GT2010-22793

ADVANCED NUMERICAL SIMULATION DEDICATED TO THE PREDICTION OF HEAT TRANSFER IN A HIGHLY LOADED TURBINE GUIDE VANE

Nicolas Gourdain* CERFACS Computational Fluid Dynamics Team 42, avenue G. Coriolis 31057 Toulouse Cedex 1 France Email: Nicolas.Gourdain@cerfacs.fr Florent Duchaine Laurent Y.M. Gicquel CERFACS Computational Fluid Dynamics Team 42, avenue G. Coriolis 31057 Toulouse Cedex 1 France

Elena Collado

TURBOMECA DT/MD/MO 64511 Bordes Cedex 1 France

ABSTRACT

Wall heat transfers take place in a wide range of industrial applications such as gas turbine and electronic components. The capacity to correctly predict these heat fluxes has a direct impact on the component life duration and the system performance. This paper proposes to investigate this specific problem in a well documented turbine guide vane configuration, by means of a numerical approach. The test case has been designed to obtain flows close to those observed in a real modern highly loaded turbine (transonic operating conditions, laminar to turbulent transition, etc.). The flow is simulated thanks to both structured and unstructured flow solvers. A wide range of numerical methods 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. As expected steady RANS fails to predict the wall heat transfer in the investigated configuration, mainly because unsteady flows and laminar-to turbulent transition are not taken into account. Another intermediate approach is the unsteady RANS method that is able to represent a part of the unsteady flow phenomena such as vortex shedding. The results underlines the role of these flows on the heat transfer coefficient, mainly due to acoustic waves that are emitted at the blade trailing edge and interact with the suction side boundary layer. However, only the LES (partially) succeeds to estimate wall heat fluxes since this method considerably improved the level of physical description by simulating a part of the turbulent flow spectrum (including boundary layer transition). However, the LES still requires validation and developments for such complex flows. This study pointed out the dependency of results to the freestream turbulence intensity, which is a difficult parameter to manage with LES. Finally, structured and unstructured flow solvers face to same difficulties but they predict a different behaviour of the boundary layer in this test case (natural or by-passed transition), depending on the external turbulence intensity.

NOMENCLATURE

- C Blade chord (C = 67.647mm),
- H Heat transfer coefficient,
- *M* Mach number,
- P Pressure,
- q Heat flux,
- Re Reynolds number,
- S Curvilinear abscissa,
- T Temperature,
- *Tu* Turbulence level,
- u_i Velocity component in direction i,
- V_x Streamwise velocity component,
- ρ Density,
- *i* Total value,

^{*}Address all correspondence to this author.

- is Isentropic value,
- s Static value,
- 0 Inlet value,
- 2 Outlet value,
- + Wall unit,
- LES Large Eddy Simulation,
- RANS Reynolds Averaged Navier-Stokes,
- SGS Sub-Grid-Scale,
- SMB Multi-block structured (flow solver),
- UNS Unstructured (flow solver).

INTRODUCTION

The transfer of thermal energy between a flow and a wall occurs in a lot of industrial applications (electronic circuit boards, gas turbine, etc.). This effect is often responsible for the reduction of the life duration and efficiency of components. The prediction of such wall heat transfers 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 temperature at the inlet of the high pressure turbine impacts directly the whole system performance. It is thus necessary to maximize this temperature to design an efficient propulsive system. The consequence is that the high pressure turbine blades experiences high temperature gradients at walls and their life duration directly rely on the capacity of designers to correctly estimate the wall heat transfer [1]. Unfortunately, this parameter is difficult to predict in such agressive environment and technological devices (such as cooling holes, tip gap, etc.) also largely impact the turbine flow (and the wall temperature). Finally, turbulence plays a major role on heat transfer and a laminar to turbulent transition is often observed on the turbine blade walls. This kind of phenomenon is complex to predict and depends on many parameter such as Reynolds number, turbulence intensity, wall roughness, shock, etc. A correct estimation of the wall heat transfer is thus out of range with classical steady state numerical simulations in most gas turbine configurations, even if no coupling is observed between thermal and dynamic boundary layers (conjugate heat transfer).

The physical understanding of such complex flow and the capacity to efficiently predict heat transfer at the design stage is thus mandatory to improve the efficiency of industrial systems such as gas turbine. 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 these 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 since the main advantage of this approach is its capacity to properly compute boundary layers. While effective, this method also suffers of major drawbacks such as the meshing of complex technological effects and the difficulty to refine localized regions. A potential answer is the use of unstructured grids that represent a promising way for local mesh refinement and taking into account complex technological devices (cooling holes, etc.) [7, 10].

Based on this state-of-the-art, this paper proposes to investigate the prediction of heat transfers in a highly loaded turbine guide vane, by means of structured and unstructured flow solvers. The test case is a well documented configuration [11] representative of a modern turbine blade and extensively used for CFD validation [3, 12]. The first section of the paper details the main parameters of this configuration. The guide vane operates at a high Reynolds number and transonic conditions, meaning it is a challenging case for numerical flow solvers. A quite important aspect resulting from the comparison against experiments lays in the capability of numerical simulation to predict the correct onset of the laminar to turbulent transition. Different numerical methods are tested in this paper to underline the capacity of current and more advanced approaches (RANS, LES, etc.) to estimate the complex flow features that develops in this turbine guide vane. These methods and the approaches followed with both structured and unstructured codes are presented in a second section of the paper. Results are then compared with experiments for the estimation of the heat transfer coefficient along the blade chord in section three. More specifically, comparisons are done to estimate the capacity of numerical methods to predict the wall heat fluxes at different freestream turbulence intensities. Finally, the results obtained with structured and unstructured approaches are discussed in a fourth section, with a particular interest for unsteady flows and transition.

