56 \times 10^{-7}$  s, corresponding to  $\Delta t^+ = 4 \times 10^{-4}$ ). Time marching relies on the dual time stepping approach [28].

## Results

Computations are performed on the scalar GENCI-CINES SGI Altix platform with up to 64 computing cores. The computational cost related to (U)RANS and LES can be summarized as follows:  $LES = 10 \times URANS = 100 \times RANS$  ( $\approx 1,500h$  CPU). However, this result should be balanced by the fact that the same grid is used for all computational methods (the grid is too fine for (U)RANS simulations and not large enough for LES). A typical view of the flow quantities, here the norm of the density gradient, are illustrated in Fig. 2 for (a) RANS, (b) URANS and (c) LES. To complement the view, a snapshot of the experiment focusing on the trailing edge region of the flow is provided in Fig. 2 (d). All three numerical formulations result in distinct flow behaviors. RANS provides a mean temporal view of the flow field for the configuration under investigation. With this approach, Fig. 2(a), the local flow acceleration issued by the suction side flow passage restriction is clearly visible and induces a region of density gradient in the upstream part of the suction side. After the blade throat, a density gradient appears indicating the potential presence of a weak shock. The highest density gradients appear on each side of the trailing edge and are linked with the wake region induced by the blade boundary layer separations at the end of the blade and the boundary layer itself. The time dependent description of the flow (URANS) provides new insights on the mean periodic solution, Fig. 2(b). With this approach, the local flow acceleration in the upstream region seems reduced if compared to RANS. The weak shock at the throat is no longer present. At the trailing edge and instead of a mixed out wake, vortex shedding appears along with a network of interacting density fronts (pressure waves in fact). Two distinct sets of waves are identified in agreement with Sieverding et al. [20, 21] and denoted on Fig. 2(d) by  $S_i$  and  $P_i$  respectively. Both sets of waves originate from the boundary layer separation point on the suction and pressure sides of the blade trailing edge. In URANS, the suction side generates pressure waves,  $S_i$ , propagating upstream and interact with the vortical structures present in the wake of the above blade. Their presence within the flow is clear although these  $S_i$ waves seem to be rapidly dissipated by the flow and the numerical model. The pressure side waves,  $P_i$ , also travel upstream but rapidly encounter the suction side wall of the neighboring blade located below. This interaction results in a reflected wave which eventually crosses the  $P_{i+1}$  wave. Further increase in the numerical complexity and turbulent modeling formulation (LES) provides an even finer view on the flow behavior, Fig. 2(c). With LES, all flow structures identified by URANS are present: the vortex shedding from the blade leading edge, both sets of pressure waves and their propagation. The  $P_i$  waves are also interacting with the main flow stream and impact the lower blade suction side wall. The main difference between URANS and LES appears on these instantaneous views to be highly local. The trailing edge sheds vortical structures that are more persistent in LES than in URANS producing more interactions between the wake and the  $S_i$  waves.

An unambiguous validation of the mean flow predictions is obtained thanks to a comparison of the isentropic Mach number distribution along the blade surface, Fig. 3. Again, going from a purely stationary numerical model to an unsteady model clearly improves the flow predictions. Hence and in agreement with the discussion started above, the RANS prediction leads to a local miss-representation of the flow field. In particular, RANS predicts that the flow is choked at the passage throat ( $M_{is} > 1$ ) when it is not observed in the experiment. URANS and LES allow net improvements when compared to RANS, with relatively small and only localized distinctions between the two unsteady flow approaches. Differentiation between URANS and LES is better

<sup>&</sup>lt;sup>3</sup>Without points inside the viscous sublayer (*i.e.*  $y^+ < 1$ ) and without appropriate functions, the  $k - \omega$  turbulence model cannot be guaranteed to predict the correct wall shear stress.



Figure 2. INSTANTANEOUS FLOW FIELD COLORED WITH THE NORM OF THE DENSITY GRADIENT: (a) RANS, (b) URANS, (c) LES AND (d) EXPERIMENT (SCHLIEREN VISUALIZATION) [20,21].



Figure 3. MEAN ISENTROPIC MACH DISTRIBUTION ALONG THE BLADE WALL PREDICTED NUMERICALLY AND MEASURED IN THE EXPERIMENT.

indicated by purely unsteady flow phenomena such as the wake shedding frequency that is provided on Table 2 and is expressed in terms of Strouhal number <sup>4</sup>. Differences are also identified when looking at the trailing edge mean pressure field as shown in Fig. 4. For this specific region, only LES seems to recover the pressure level measured experimentally, URANS offering an important alternative to RANS.