EXPERIMENTAL CONFIGURATION

The experimental facility is the isentropic light piston compression tube located at the Von Karman Institute (Fig.1). The light weight piston is driven by the air of a high pressure tank in order to achieve the desired freestream gas conditions in terms of Mach and Reynolds numbers. More information about this facility is available in Consigny and Richards [13]. The tested configuration is a 2D turbine blade cascade largely described by



Figure 1. EXPERIMENTAL FACILITY AND CONFIGURATION INVES-TIGATED BY ARTS ET AL. [11].

Arts et al. [11]. The design of the high pressure nozzle guide vane (the so-called LS89 blade) was optimized for a downstream isentropic Mach number equal to 0.9 and the vane is mounted in a linear cascade made of five profiles (only the central passage is investigated to ensure periodic flow conditions). The blade chord C is 67.647 mm with a pitch/chord ratio of 0.85 and a staggered angle γ equal to 55°. Experimental investigations were done to measure the blade velocity distribution by means of static pressure tappings, convective heat transfer by means of platinum thin films, downstream loss coefficient and exit flow angle by using a fast traversing mechanism (based on a Pitot probe). Uncertainties were quantified for all these measurements (pressure: $\pm 0.5\%$, integrated loss coefficient: ± 0.2 points, exit flow angle: ± 0.5 degrees, heat transfer coefficient: $\pm 5\%$). Measurements of the freestream turbulence intensity and spectrum are also available (the freestream turbulence is generated by a grid of spanwise oriented parallel bars located upstream the guide vane). A large range of freestream conditions have been experimentally investigated. More specifically, two configurations are explored in this paper, as shown in Table 1 (test cases MUR129 and MUR235). The considered Reynolds number is about 10^6 (based on the chord and outlet velocity), corresponding to an outlet Mach number of 0.9. The most important difference between the two configurations is the inlet turbulence intensity ($Tu_{MUR129} = 1\%$ and $Tu_{MUR235} = 6\%$).

Local wall convective heat fluxes are of primary importance for this study. The measurement technique is based on information provided by thin film gauges painted on the cascade central blade (made of glass ceramic) and an electrical analogy (Schultz and Jones [14]). The thin films used for measurements are located in the "clean" region (the time-averaged flow is 2D) and

Table 1. DETAILS OF THE EXPERIMENTAL CONDITIONS FOR THE INVESTIGATED CONFIGURATIONS.

Test case	Re_2	$P_{i,0}$	$T_{i,0}$	$T_{s,wall}$	Tu_0
MUR129	1.1310^{6}	1.87 10 ⁵ Pa	409 K	298 K	1.0%
MUR235	1.1510^{6}	1.85 10 ⁵ Pa	413 K	301 K	6.0%

they cover 20% of the blade span (*i.e* 20mm). The convective heat transfer coefficient H is then defined as the ratio between the wall heat flux and the difference between the total freestream and the local wall temperatures:

$$H = \frac{q_{wall}}{T_0 - T_{wall}} \tag{1}$$

Thanks to the experimental investigations led by Arts *et al.* [11], most flow features have been clearly identified in this configuration. At the investigated Mach and Reynolds numbers $(M_{2,is} = 0.9, Re_2 = 10^6)$, the heat transfer coefficient is largely affected by the inlet turbulence intensity. For the lowest value of the freestream turbulence intensity ($Tu_0 = 1\%$), the heat transfer falls quickly after the leading edge, corresponding to the development of a laminar boundary layer both on pressure and suction sides. A transition is only observed close to the trailing edge on the suction side around S = 70 mm. For a higher level of the turbulent intensity ($Tu_0 = 6\%$), the onset of transition is observed earlier along the suction side (S = 25 mm) and thus the level of the guide vane heating is significantly increased from this point. It is expected that these flow features will be challenging for flow solvers, mainly due to the transition phenomenon.

TOOLS AND METHOD Governing equations

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:

$$\frac{dW}{dt} + divF = 0 \tag{2}$$

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.

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 $E(\mathbf{x},t) \equiv e_s + 1/2 u_i u_i$. The fluid follows the ideal gas law, $p = \rho r T$ and $e_s = \int_0^T C_p dT - p/\rho$, where e_s is the sensible energy, *T* the temperature, C_p the fluid heat capacity at constant pressure and *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.

Simulating directly turbine flows by means of the fully compressible three-dimensional Navier-Stokes equations is still unpractical mainly because of the flow Reynolds number which implies that all the flow scales be represented by Direct Numerical Simulation (DNS). Such a DNS requirement clearly yields unfeasable computations (the size of the mesh should be around $Re^{\frac{9}{4}}$). 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, 15]. At the same time, super-computers have reached performace and memory increases which allow to consider full three-dimensional simulations of real flow in experimental and industrial configurations [16]. The most common approach for complex configurations is still the Reynolds-Averaged Navier Stokes methods (RANS) that propose to model all the turbulent scales. This approach can be used to obtain either a steady-state flow (RANS) or an unsteady flow that contains the deterministic flow scales (unsteady RANS), such as rotor-stator interactions or vortex shedding. In these cases, the transition from laminar to turbulent boundary layers require to use transition criteria [3,6]. A more universal method is the Large-Eddy Simulation (LES) that introduces a separation between the resolved (large) turbulent scales and the modelled (small) scales [17]. This separation of scales is explicitely or implicitely 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 socalled Sub-Grid-Scale (SGS) model [2, 18]. LES involves the spatial Favre filtering operation that reduces for spatially, temporally invariant and localised filter functions [19] to:

$$\widetilde{f(\mathbf{x},t)} = \frac{1}{\overline{\rho(\mathbf{x},t)}} \int_{-\infty}^{+\infty} \rho(\mathbf{x}',t) f(\mathbf{x}',t) G(\mathbf{x}'-\mathbf{x}) d\mathbf{x}', \quad (3)$$

where G denotes the filter function.

The unresolved SGS stress tensor $\overline{\tau_{ij}}^t$ is modelled using the Boussinesq assumption [20]:

$$\overline{\tau_{ij}}^{t} - \frac{1}{3} \,\overline{\tau_{kk}}^{t} \,\delta_{ij} = -2\,\overline{\rho}\,\nu_t\,\widetilde{S}_{ij}\,, \qquad (4)$$

with
$$\widetilde{S}_{ij} = \frac{1}{2} \left(\frac{\partial \widetilde{u}_i}{\partial x_j} + \frac{\partial \widetilde{u}_j}{\partial x_i} \right) - \frac{1}{3} \frac{\partial \widetilde{u}_k}{\partial x_k} \,\delta_{ij}.$$
 (5)

In Eq. (4), \tilde{S}_{ij} is the resolved strain rate tensor and v_t is the SGS turbulent viscosity. The SGS energy flux \bar{q}_i^t is modelled using a SGS turbulent heat conductivity obtained from v_t by $\lambda_t = \bar{\rho} v_t C_p / Pr_t$ where $Pr_t = 0.7$ is a constant turbulent Prandtl number:

$$\overline{q_i}^t = -\lambda_t \frac{\partial \widetilde{T}}{\partial x_i}.$$
(6)

In Eq. (6), \tilde{T} is the Favre filtered temperature which satisfies the modified filtered state equation $\bar{p} = \bar{p} r \tilde{T}$ [21–24].

The LES method proposes to improve the level of flow description, as already shown by Duchaine et al. [7] for conjugate heat transfer prediction in a turbine blade. However the capacity of LES to describe laminar to turbulent transition in complex configurations such as the LS 89 blade is not well established. While unsteady flows (including turbulence) are clearly critical for heat transfer predictions, the need for LES still requires to be demonstrated, mainly because the computational effort is largely increased with respect to classical (U)RANS methods. In the present paper, two strategies are considered to simulate turbulent flows in the turbine guide vane: either an implicit flow solver that requires structured multi-block (SMB) meshes or an explicit flow solver that uses unstructured (UNS) meshes. On the one hand, an implicit SMB approach allows to finely mesh the boundary layers without major restriction regarding the definition of the time step. However the wall boundary fine grid topology is propagating far into the computational domain. On the other hand, unstructured (or hybrid) grids are more flexible and the grid refinements can be concentrated in the region of interest where strong gradients are expected in the flow. The difficulty with an explicit scheme is that stability criterion (CFL < 1) imposes more stringent conditions on the definition of the time step. As a consequence, when the Reynolds number is high (this is the case of the studied configuration), mesh requirements imply very small time-steps to compute the viscous sub-layer (i.e. the computational cost increases drastically).

Strategy with a structured multi-block flow solver

The *elsA* software uses a cell centered approach on SMB meshes. More information about this flow solver can be found in Cambier and Veuillot [25]. For this application, convective fluxes are computed with a third-order AUSM scheme considering a minimal artificial dissipation [26] and diffusive fluxes are calculated with a second-order centered scheme. For steady-state RANS simulations, the pseudo time-marching is performed by using an efficient implicit time integration scheme, based on the backward Euler scheme and a scalar Lower-Upper (LU) Symmetric Successive Over-Relaxation (SSOR) method as proposed by Yoon and Jameson [27]. To obtain a time consistent solution (for unsteady RANS and LES), the time-marching method

is a 4 step Runge-Kutta scheme coupled with an implicit residual smoothing (IRS) stage as proposed by Lerat *et al.* [28] (this method ensures a second order accuracy). For unsteady flow simulations, about 50,000 iterations are used to discretize one flow through time (about 0.5 ms), which is equivalent to 2,000 time steps to describe the vortex shedding frequency of the LS 89 blade. For (U)RANS simulations, the turbulent viscosity is computed with the two equations model of Wilcox [29] based on a k- ω formulation (the flow is assumed to be fully turbulent). For LES, the subgrid scale model is the Wall-Adapting Local Eddy-Viscosity (Wale) model [30], specially built to compute the turbulence effects near walls.

The flow domain is discretized with a multi-block approach, using an O-4H meshing strategy for the guide vane passage. A view of the computational domain is presented in Fig. 2(a). The mesh extends up to one axial chord uptream and two axial chords dowstream the blade (in order to limit the dependancy of the solution to the inlet/outlet positions). The mesh represents exactly 10% of the blade span (i.e. 10 mm in the spanwise direction). Typical grid dimensions are 651 points in the streamwise direction (781 points around the blade), 175 points in the pitchwise direction and 101 points in the spanwise direction. Experiments indicate that the mean flow is 2D. The number of points in the spanwise direction has thus been reduced to 5 for (U)RANS simulations, since the turbulence is only taken into account with the model. A low-Reynolds method is applied for the near wall mesh, and at least 30 points are used in the boundary layer in the wall-normal direction. Since the numerical method considers implicit schemes, the minimum cell size can be set to less than 2 μ m all around the blade (corresponding to a mean wall distance y^+ of 1) with an expansion ratio near the wall around 1.05. Figure 3 presents the evolution of the y^+ parameter around the blade. It shows that the maximum value of y^+ is always below 2 and is reached close to the trailing edge. In other directions, normalized wall distances are kept under acceptable values ($\Delta z^+ = 50$, $\Delta x^+ = 10 - 150$ with a mean value $\overline{\Delta x^+} = 100$). This mesh is thus well adapted to compute boundary layers without any wall function. Based on this meshing strategy, the blade passage is represented with a 14.4×10^6 points grid (for LES) and a 0.7×10^6 points grid (for (U)RANS).