Preliminary conclusions on the numerical formulation to be used to reproduce the turbulent flow encountered in a turbine blade at high subsonic outlet number are as follows. First, it



Figure 4. MEAN PRESSURE DISTRIBUTION ALONG THE BLADE TRAILING EDGE AS PREDICTED NUMERICALLY AND MEASURED IN THE EXPERIMENT.

Table 2. VORTEX SHEDDING FREQUENCY EXPRESSED IN TERMS OF STROUHAL NUMBER.

Experiment	URANS (error)	LES (error)
0.219	0.276(+26%)	0.228(+4%)

seems important to be able to take into account the unsteady nature of the physics involved. This observation implicitly disqualifies the RANS approach although the use of second order modeling strategies may be of interest (which is not the type of closure proposed here). Second, the use of URANS offers a net improvement over RANS and again higher order closures seem recommended to better capture turbulence interactions. Finally, LES, which is a fully unsteady numerical approach, captures most of the physics reported by the experimentalists. Further investigations need however to be conducted as LES predictions are by construction mesh dependent as well as very sensitive to numerics and wall modeling.

## PREDICTION OF THE AEROTHERMAL PERFOR-MANCE OF A NOZZLE GUIDE VANE Experimental test case

The second configuration is a 2D turbine blade cascade (the so-called LS 89 blade) largely described in [29]. The vane is mounted in a linear cascade made of five profiles (only the central vane is investigated to ensure periodic flow conditions). The vane chord C is 67.647 mm with a pitch/chord ratio of 0.85. The real vane span is 100 mm. Experimental investigations were done to measure the vane velocity distribution by means of static pressure tappings and convective heat transfer by means of a transient technique using platinum thin films. Uncertainties were

<sup>&</sup>lt;sup>4</sup>The spectral analysis relies on a time series of 3 ms obtained for a numerical and experimental probes located at  $x/c_{ax} = 0.933$ . Note that this duration corresponds approximately to 20 cycles of the wake shedding.

quantified for these measurements (pressure:  $\pm 0.5\%$  and heat transfer coefficient:  $\pm 5\%$ ). Two freestream conditions are explored in this paper, as shown in Table 3 (test cases MUR129 and MUR235). The Reynolds number (based on the chord and outlet velocity) is  $1.1 \times 10^6$  and the inlet turbulence intensity  $Tu_0$  ranges from 1% to 6%.

	Table 3. Test cases and flow conditions					
Test case	$Re_2$	$M_{is,2}$	$P_{i,0}$	$T_{s,wall}$	$Tu_0$	
MUR129	$1.1310^{6}$	0.927	1.87 10 <sup>5</sup> Pa	298 K	1.0%	
MUR235	$1.1510^{6}$	0.927	1.85 10 <sup>5</sup> Pa	301 K	6.0%	

#### **Numerical parameters**

Experiments show that the mean flow is 2D. Preliminary computations have shown that considering about 10% of the span is sufficient to provide span-independent results (a similar observation is reported by Bhaskaran and Lele [30] for the use of LES in this configuration). Indeed, only 10% of the vane span is meshed (this simplification neglects end-wall effects but retains the three-dimensionality of the flow). Periodic conditions are assumed for lateral and radial boundaries (only the flow in the central passage is computed). The inlet (resp. outlet) is located up to one axial chord uptream (resp. two axial chords downstream) of the vane. Static pressure is imposed at the outlet to set the targeted Mach number ( $M_{is,2} = 0.92$ ). An isothermal no-slip condition is applied at the walls with  $T_{wall}$ =301.15K, as imposed in the experiment. To study the influence of the turbulence intensity on boundary layers, flow simulations must account for the correct level of the turbulence intensity at the inlet. Both flow solvers use a synthetic turbulent eddy method to mimic the turbulence effects [31, 32]. More information about the inlet turbulent boundary condition (and its effect on the wall heat transfer) can be found in Gourdain et al. [33].

<u>a) For the SMB flow solver</u>: the flow domain in the guide vane passage is discretized with a multi-block approach. A view of the computational domain is presented in Fig. 5(a). The minimum cell size is set to less than 2  $\mu$ m all around the blade. The maximum value of  $y^+$  is below 2 all around the vane. In other directions, normalized mesh spacings are kept under acceptable values ( $\overline{\Delta z^+} = 25$  and  $\overline{\Delta x^+} = 150$ ). The vane passage is represented with a 29.7 × 10<sup>6</sup> points grid (for LES) and a 0.7 × 10<sup>6</sup> points grid (for RANS<sup>5</sup>). For RANS simulations, convective fluxes are computed with a second order centered scheme using classical artificial dissipation parameters ([34]). Diffusive fluxes are computed with a second-order centered scheme. The pseudo time-marching is performed by using an implicit time integration scheme, based on the backward Euler scheme and a scalar Lower-Upper (LU) Symmetric Successive Over-Relaxation (SSOR) method as proposed in [27]. The turbulent viscosity is computed with the two equations model of Smith [35] based on a k - l formulation and transition is detected with the criterion proposed in [36]. For LES, convective fluxes are computed with a fourth order centered scheme, considering a minimal artificial dissipation [37]. The subgrid scale model is the WALE model  $[25]^6$ . The time-marching scheme is based on a second order Dual Time Stepping method [28]. 4,000 time steps are necessary to describe one through-flow time (the time for a particle dropped at the inlet to reach the outlet, *i.e.*  $\approx 2.0ms$ ). It corresponds to a time step of  $5 \times 10^{-7}$ s (*i.e.*  $\Delta t^+ = 3 \times 10^{-3}$ ).

b) For the UNS flow solver: a hybrid approach is adopted in order to reduce the number of cells near walls (prisms layers at the wall and tetrahedra in the domain). The grid considered by the UNS flow solver is shown in Fig. 5(b). To achieve a mean wall distance  $y^+$  of 10, the minimum wall cell size needed is approximately  $8\mu$ m. The solution adopted has 5 layers of prisms in the wall normal direction where the vertical length of the prism  $\Delta y$  is smaller than the triangle basis length  $\Delta x$ . A consequence is that the minimum cell volume is increased in comparison to a full tetrahedral mesh. A limit is imposed to this mesh adaptation to avoid numerical errors in these layers: the aspect ratio of the first and thinnest layer is set to  $\Delta x^+ \approx \Delta z^+ \approx 3y^+$ . With this strategy, the grid requires  $29.3 \times 10^6$  cells ( $6.3 \times 10^6$  prisms and  $23 \times 10^6$  tetrahedra). At the inlet of the domain, total temperature and total pressure are enforced using the Navier-Stokes Characteristic Boundary Condition -NSCBC- formalism [38]. As for the SMB flow solver, the SGS is the WALE model. Convective fluxes are solved using a finite element TTG4A based on a two step Taylor Galerkin formulation with a cell-vertex diffusion scheme which is especially designed for LES on hybrid meshes. This explicit scheme is adequate for the low-dissipation demand of the LES applications and provides 3rd order space and time accuracy [39]. The time-marching scheme is based on an explicit method (CFL < 1). In the present case, 100.000 iterations are used to discretize one through-flow time (i.e 2ms), corresponding to a time step of  $2 \times 10^{-8} s$  (*i.e.*  $\Delta t^+ = 1.2 \times 10^{-4}$ ).

<sup>&</sup>lt;sup>5</sup>the RANS grid is identical to the LES grid except in the spanwise direction where only 5 points are used (instead of 200 for LES). This grid is fine enough to provide grid-independent results for RANS simulations.

<sup>&</sup>lt;sup>6</sup>A major asset of the WALE model is its behavior close to walls (the turbulent viscosity tends toward 0). In this case the turbulent contribution to wall heat transfer is thus negligible.



Figure 5. COMPUTATIONAL DOMAIN AND MESHES FOR THE FLOW SIMULATION IN THE INVESTIGATED CONFIGURATION - (A) STRUCTURED GRID, (B) UNSTRUCTURED GRID.

## Results

Flow simulations are performed on the GENCI-CINES SGI Altix platform (with up to 256 scalar computing cores). The computational cost can be summarized as follow:  $LES_{UNS} =$  $4 \times LES_{SMB} = 6500 \times RANS$  ( $\approx 140,000h$  CPU). The cost for LES is mainly related to the increase of grid density, the time needed to converge flow statistics (about 10 through-flow times) and extract data for analysis (yet 5 more through-flow times). The evolution of the isentropic Mach number is plotted in Fig. 6 with respect to the curvilinear abscissa *S* confirming that the simulation roughly correctly matches the experimental flow conditions. The only discrepancy comes from the strength of the shock on the suction side that is slightly overestimated with both flow solvers.