Strategy with an unstructured flow solver

The parallel LES code AVBP [31, 32] solves the full compressible Navier-Stokes equations using a finite volume cellvertex and residual distribution formulation of the Lax-Wendroff scheme [33, 34]. This explicit scheme, which provides secondorder accuracy on hybrid meshes, is particularly adequate for low-dissipation requirements of LES applications [35]. In the present case, about 20,000 iterations are used to discretize one flow through time (*i.e.* 0.5 ms). Boundary conditions are handled with the Navier-Stokes Characteristic Boundary Con-



Figure 2. COMPUTATIONAL DOMAIN AND MESHES FOR THE FLOW SIMULATION OF THE LS89 BLADE - (A) STRUCTURED GRID, (B) UN-STRUCTURED GRID.

dition (NSCBC) formulation [17]. The dynamic Smagorinsky model [36] is chosen to model the SGS viscosity:

$$\mathbf{v}_t = (C_{S_D} \Delta)^2 \sqrt{2\widetilde{S}_{ij}\widetilde{S}_{ij}}.$$
(7)

In Eq. (7), Δ denotes the filter characteristic length (approximated by the cubic-root of the cell volume) and C_{S_D} is the closure coefficient obtained from the Germano inequality following Lilly's procedure [37]:

$$C_{S_D}^2 = \frac{1}{2} \frac{M_{ij} M_{ij}}{L_{ij} L_{ij}}.$$
 (8)

Copyright © 2010 by ASME



Figure 3. EVOLUTION OF THE NORMALIZED WALL DISTANCE Y^+ - SMB FLOW SOLVER (*elsA*).

In the expression Eq. 8, the tensors M_{ij} and L_{ij} are defined by:

$$M_{ij} = 2\hat{\Delta}^2 \sqrt{2 < \widetilde{S}_{ij} > < \widetilde{S}_{ij} >} < \widetilde{S}_{ij} >, \qquad (9)$$

$$L_{ij} = \langle \widetilde{u}_i \rangle \langle \widetilde{u}_j \rangle - \langle \widetilde{u}_i \widetilde{u}_j \rangle, \tag{10}$$

and introduce the notion of "test" filter of characteristic length $\hat{\Delta}$ equal to the cubic root of the volume defined by all the cells surrounding the cell of interest. Note that clipping and smoothing ensures none negative values for C_{S_D} .

The dynamic Smagorinsky subgrid model is used in conjunction with isothermal wall law conditions using NSCBC formalism [38]. Total pressure and total temperature with velocity angle are imposed using NSCBC formalism at the inlets of the fluid domain. Static pressures are enforced at outlet boundaries in characteristic NSCBC form. The computational domain extends up to one axial chord uptream and two axial chords dowstream the blade. Only one blade is represented in the domain with a periodicity condition. As with the SMB approach, only 10 mm of the blade span are represented with periodicity enforced on each side. This last simplification neglects end-wall effects but retains the three-dimensionality of the flow and greatly reduces the number of cells required to model the blade.

To reach minimum cell size close to $40 \,\mu\text{m}$ at the wall, about 13.6 million tetrahedral cells are required. In order to reduce the number of cells within the wall region, an hybrid grid strategy



Figure 4. EVOLUTION OF THE NORMALIZED WALL DISTANCE Y^+ - UNS FLOW SOLVER (AVBP).

allows mesh adaptation in wall regions: with prism layers at the wall and tetrahedra in the domain. Indeed, for a same spatial resolution in the normal direction, the prism layer approach uses less elements and leads to a higher minimum cell volume than the full tetrahedral grid approach because prismatic elements can have a large aspect ratio. Thus, the near-wall region is meshed using five layers of prismatic elements where the height of the layer, Δh , is smaller than the size of their triangle basis, Δx . To avoid numerical errors in the prism layers, the aspect ratio of the thinnest layer (inner layer adjacent to the wall) has been limited to $\Delta x/\Delta h = 3$ so that $x^+ \approx z^+ \approx 3y^+$. The streching ratio between the height of an inner layer to the adjacent outer layer has been set to 1.09. Using this mesh constraint in the near-wall region and a minimum cell size close to $40 \,\mu$ m at the wall leads to a 3.6 million cell grid (about 1200 points around the blade are necessary to ensure such mesh constraints). Improvement of the resolution results for a minimum cell size set to 20 μ m on the suction side and 40 μ m on the pressure side. The resulting mesh is composed of about 8 million cells (Fig. 2(b)). Figure 4 shows the evolution of the y^+ parameter around the blade. It indicates that the mean value of y^+ is around 20 and its maximum value is 50 on the pressure side close to the trailing edge.

Boundary conditions

An overview of the boundary conditions used with both numerical methods is shown on Table 2. A particular attention has been brought to the definition of flow conditions to ensure a similar behaviour with both flow solvers. An injection condition is applied at the inlet with identical parameters (based on experimental data) and a static pressure is used downstream to set the isentropic Mach number. An isothermal wall condition is applied at the blade walls with a uniform temperature $(T_{wall} = 301 K)$. However, a few differences exist between both flow solvers since strictly identical boundary conditions are not always available. For example, for radial sections ("top" and "bottom" of the numerical domain), a symmetry condition is used with the SMB flow solver while a periodicity condition is imposed with the UNS flow solver. Another difference comes from the capacity to define the inlet turbulence level Tu_0 . With the SMB code (elsA), a broadband noise is injected at the inlet to set the turbulence level. However, it is well know that this approach does not respect the true nature of turbulence since it does not take into account any turbulent correlation and the energy is distributed along a wide range of frequencies. Indeed, most fluctuations are rapidly damped by the simulation before to reach the blade leading edge. For the UNS flow solver (AVBP), no condition is available (at the moment) for this parameter and the turbulence naturally develops in the numerical domain (due to numerical errors, etc.).

The evolution of the isentropic Mach number is plotted Fig. 5 with respect to the curvilinear abscissa S (S = 0 corresponds to the blade leading edge) to ensure that numerical approaches correctly compute the experimental flow operating conditions. Results indicate that both methods reproduce well experimental aerodynamic conditions on the pressure side and that only small discrepancies are observed on the suction side around S = 60 - 75 mm. Moreover, the small plateau experimentally pointed out on the suction side around S = 25 mm is well predicted by both numerical methods. A small shock is found on the suction side near the trailing edge, around the position S = 60 - 70 mm (this flow feature is also observed experimentally thanks to Schlieren pictures). As a consequence, this comparison indicates that both numerical methods proposed in this paper are able to correctly compute the mean aerodynamic flow that is a pre-requisite for the prediction of the aero-thermal behaviour.

RESULTS

As already mentioned, different methods exist to simulate the flow in a high-pressure turbine guide vane. The objective of this paper is to show an overview of the capacity of these numerical methods to predict the wall heat transfer coefficient H in a turbine guide vane configuration. Indeed, results are presented and discussed for steady state methods (laminar and RANS simulations) and unsteady flow methods (unsteady RANS and LES). All numerical simulations are performed on a SGI Altix computing platform and results are compared to experimental values obtained for two inlet turbulence intensities ($Tu_0 = 1\%$ and $Tu_0 = 6\%$). All comparisons are based on figures that show the evolution of H with respect to the curvilinear abscissa S (positive values of S corresponds to the suction side). Table 2. DETAILS OF THE NUMERICAL PARAMETERS USED WITH SMB AND UNS APPROACHES.

	SMB (elsA)	UNS (AVBP)	Value
Inlet	injection	injection	$P_i = 1.85 10^5 Pa$
	+ noise		$T_i = 413 K$
			$\alpha = 0^{o}$
Outlet	pressure	pressure	$P_s = 1.06 10^5 Pa$
			$(\approx M_{is}=0.92)$
Walls	isotherm	isotherm	$T_{wall} = 301 K$
	+ no-slip	+ wall-function	
Azimuth	periodicity	periodicity	-
Radial	symmetry	periodicity	-



Figure 5. TIME-AVERAGED ISENTROPIC MACH DISTRIBUTION ALONG THE BLADE WALL.

Laminar simulation

As a first approximation, the flow can be assumed to be in a purely laminar state (including boundary layers). This assumption is not far to reality when considering the small value of the inlet turbulence level. Indeed, this simulation is only representative of the case $Tu_0 = 1\%$. One simulation of the flow with this method requires only 20 CPU hours. Results are indicated on Fig. 6 for the prediction of the heat transfer coefficient H. As expected, the wall heat transfer is globally well estimated by the laminar simulation when comparing with experiments at



Figure 6. HEAT TRANSFER COEFFICIENT H PREDICTED BY A LAM-INAR SIMULATION (SMB -*elsA*-).

 $Tu_0 = 1\%$. On the pressure side, discrepancies are less than 5% (which is equivalent to the experimental uncertainty). When comparing results on the suction side, results agree well until S = 40 mm. After this point, the laminar simulation tends to underestimate the value of H (by 30%), indicating that transition of the flow experimentally begins at this location. After S = 75 mm, experimental data indicate a fully developed turbulent boundary layer and the simulation completely failed to predict the correct level of H (error is more than 70%). The experimental data registered for the inlet turbulence intensity $Tu_0 = 6\%$ are also inticated in Fig. 6, showing that the laminar simulation fails to estimate the correct level of H at these flow conditions, including on the pressure side.

Steady state RANS simulation

Another approximation is to consider a fully turbulent flow in a steady state. It is thus expected that a RANS simulation is more representative of the case $Tu_0 = 6\%$ (this value of Tu is imposed to define the inlet turbulence level). As for laminar simulations, 20 CPU hours are required to reach convergence. Results are shown on Fig. 7. The RANS simulation underpredicts the value of H near the leading edge (by about 10%). Close to the leading edge ($S = \pm 5$ mm), the predicted level tends to be coherent with experimental observations. However, the value of H increases rapidly after these locations and the simulation finally overestimates the heat transfer by more than 200% both on the pressure and suction sides (before the transition point). Once the experimental boundary layer is fully turbulent (S > 65 mm), the simulation predicts a similar magnitude order of the guide vane heating ($H \approx 800 W.m^{-2}.K^{-1}$). This result is well coherent with



Figure 7. HEAT TRANSFER COEFFICIENT H PREDICTED WITH A STEADY RANS SIMULATION (SMB -*elsA*-).

other numerical works that consider a RANS method [3]. A way to improve the result quality is to couple RANS simulations with transition criteria [3, 6]. Nevertheless, this complex subject requires a full study and is thus not investigated in this paper.

Unsteady RANS simulation

The next step is to assess the role of unsteady flows on the prediction of heat transfer. Starting from RANS simulations, the most natural method is to use an unsteady RANS approach, that solves the deterministic part of the flow unsteadiness (vortex shedding, etc.). In the LS 89 configuration, most unsteady flows are generated in the guide vane trailing edge region. Indeed, the vortex shedding needs to be properly simulated to capture the unsteady flow features and their influence on the blade boundary layers. As with the RANS method, the inlet turbulence intensity is set to Tu = 6%. The simulation cost is 250 CPU hours to obtain a periodic state solution. Time-averaged results obtained for the heat transfer coefficient H are plot in Fig. 8 and compared with experimental data. As already mentioned, the full turbulent assumption leads to an overestimation of the blade heating both on pressure and suction sides. However, important differences are observed with the steady state solution. First the begining of the heat transfer jump shown on the pressure side is shifted from S = -5 mm (RANS) to S = -25 mm (URANS). The value of H close to the trailing edge predicted by the unsteady RANS method tends also to be lower ($H \approx 650 \text{ W}.m^{-2}.K^{-1}$ instead of 800 $W.m^{-2}.K^{-1}$). Then the value of H on the suction side is largely reduced, especially in the region 40 mm < S < 60 mm where H drops from 850 $W.m^{-2}.K^{-1}$ to 500 $W.m^{-2}.K^{-1}$. This observation points out that unsteady flows (generated in the trail-



Figure 8. HEAT TRANSFER COEFFICIENT H PREDICTED WITH AN UNSTEADY RANS SIMULATION (SMB -*elsA*-).

ing edge region in the present case) have a significant impact on the local heat transfer. It is thus mandatory to correctly predict these flows in order to better estimate the value of H.