A qualitative analysis of the flow can be deduced from the instantaneous flow field of wall heat fluxes shown in Fig. 7 (MUR129) and Fig. 8 (MUR235). For a low inlet turbulence intensity (MUR129,  $Tu_0 = 1\%$ ), the boundary layer transition on the suction side is mainly driven by the adverse pressure gradient. As shown in Fig. 7, both flow solvers predict that the flow



Figure 6. TIME-AVERAGED ISENTROPIC MACH NUMBER DISTRI-BUTION ALONG THE BLADE WALL PREDICTED WITH SMB ANS UNS FLOW SOLVERS (MUR 129).

remains perfectly uniform in the spanwise direction only until S = 40mm. At this location, acoustic waves that are emitted at the blade trailing edge impact the vane suction side and disturbances are observed in the radial direction. However these disturbances are rapidly damped and the transition is finally triggered by the interaction between the normal shock and the laminar boundary layer close to S = 60mm. Predictions with both flow solvers are quite similar all along the vane chord. However, the UNS flow solver tends to predict a lower value of the wall heat flux after transition when compared to the SMB prediction. The transition process is also a bit different: the results obtained with the UNS flow solver show the development of turbulent spots at S = 50mm (before the global transition occurs) while the SMB flow solver predicts a uniform transition at S = 60 mm. At a higher turbulence intensity (MUR235,  $Tu_0 = 6\%$ ), the transition proceeds more rapidly and the natural transition is by-passed. The impact of turbulent flow patterns at the blade leading edge is well pointed out in Fig. 8. The development of long streaky flow features is also observed on the pressure side. This mechanism looks like Gortler vortices ([40]) and is responsible for a raise of the wall heat transfer (experiments show an increase by 80%). On the suction side, the boundary layer is laminar until S = 20mm. After this point, turbulent spots develop in the boundary layer (2 < z < 4) due to the adverse pressure gradient (see the small plateau at S = 25mm on the isentropic Mach number curve in Fig. 6) and the boundary layer becomes fully turbulent slightly before the shock at S = 55mm.

A more quantitative comparison is shown in Fig. 9 and 10 (SMB flow solver) and Fig. 11 (UNS flow solver). The MUR129 test case is the simplest test case since the boundary layers re-



Figure 7. INSTANTANEOUS WALL HEAT FLUXES Q(W/CM2) COM-PUTED WITH LES (MUR129): (a) SMB FLOW SOLVER, (b) UNS FLOW SOLVER.



Figure 8. INSTANTANEOUS WALL HEAT FLUXES Q(W/CM2) COM-PUTED WITH THE SMB FLOW SOLVER (LES, MUR235).

main mainly laminar on both suction and pressure sides. The heat transfer coefficient H is plotted on Fig. 9(a) (SMB RANS), Fig. 9(b) (SMB LES) and Fig. 11(a) (UNS LES). Both RANS and LES correctly estimate the wall heat fluxes on the pressure side and on most of the suction side. The heat transfer falls quickly after the leading edge, corresponding to the development of a laminar boundary layer both on pressure and suction sides. However, RANS and LES predict a boundary layer transition at S = 62mm (corresponding roughly to the shock position) that is not seen on the experimental curve (what is interest-

ing is that all simulations predict this transition, including SMB and UNS flow solvers). This difference is related to the overestimation of the shock strength by the simulation (with both flow solvers), as shown in Fig. 6. The main difference between RANS and LES after transition is the level of the heat transfer coefficient:  $H_{SMB,RANS} = 1100W/m^2.K$ ,  $H_{SMB,LES} = 600W/m^2.K$  and  $H_{UNS,LES} = 450W/m^2.K$  (experimental data give  $H_{Exp.} = 600W/m^2.K$ ).

The MUR235 test case is much more complicated than the MUR129 test case, mainly due to the high inlet turbulence intensity. The heat transfer coefficient H is plotted on Fig. 10(a) (SMB RANS), Fig. 10(b) (SMB LES) and Fig. 11(b) (UNS LES). Experiments show a "pre-transition" region from S = 25mm (corresponding to the plateau on the isentropic Mach number curve in Fig. 6(a)) to S = 60mm. After transition (S > 60mm), the heat transfer on suction side is largely increased ( $H \approx 800W/m^2.K$ ). Another effect of the high inlet turbulence intensity is to increase the heat flux on the pressure side wall by 50% with respect to the "purely" laminar boundary layer (MUR129 test case). The analysis of numerical data indicate that the RANS simulation fails to accurately predict the wall heat transfer in this configuration. On the pressure side, the criterion does not detect any transition and the heat transfer coefficient H is identical to the MUR129 test case. As a consequence, the value of H is underestimated by 50%. The value of H near the leading edge is also underestimated by about 30%, highlighing the influence of inlet turbulent flow patterns. On the suction side, the RANS simulation finds the correct location for the onset of transition ( $S \approx 20mm$ ) but it fails to estimate the transition length, leading to a strong overestimation of the heat transfer coefficient (at S = 30mm, the heat transfer is overestimated by 250%). After S > 65mm, the boundary layer is fully turbulent and RANS predicts the correct order of magnitude for the guide vane heating. These results are in agreement with other numerical works that consider RANS methods [3].