LES method

As mentioned, a promising method to describe unsteady flows (including turbulent flows) is the LES approach. The (assumed) most energetic turbulent scales are directly simulated, leading to a better description of the flow unsteadiness. A converged solution (from the statistical point of view) of the flow in the LS 89 blade requires 12,000 CPU hours with the SMB flow solver (corresponding to 10 flow through times) and 11,000 CPU hours with the UNS flow solver (corresponding to 20 flow through times). In this context, parallel computation greatly helps to reduce the simulation time [16] (32 to 128 computing cores have been used for LES). Time-averaged solutions obtained with both SMB and UNS flow solvers are compared with experiments on Fig. 9(a) and (b). Part (a) of Fig. 9 gives an overview of the evolution of the heat transfer coefficient along the whole blade chord while part (b) is a close view around the leading edge. As mentioned, both flow solvers do not consider the same method to set the turbulence level at the inlet. With the SMB code, despite the fact that $Tu_0 = 6\%$ is imposed at the inlet, results scale better with the experimental case considering $Tu_0 = 1\%$. As already explained, the reason is that the broadband noise imposed at the inlet is not an efficient strategy since most turbulent fluctuations are damped, due to numerical viscosity (most numerical schemes require 10 points to correctly transport a wavelength). It has been checked in the simulation with the SMB code that the freestream turbulence intensity close to the the blade leading edge is less than 1%. For the UNS code, the problem is that no turbulence is imposed at the inlet. Indeed, turbulence naturally occurs since the numerical scheme used is associated to a very low numerical dissipation. The result is that the turbulence level evolves along the blade chord. Close to the trailing edge, the freestream turbulence intensity is about 5%. Hence this case scales better with the experimental case at $Tu_0 = 6\%$.

Despite these difficulties, results are encourageous since both approaches show that the global shape of the experimental curves are quite well reproduced. When comparing with experimental data at $Tu_0 = 1\%$, the SMB approach shows its ability to estimate precisely the value of H close to the leading edge (relative error is less than 5%). At the position S = 5 - 15 mm (suction side), experiments indicate the development of a plateau (H is constant) that is also well predicted by the simulation (Fig. 9(b)). The simulation predicts a purely laminar boundary layer on the pressure side (as experimentally observed) and a laminar to turbulent transition occurs on the suction side close to the position S = 70 mm. The position of the boundary layer transition and the heat transfer coefficient after the transition are thus correctly predicted by the simulation (the experimental position for transition is S = 65 mm). Finally, the simulation correctly predicts the value of H all around the blade with a precision around 5%.

The UNS flow solver also correctly reproduces a part of the experimental flow features at $Tu_0 = 6\%$. Since the numerical turbulence intensity varies along the blade chord, results are a bit complex to interprete. Close to the leading edge, the turbulence intensity is lower than 1% and a laminar boundary layer develops both on pressure and suction sides until $S = \pm 5$ mm. The turbulence level increases rapidly after S = -10 mm (pressure side), leading to an increase of the heat transfer coefficient (that perfectly scales with experiments at this location). The numerical turbulence level still increases close to the trailing edge and the value of H is slightly overestimated around S = -50 mm (by 7%) with respect to experimental data). Similar observations are done on the suction side. At S = 10 mm (suction side), the numerical turbulence intensity increases and the value of H matches the experimental data at $Tu_0 = 6\%$. The simulation predicts the begining of the transition at S = 30 mm, very close to the experimental value (error is only 10%). From S = 30 mm to S = 60 mm, the numerical data follow the experimental observations. However, once the boundary layer is fully turbulent, the simulation largely underestimates the value of H (by about 50%).

DISCUSSION Unsteady flow features

Previous results clearly underline that unsteady flows affect the development of boundary layers and thus the heat transfer coefficient. Only unsteady RANS and LES are able to reproduce a part of these flow features. A view of the instantaneous flow



Figure 9. HEAT TRANSFER COEFFICIENT H PREDICTED WITH LES (UNS -AVBP- and SMB -elsA-) (A) AND CLOSED VIEW AROUND THE LEADING EDGE (B).

around the LS 89 blade is shown in Fig. 10, that points out the main flow features that are computed with the LES method. The local flow acceleration issued by the suction side flow passage restriction is clearly visible and induces a region of density gradient in the blade passage. Figure 10 indicates that the flow is chocked at the most reduced section (the throat). Density gradients are also observed 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. The resulting vortex shedding develops and interacts with a weak shock about half a chord behind the trailing edge. One can observe that numerical dissipation plays a major role since the wake is largely damped about one chord behind the trailing edge. This observation is more pronounced when considering RANS methods than with LES. The vortex shedding is also a good candidate to underline the capacity of numerical methods to reproduce unsteady flows. A signal of axial velocity V_x is registered at a fourth of the blade chord



Figure 10. INSTANTANEOUS FLOW FIELD OF $grad\rho/\rho$ OBTAINED WITH LES (SMB CODE elsA).

behind the trailing edge, in the middle of the wake. Results are shown on Fig. 11 both for unsteady RANS and LES approaches. First, both methods indicate very large fluctuations since the fluctuating part V'_x is equivalent to 100% of the time-averaged value $\overline{V_x} \approx 100 \text{ m.s}^{-1}$. Negative values of the axial velocity are even observed in the wake region. Second, the difference betwen unsteady RANS and LES is well pointed out when looking at the signal shape in Fig. 11: URANS exhibits only one well identified frequency but LES clearly shows a more complex behaviour and larger frequency spectrum. The time-averaged value predicted with both methods is roughly identical. However, peak-to-peak fluctuations are more pronounced with LES.

While the flow features in the wake region have only a small influence on the boundary layers (and thus on the heat transfer coefficient), acoustic waves that are emitted at the blade trailing edge can potentially modify the boundary layer nature (the transition point). These waves are emitted at the same frequency that vortex shedding and impact the opposite blade on the suction side, potentially stimulating the boundary layer. Both methods are able to represent these flow features, but the LES clearly better estimates the complex interactions that occur between the boundary layer flow and the acoustic waves. Moreover, the effect of such waves is out of range for purely RANS simulations.

Laminar to turbulent transition

It is now established that unsteady flows phenomena can modify the position for the boundary layer transition. But the main criteria to estimate the nature of the transition is the turbulence intensity outside the boundary layer. In the case of low turbulence levels (Tu < 1%), a natural transition develops along the



Figure 11. COMPARISON BETWEEN UNSTEADY RANS AND LES - SIGNAL OF AXIAL VELOCITY IN THE WAKE REGION (SMB CODE elsA).

blade chord due to Tollmien-Schlichting waves. With a higher turbulence intensity, the transition proceeds more rapidly and the natural transition is by-passed [39]. It is exactly what is observed with the LES performed in the present paper. Figure 12 shows the instantaneous solution for the wall heat flux predicted with LES on the blade suction side. The nature of the boundary laver is well highlighted: close to the leading edge (x = 0 mm), SMB and UNS flow solvers both predict a laminar behaviour of the boundary layer. However, the transition process is clearly different. The external turbulence level is around Tu = 1% with the SMB approach. Indeed, as shown in Fig. 12(a), the boundary layer is perfectly laminar until x = 25 mm, that corresponds to the position where acoustic waves impact on the suction side. At this point, spanwise modes develop in the boundary layer and the heat flux is no longer uniform along the blade span. However, these modes tend to be damped after x = 30 mm and the laminar to turbulent transition is finally triggered by the shock at x = 35mm. With the UNS flow wolver, the turbulence intensity evolves along the chord to reach 5% close to the trailing edge (before the vortex shedding). As expected, the natural transition process is by-passed and turbulent spots are already observed at the position x = 5 mm (Fig. 12(b)). The boundary layer appears then to be fully turbulent along the blade suction side. The wall heat flux is clearly increased at x = 32 mm, corresponding to the shock position. The shock position is thus shifted upstream with respect to the results obtained with the SMB flow solver. This observation is coherent with the experimental results [11] that report this modification of the shock position with respect to the turbulence intensity. Finally close to the trailing edge, the boundary layer is fully developped in both cases. However, the SMB flow solver



Figure 12. INSTANTANEOUS SOLUTION OF THE WALL HEAT FLUX Q COMPUTED WITH LES ON THE SUCTION SIDE - (A) SMB FLOW SOLVER, (B) UNS FLOW SOLVER.

predicts better the experimental value of the heat flux close to the trailing edge ($Q_{exp.} \approx Q_{SMB} \approx 8 \ W.cm^{-2}$) than the UNS flow solver ($Q_{UNS} \approx 5 \ W.cm^{-2}$). The reason is that the boundary layer thickness is better estimated with a low-Reynolds method than with wall functions, especially close to the position x = 30 - 35mm where the boundary layer is significantly thickened due to the interaction with the shock.

CONCLUSION

The study presented in this paper has been performed to investigate the flow in a highly loaded turbine guide vane (the so-called LS 89 blade). This well documented configuration is a good candidate to assess the capacity of numerical methods to predict wall heat transfers in a complex test case exhibiting shock-boundary layer interactions and laminar to turbulent transition. Four kinds of methods have been applied: purely laminar, steady RANS, unsteady RANS and LES. First, the computational time T required by these methods to obtain a solution can be summarized as follow: $T_{LES} = 40.T_{URANS} = 500.T_{RANS} =$ 500.TLAMINAR. As expected, the cost associated to LES is significantly increased with respect to classical (U)RANS methods. The main reasons are the size of the mesh and the number of time steps required to describe a characteristic period of the flow. Even for a "2D" flow, the size of the domain is considerably increased due to the 3D nature of turbulence.

Comparisons of simulations with experiments indicate that the effectiveness of numerical methods depends on the freestream conditions (the turbulence intensity Tu in the present case). Two challenging flows are identified in the studied configuration. First the laminar to turbulent transition that occurs

on the blade suction side can be natural (Tollmien-Schlichting waves) or by-passed. It can also be triggered by the interaction between the shock and boundary layer. Then, unsteady flows at the trailing edge are responsible for the development of acoustic waves that impact the suction side. These waves interact with the boundary layer, leading to a local increase of the wall heat flux. These observations explain why the flow in this configuration is so difficult to simulate. For small values of $Tu \ (\approx 1\%)$, a simple laminar approache gives quite satisfactory results, even better than RANS methods. In fact steady (U)RANS simulations fail to estimate heat transfer in this configuration mainly because the transition phenomenon is not taken into account (discrepancies are around 200% at some locations). Previous studies have shown the interest of transition criteria for such turbine configurations [3, 6]. However a difficulty to apply these criteria is that they require a knowledge a priori of freestream conditions to determine the nature of transition (by-passed, natural, shock, etc.). It has been chosen in this paper to apply a more universal method, based on the LES approach.

The main interest for LES is its potential capacity to describe most kinds of transition and unsteady flows in a large range of industrial applications. Moreover, in complex and agressive environements (such as in a high-pressure turbine), freestream conditions are not always known, meaning the numerical method must be able to deal with this constraint to provide reliable information. Two flow solvers have been used to simulate the turbulent flow: one considering structured multiblock grids and one considering unstructured grids. While structured grids are well adapted to the simulation of boundary layers, the unstructured approach has the advantage of flexibility (especially if technological effects are considered, which is not the case here). In both cases, the application of LES gives very encouraging results regarding the prediction of the heat transfer coefficient. On the one hand, the value of the heat transfer coefficient is estimated with a good accuracy by the SMB flow solver for the case corresponding to $Tu_0 = 1\%$ (error is less than 5% all around the blade, including close to the trailing edge). On the other hand, the value of H and most flow phenomena are correctly computed with the UNS flow solver for the case corresponding to $Tu_0 = 6\%$ (error is only 5 to 10% at most locations). Thus LES demonstrates its capacity to compute unsteady flows and flow transition with a reasonably good accuracy. However, difficulties still exist. First, both flow solvers used in this paper are not able to properly set the inlet turbulence intensity Tu_0 (while this parameter is easy to define for RANS simulations). Since the flow is very sensitive to this parameter (and thus the heat transfer coefficient), it is mandatory for future investigations to implement an efficient strategy to define the inlet turbulence intensity (following the work of Jarrin et al. [40]). Another solution could be the simulation of the whole experimental system that generates the turbulence at the inlet. However, such a method will require the use of a very large mesh (probably dozens of million points). Then, the use of low dissipative numerical schemes is also necessary, especially if a description far to the walls is requested (for example to propagate a vortex shedding or turbulence at the inlet of the domain).