On the one hand, at these flow conditions the contribution of LES with the SMB flow solver is very interesting (Fig. 10(b)). First, the impact on the pressure side of inlet turbulent flow features is partially taken into account, as already shown by [41]. LES still underestimates the heat transfer on the pressure side, but the difference with experimental data is reduced to 20%. Then, the "pre-transition" region from S = 25mm to S = 60mm is correctly predicted by a LES. On the suction side, the heat transfer coefficient is estimated with an error less than 5% (i.e. the experimental uncertainty) until S = 65mm. When the experimental boundary layer becomes fully turbulent (close to the trailing edge), a LES with the SMB flow solver underestimates the wall heat transfer by 25%. On the other hand, the LES with the UNS flow solver is able to predict the wall heat transfer for low inlet turbulence intensities but it fails to estimate the effect of  $Tu_0$  on the wall heat transfer (Fig. 11(b), except on the pressure side). The reason seems to be that the dissipation in the boundary lay-



Figure 9. HEAT TRANSFER COEFFICIENT H PREDICTED WITH THE SMB FLOW SOLVER (MUR129): (a) RANS, (b) LES.

ers with the UNS flow solver is too strong. The result is that the turbulent flow patterns are damped and the boundary layer transition can not occur before the shock.

Preliminary conclusions are: 1) the RANS approach coupled with a transition criterion is effective in predicting wall heat transfer only when the boundary layer transition does not play a major role (moreover the drawbacks of transition criteria are their lack of "universality" and the difficulty to use them in an industrial context), 2) LES is a very promising method for predicting wall heat transfer in industrial turbine configurations, especially when an accurate description of the boundary layer transition is necessary since this method is able to describe natural and bypass transitions as well as the transition triggered by the shockboundary layer interaction. However, its major drawback is its computational cost.



Figure 10. HEAT TRANSFER COEFFICIENT H PREDICTED WITH THE SMB FLOW SOLVER (MUR235): (a) RANS, (b) LES.

# PREDICTION OF THE FLOW INSIDE INTERNAL COOL-ING DUCTS

## Experimental configuration

The third configuration is a turbine cooling channel as shown in Fig. 12. The measurements were performed at a fixed Reynolds number  $(40 \times 10^3 \text{ with an uncertainty of about 2\%})$ , corresponding to a bulk velocity  $U_b$  of 8.8 m/s. Highly resolved inlet and exit conditions were carefully determined by miniature pressure and temperature probes. Detailed measurements allowed quantifying the overall pressure drop across the channel with an uncertainty below 3%. Two-dimensional PIV measurements provided mean velocity and kinetic energy fields in the symmetry plane as well as in a plane close to the bottom wall. The corresponding uncertainties were respectively 2 and 5%. A detailed description of the experiments is provided in [42].

## **Numerical parameters**

The calculation domain presented in Fig. 12 is used in this study for both RANS simulations and LES. The channel had a



Figure 11. HEAT TRANSFER COEFFICIENT H PREDICTED WITH THE UNS FLOW SOLVER (LES): (a) MUR129 AND (b) MUR235.

square cross-section with a side length  $D_h = 0.075m$ . The inlet and outlet legs are both parallel to the *y*-axis and are linked with a  $180^{\circ}$  U-bend. The outer radius of the turn is equal to  $1.26D_h$  and both legs are  $8.1D_h$  long, but a preliminary RANS computation with a  $20D_h$  long inlet leg has been carried out to obtain a more developped velocity profile to impose at the inlet of the domain shown in Fig.12.

RANS data are computed using the open source software OpenFoam (a SMB flow solver). The structured multiblock mesh is composed of 900,000 cells (which is enough to ensure gridindependent results). The low-Reynolds turbulence model of Launder-Sharma  $k - \varepsilon$  model [43] is adopted for this study. For LES, the UNS code AVBP is used. The WALE model [25] is chosen to model the SGS viscosity. Convective fluxes are computed with the TTGC scheme [39]. The mesh used for the LES computation is made of approximately  $6 \times 10^6$  cells. The mean normalized wall distance  $y^+$  is around 10 and the other grid spacings at walls are set to  $\Delta x^+ \approx \Delta z^+ \approx y^+$ . Classic non-slip adiabatic boundary conditions are imposed at the walls. At the domain



Figure 12. CALCULATION DOMAIN WITH  $180^\circ$  U-BEND AND INLET AND OUTLET LEGS.

inlet, a velocity profile supplied by the 3D RANS computation previously mentionned is applied with a uniform temperature of 293*K*. At the outlet, a static pressure condition is imposed using an NSCBC approach [38]. The time step is set to  $4.5 \times 10^{-7}s$  (*i.e.*  $\Delta t^+ = 1 \times 10^{-4}$ ), corresponding to 400.000 time steps to describe one through-flow time (*i.e* 0.2*s*).

#### Results

Figure 13 shows the PIV measurement results in the midheight plane (ie.  $Z/D_h = 0.5$ ), with inlet leg on the left under the line L1, the outlet leg on the right under the line L3 and the bend between L1 and L3. The magnitude of the mean velocity is presented, non-dimensionalized by the experimentally measured bulk velocity. In the inlet leg, the classical channel flow is more and more curved because of the presence of the U-bend. Then the fluid accelerates on the inner wall of the first mid-part of the turn (between L1 and L2) and separates around the extremity of the line L2. As a consequence, a recirculation bubble is created in the second part of the bend (between L2 and L3), which is propagated downstream of the bend until about  $1D_h$  on the inner wall of the outlet leg. On the outer wall, the fluid velocity initialy decreases to  $0.1U_b$  when it enters in the turn, then progressively increases all along the bend to a maximum around  $1.8U_b$  at the beginning of the outlet leg, due to the large recirculation zone on the opposite inner wall. This topology leads to two zones where the turbulent kinetic energy (TKE) exceeds 5%. The first one, where TKE stay lower than 10%, is in the the shear layer between the fast mid-channel flow and the slower flow on the outer wall. The second one, much more turbulent, is produced by the separated flow on the inner wall of the turn and the recirculation



Figure 13. 2D MEAN FLOW TOPOLOGY FROM PIV IN MID-PLANE.

bubble. The TKE increases here towards much higher values until 25 - 30%.

All simulations are performed on the GENCI-CINES SGI Altix computer with up to 512 scalar computing cores. The computational cost can be summarized as follows:  $LES = 1700 \times$ *RANS* ( $\approx 65,000h$  CPU). For the comparison of RANS and LES results with PIV measurements, the previous three lines L1, L2 and L3 are chosen to probe data. The TKE is plotted in Fig. 14 (station L1), Fig. 15 (station L2) and Fig. 16 (station L3). At station L1, RANS and LES accurately estimate the TKE at  $-1.2 < x/D_h < -1.1$ . In the rest of the channel, discrepancies between numerical and experimental data increase: RANS overestimates the TKE by 5% to 100% while LES predicts that TKE tends to 0 at  $-1.1 < x/D_h < 0.3$  (LES predicts a non-zero TKE only in the separated region). At station L2, the interest for LES is obvious: RANS predicts a quasi-uniform value of the TKE while LES shows that TKE is mainly produced in the separated region close to the inner wall, as shown by experiments  $(10.4 < y/D_h < 10.5)$ . At station L3, the shape of the experimental curve is correctly estimated with LES (TKE is mainly produced in the separated flow region close to the inner wall) but not by the RANS simulation (discrepancies are close to 50%).

The preliminary conclusion is that LES estimates the flow separation (and its consequences on the mean flow) with a better accuracy than RANS. However, the effect of the inlet turbulence intensity in this cooling channel is still not well predicted both with RANS and LES (see station L1, Fig. 14).

## CONCLUSION

This paper describes the investigations done about the prediction of unsteady flows and wall heat transfer in turbine com-



Figure 14. TURBULENT KINETIC ENERGY AT STATION L1.



Figure 15. TURBULENT KINETIC ENERGY AT STATION L2.

ponents (two guide vane at high Reynolds numbers and one cooling passage). Results obtained with (U)RANS and LES methods have been compared. For all investigated flows, LES provides (as expected) the best accuracy. The main interest for LES is its capability to describe laminar to turbulent transition of boundary layers and unsteady flows (such as vortex shedding). LES is also able to take into account the effect of the inlet turbulence intensity on the wall heat transfer (while (U)RANS coupled with transition criteria is unable to do it). However, the over cost related



Figure 16. TURBULENT KINETIC ENERGY AT STATION L3.

to LES compared to RANS is more than a factor 1000 which still limits its application in an industrial context. This ratio should still be higher in the case of full 3D configurations [44]. Indeed, while LES provides detailed data, it should be considered as a very powerful tool to validate a design more than a design tool in itself. From the research point of view, LES gives valuable data to improve the knowledge of complex flows such as those observed in gas turbines.

## ACKNOWLEDGMENT

Many thanks to the VKI teams for the experimental test cases that are used in this paper. The work presented has also largely benefited from CERFACS internal and GENCI-CINES computing facilities (under the project fac 6074). This support is greatly acknowledged. The authors also thank the CERFACS CFD team, for helpful discussions about LES and flow physics.