ACKNOWLEDGMENT

Many thanks to Tony Arts (VKI) for the effective cooperative work about the numerical simulation of flows in turbine configurations. The work presented in this paper has also largely benefit from CERFACS internal and GENCI-CINES computing facilities (under the project fac 6074). These supports are 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. Turbomach., 130(2).
- [6] Liu, Y., 2007. "Aerodynamics and heat transfer predictions in a highly loaded turbine blade". *Int. Journ. 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. Journ. 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. Journ. 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] Kwon, O. J., and Hah, C., 1995. "Simulation of three-dimensional turbulent flows on unstructured meshes". *AIAA Journal*, 33(6).
- [11] Arts, T., Lambert de Rouvroit, M., and Rutherford, A. W., 1990. Aero-thermal investigation of a highly loaded tran-

sonic linear turbine guide vane cascade. Technical Note 174, Von Karman Institute.

- [12] Gehrer, A., and Jericha, H., 1999. "External heat transfer predictions in a highly loaded transonic linear turbine guide vane cascade using an upwind based navier-stokes solver". *J. Turbomach.*, 121(3), pp. 525–531.
- [13] Consigny, H., and Richards, B., 1982. "Short duration measurements of heat transfer rate to a gas turbine rotor blade". *J. of Engineering for Power*, *104*, pp. 542–551.
- [14] Schultz, D. L., and Jones, T. V., 1973. Heat transfer measurements in short duration hypersonic facilities. Report 165, AGARD.
- [15] Pope, S. B., 2000. *Turbulent flows*. Cambridge University Press.
- [16] 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 Sciences and Discovery, 2(015004).
- [17] Poinsot, T., and Veynante, D., 2005. *Theoretical and Nu*merical Combustion. R.T. Edwards, 2nd edition.
- [18] Ferziger, J. H., 1977. "Large eddy simulations of turbulent flows". AIAA Journal, 15(9), pp. 1261–1267.
- [19] Vreman, B., Geurts, B., and H. Kuerten, H., 1994. "On the formulation of the dynamic mixed subgrid-scale model". *Phys. Fluids*, 6(12), pp. 4057–4059.
- [20] Smagorinsky, J., 1963. "General circulation experiments with the primitive equations: 1. the basic experiment.". *Mon. Weather Rev.*, 91, pp. 99–164.
- [21] Moin, P., Squires, K. D., Cabot, W., and Lee, S., 1991. "A dynamic subgrid-scale model for compressible turbulence and scalar transport". *Phys. Fluids*, A 3(11), pp. 2746– 2757.
- [22] Erlebacher, G., Hussaini, M. Y., Speziale, C. G., and Zang, T. A., 1992. "Toward the large-eddy simulation of compressible turbulent flows". *J. Fluid Mech.*, 238, pp. 155– 185.
- [23] Ducros, F., Comte, P., and Lesieur, M., 1996. "Large-eddy simulation of transition to turbulence in a boundary layer developing spatially over a flat plate". J. Fluid Mech., 326, pp. 1–36.
- [24] Comte, P., 1996. New tools in turbulence modelling. vortices in incompressible les and non-trivial geometries. Springer-Verlag, France. Course of Ecole de Physique des Houches.
- [25] 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.
- [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] Lerat, A., Sides, J., and Daru, V., 1982. "An implicit finitevolume method for solving the euler equations". In 8th International Conference on Numerical Methods in Fluid Dynamics, Vol. 170, pp. 341–349.
- [29] Wilcox, D., 1988. "Reassessment of the scale-determining equation for advanced turbulence models". *AIAA Journal*, 26, pp. 1299–1310.
- [30] 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.
- [31] 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.
- [32] Mendez, S., and Nicoud, F., 2008. "Large-eddy simulation of a bi-periodic turbulent flow with effusion". J. Fluid Mech., 598, pp. 27–65.
- [33] Lax, P. D., and Wendroff, B., 1960. "Systems of conservation laws". *Communication on Pure and Applied Mathematics*, 13(2), pp. 217–237.
- [34] Lax, P. D., and Wendroff, B., 1964. "Difference schemes for hyperbolic equations with high order of accuracy". *Communication on Pure and Applied Mathematics*, 17, pp. 381–392.
- [35] Colin, O., and Rudgyard, M., 2000. "Development of highorder taylor-galerkin schemes for unsteady calculations". *J. Comput. Phys.*, 162(2), pp. 338–371.
- [36] Germano, M., 1992. "Turbulence: the filtering approach". *J. Fluid Mech.*, 238, pp. 325–336.
- [37] Lilly, D. K., 1992. "A proposed modification of the germano sub-grid closure method". *Phys. Fluids*, 4(3), pp. 633–635.
- [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] Simon, T. W., and Kaszeta, R. W., 2006. "Transition to turbulence under low-pressure turbine conditions". *Annals* of the New York Academy of Sciences, 934, pp. 37–51.
- [40] Jarrin, N., Benhamadouche, S., Laurence, D., and Prosser, R., 2006. "A synthetic eddy method for generating inflow conditions for large eddy simulations". *International Journal of Heat and Fluid Flow*, 27(585-593).