## REFERENCES

- Han, J. C., Dutta, S., and Ekkad, S. V., 2001. Gas Turbine Heat Transfer and Cooling Technology. Taylor & Francis, New York, NY, USA.
- [2] Sagaut, P., 2000. *Large Eddy Simulation for incompressible flows*. Scientific computation series. Springer-Verlag.
- [3] Martelli, F., Adami, P., and Belardini, E., 2003. Heat transfer modelling in gas turbine stage. Tech. Rep. ADA419187, University of Florence.
- [4] Smirnov, E., and Smirnovsky, A., 2009. "Turbine vane cascade heat transfer predictions using a modified version of

the -ret laminar-turbulent transition model". In Int. Symp. On Heat Transfer in Gas Turbine Systems.

- [5] Stripf, M., Schulz, A., and Bauer, H.-J., 2008. "Modeling of rough-wall boundary layer transition and heat transfer on turbine airfoils". *J. Turbomachinery*, 130(2).
- [6] Liu, Y., 2007. "Aerodynamics and heat transfer predictions in a highly loaded turbine blade". *Int. J. of Heat and Fluid Flow*, 28, pp. 932–937.
- [7] Duchaine, F., Mendez, S., Nicoud, F., Corpron, A., Moureau, V., and Poinsot, T., 2009. "Coupling heat transfer solvers and large eddy simulations for combustion applications". *Int. J. of Heat and Fluid Flow,* 30, pp. 1129–1141.
- [8] Niceno, B., Dronkers, A. D. T., and Hanjalic, K., 2002. "Turbulent heat transfer from a multi-layered wall-mounted cube matrix: a large eddy simulation". *Int. J. of Heat and Fluid Flow*, 23, pp. 173–185.
- [9] Zhong, B., and Tucker, P. G., 2005. "Les and hybrid les/rans simulations for conjugate heat transfer over a matrix of cubes". In 43rd AIAA Aerospace Sciences Meeting and Exhibit.
- [10] Boyle, R. J., and Ameri, A. A., 1997. "Grid orthogonality effects on turbine midspan heat transfer and performance". *J. Turbomachinery*, **119**(1), pp. 31–38.
- [11] Kwon, O. J., and Hah, C., 1995. "Simulation of three-dimensional turbulent flows on unstructured meshes". *AIAA J.*, 33(6).
- [12] Pope, S. B., 2000. *Turbulent flows*. Cambridge University Press.
- [13] Poinsot, T., and Veynante, D., 2005. *Theoretical and Numerical Combustion*. R.T. Edwards, 2nd edition.
- [14] Ferziger, J. H., 1977. "Large eddy simulations of turbulent flows". AIAA J., 15(9), pp. 1261–1267.
- [15] Schönfeld, T., and Poinsot, T., 1999. "Influence of boundary conditions in LES of premixed combustion instabilities". In *Annual Research Briefs*. Center for Turbulence Research, NASA Ames/Stanford Univ., pp. 73–84.
- [16] Mendez, S., and Nicoud, F., 2008. "Large-eddy simulation of a bi-periodic turbulent flow with effusion". J. Fluid Mech., 598, pp. 27–65.
- [17] Cambier, L., and Veuillot, J. P., 2008. "Status of the elsa cfd software for flow simulation and multidisciplinary applications". In 46th AIAA Aerospace Science Meeting and Exhibit, no. 664.
- [18] Gourdain, N., Gicquel, L., Montagnac, M., Vermorel, O., Gazaix, M., Staffelbach, G., Garcia, M., Boussuge, J.-F., and Poinsot, T., 2009. "High performance parallel computing of flows in complex geometries - part 1: Methods". J. of Computational Sciences and Discovery, 2(015003).
- [19] Gourdain, N., Gicquel, L., Staffelbach, G., Vermorel, O., Duchaine, F., Boussuge, J.-F., and Poinsot, T., 2009. "High performance parallel computing of flows in complex geometries - part 2: Applications". J. of Computational Sci-

ences and Discovery, 2(015004).

- [20] Sieverding, C., Richard, H., and Desse, J.-M., 2003. "Turbine blade trailing edge flow characteristics at high subsonic outlet mach number". *Transaction of the ASME*, *125*, pp. 298–309.
- [21] Sieverding, C., Ottolia, D., Bagnera, C., Comadoro, A., Brouckaert, J.-F., and Desse, J.-M., 2004. "Unsteady turbine blade wake characteristics". *J. Turbomach.*, *126*, pp. 551–559.
- [22] Leonard, T., Duchaine, F., Gourdain, N., and Gicquel, L. Y. M., 2010. "Steady/unsteady reynolds averaged navierstokes and large eddy simulations of a turbine blade at high subsonic outlet mach number". In ASME Turbo Expo.
- [23] Wilcox, D., 1988. "Reassessment of the scale-determining equation for advanced turbulence models". *AIAA J.*, 26, pp. 1299–1310.
- [24] Zheng, X., and Liu, F., 1995. "Staggered upwind method for solving navier-stokes and k-ω turbulence model equations". AIAA J., 33(6), pp. 991–998.
- [25] Nicoud, F., and Ducros, F., 1999. "Subgrid-scale stress modelling based on the square of the velocity gradient". *Flow, Turb. and Combustion*, 62(3), pp. 183–200.
- [26] Liou, M. S., 1996. "A sequel to ausm: Ausm+". J. Computational Physics, 129, pp. 364–382.
- [27] Yoon, S., and Jameson, A., 1987. "An LU-SSOR scheme for the euler and navier-stokes equations". In AIAA 25th Aerospace Sciences Meeting, no. 0600 in 87.
- [28] Jameson, A., 1991. "Time dependent calculations using multigrid, with applications to unsteady flows past airfoils and wings". In AIAA Computational Fluid Dynamics Conference.
- [29] Arts, T., Lambert de Rouvroit, M., and Rutherford, A. W., 1990. Aero-thermal investigation of a highly loaded transonic linear turbine guide vane cascade. Technical Note 174, Von Karman Institute.
- [30] R., B., and Lele, S. K., 2010. "Large eddy simulation of free-stream turbulence effects on heat transfer to a highpressure turbine cascade". J. of Turbulence, 11(6), pp. 1– 15.
- [31] Jarrin, N., Benhamadouche, S., Laurence, D., and Prosser, R., 2006. "A synthetic eddy method for generating inflow conditions for large eddy simulations". *Int. J. of Heat and Fluid Flow*, 27(585-593).
- [32] Celik, I., A., S., and Smith, J., 1999. "Appropriate initial and boundary conditions for les of a ship wake". In Proceedings of 3rd ASME/JFE Fluids Engineering Conference.
- [33] Gourdain, N., Gicquel, L., and Collado, E., 2011. "Comparison of RANS simulation and LES for the prediction of heat transfer in a highly loaded turbine guide vane". In European Turbomachinery Conference.
- [34] Jameson, A., Schmidt, W., and Turkel, E., 1981. "Numeri-

cal solution of the euler equations by finite volume methods using runge-kutta time stepping schemes". In AIAA 14th Fluid and Plasma Dynamics Conference.

- [35] Smith, B. R., 1995. "Prediction of hypersonic shock wave turbulent boundary layer interactions with the k-l two equaton turbulence model". In AIAA 33rd Aerospace Sciences Meeting and Exhibit.
- [36] Abu-Ghannam, B., and Shaw, R., 1980. "Natural transition of boundary layers - the effects of turbulence, pressure gradient, and flow history.". J. of Mechanical Engineering Science, 22(5), pp. 213–228.
- [37] Ducros, F., Ferrand, V., Nicoud, F., Weber, C., Darrack, D., and Poinsot, T., 1999. "Large-eddy simulation of the shock/turbulence interaction". *J. Computational Physics*, *152*, pp. 517–549.
- [38] Poinsot, T., and Lele, S., 1992. "Boundary conditions for direct simulations of compressible viscous flows". J. Comput. Phys., 101(1), pp. 104–129.
- [39] Colin, O., and Rudgyard, M., 2000. "Development of highorder taylor-galerkin schemes for unsteady calculations". *J. Comput. Phys.*, 162(2), pp. 338–371.
- [40] Saric, W. S., 1994. "Gortler vortices". Annu. Rev. Fluid Mech., 26.
- [41] Bhaskaran, R., and Lele, S. K., 2010. "Large eddy simulation of free stream turbulence effects on heat transfer to a high-pressure turbine cascade". *J. of Turbulence*, *11*(6).
- [42] Verstraete, T., Coletti, F., Bulle, J., Vanderwielen, T., and Arts, T., 2011, under review. "Optimisation of a u-bend for minimal presure loss in internal cooling channels – part 2: Measurements". In IGTI Conference, no. GT2011-46541.
- [43] Launder, B. E., and Sharma, B. I., 1974. "Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc". *Letter in heat and mass transfer*, *1*(2), pp. 131–138.
- [44] Gomar, A., Gourdain, N., and Dufour, G., 2011. "Highfidelity simulation of the turbulent flow in a transonic axial compressor". In European Turbomachinery Conference.