A Comparative Study of RANS, URANS and NLES Approaches for Flow Prediction in Pin Fin Array

Proceedings of
10th European Conference on Turbomachinery Fluid dynamics & Thermodynamics
ETC10, April 15-19, 2013, Lappeenranta, Finland

A Comparative Study of RANS, URANS and NLES Approaches for Flow Prediction in Pin Fin Array

M. Carnevale[∗♯] - *S. Salvadori*[∗] - *M. Manna*[◊] - *F. Martelli*[∗]

[∗] Industrial Engineering Department
Universitá degli Studi di Firenze, via di S. Marta, 3 50139 Florence, Italy
m.carnevale@imperial.ac.uk

[◊]Dipartimento di Meccanica per l'Energetica
Universitá degli Studi di Napoli Federico II, via Claudio 21, 80125 Naples, Italy

[♯] Currently in VUTC Lab
Imperial College London, South Kensington Campus, London SW7 2AZ, UK

ABSTRACT

Gas Turbine are nowadays largely used for aircraft propulsion and land-based power generation. The increased attention to environmental aspects has promoted research and development efforts both from manufacturers and research centres. The latest developments in turbine-cooling technologies play a critical role in the attempt to increase the efficiency and the specific power of the most advanced designs. Pin fin arrays, in particular, are widely used in jet engine application because of their ability to enhance cooling by providing extended surfaces for conduction and convection. They are also known to be an effective means to create turbulence which naturally increases heat transfer. Pin fin turbulators are typically located inside the trailing edge of high pressure turbine blade where they also act as a structural support.

The optimum shapes and spacing of such elements are usually determined experimentally, or more recently, by using Computational Fluid Dynamics (CFD). On the other hand, the comprehension of the real physics controlling the heat transfer enhancement process and the role played by the large scale vortical structures generated by the inserts, still represent a great challenge for fluid mechanic researchers. The problem has been intensively investigated by Ames ***et al*****. (2005) by means of an experimental campaign on pin fin matrix. From the numerical point of view, the principal bottleneck of the CFD approach as applied to this kind of massively unsteady flow is related to the high computational cost and to the reliability of the turbulence models.**

The main objective of this work is to offer a critical analysis of the performance of a cooling device consisting of a pin fin turbulators geometry, as predicted by different CFD models of various complexity, using similar computational technology to integrate the corresponding governing equations. Local velocity and turbulence distributions are presented and compared with available experimental data.

NOMENCLATURE

C_p	Pressure coefficient distribution
$p_0\,(\theta = 0)$	Total pressure in θ_0 position
p	Static pressure
ρ	Density
T	Temperature
U	Streamwise velocity
θ	θ position on cylinder surface
V_b	Bulk gap flow velocity
Δt	Time step
Δt^*	Non-dimensional time step $\Delta t^* = \Delta t \times V_{max}/D$
E	Turbulent kinetic Energy
k	wavenumber
Nu	Nusselt number
Re	Reynolds number
St	Strouhal number
Pr	Prandtl number
D	Pin diameter
Lu	Upstram length
Ld	Downstram length
Lp	Distance of the periodic faces.

INTRODUCTION

Industrial requirement for computational modeling of turbomachinery flow is driven by the necessity for fast design methods. Up to now this has been done integrating 1D mean-line methods, 2D trough-flow codes and 3D steady/unsteady simulations.

3D Unsteady Reynolds-Averaged Navier-Stokes (*URANS*) simulations represent an efficient instrument of study for several problems and phenomenologies in the turbomachinery fields. Despite this, calculations of intrinsically unsteady phenomena performed by *RANS* and *URANS* typically lead to unsatisfactory results.

Moreover *URANS* approaches are characterized by limitations connected both to the prediction of flows in transitional regime, and to the capability of properly tackling large scale coherent structures. On the other hand Large Eddy Simulation (*LES*) is known able to satisfactory model these structures. In the open literature *LES* is usually applied to structured grids and mainly to simplified test-cases.
The goal of this work is to compare the capabilities of three different turbulence closure in predicting flow features related to a realistic configuration. *RANS*, *URANS* and *LES* approaches have been implemented in a in-house developed unstructured finite volume code which is capable to handle geometric configuration representative of realistic turbomachinery components.
From this point of view a pin fin matrix represents a challenge for any CFD solver since it involves complex flow structures occurring in complex geometries. Pin fin arrays are one of the most common internal cooling device used in the trailing edge region of high pressure turbine blades. They increase flow turbulence and internal heat transfer surface area. Arrays are mostly designed using empirical correlations to predict heat transfer rates and pressure drops. It's commonly accepted that computational models for heat transfer and velocity distribution predictions can be improved if the characteristics of turbulence and its response near surfaces such as pins and end-walls are better understood.
Ames *et al*. (2005) provided experimental data for such a kind of flows at different Reynolds numbers.

From the numerical point of view, the same geometry has been widely investigated by Delibra *et al*. (2009), Delibra *et al.*, 2010 (a) and Delibra *et al.*, 2010 (b), Carnevale *et al*. (2012) . Carnevale *et al*. (2012) also investigated the effects of numerical uncertainties on the evaluation of the aero-thermal parameters using *LES*. According to the authors, steady *RANS* calculations obtained with eddy viscosity models coupled with transition models, experience difficulties in predicting the time-averaged characteristics of even basic unsteady transitional flows.

The *URANS* approach slightly improves the prediction of these flows, while more sophisticated closures are required for accurate aero-thermal characterization of separated transitional flows. Thus unsteady approaches are necessary. The deficiencies of *URANS* approaches in responding to weak instabilities and in capturing the break-up of the initially two-dimensional structures and the transition are clearly put forward.

HYBFLOW CODE: NUMERICAL FEATURES

Simulations have been carried out using hybFlow, *URANS* solver HybFlow, which has been developed at the Energy Engineering Department of the University of Florence. The code has been extensively validated for heat transfer problems and it has also been used in *DES* and *LES* simulations, with the Smagorinsky sub-grid model (Adami *et al*. (2000), Montomoli *et al*. (2011), Bernardini *et al*. (2011)).

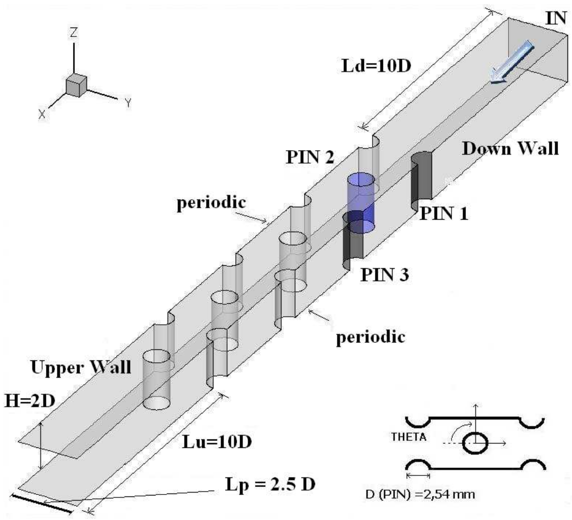
The solver is based on 3D unstructured finite-volume formulation, which is kept in the *LES* approach in order to face complex geometries, and it is capable to handle unsteady multi-stage simulations. Details about the spatial discretization cans be found in the cited references. Due to the critical dependency of numerical dissipation from the numerical scheme, the flux calculations at cells interface in the low Mach regions have been modified according to the preconditioning method suggested by Thornber *et al*. (2008). The steady solver is based on an iterative implicit time-marching solution. The implicit iterative time-relaxed Newton method is applied along with the linear solver *GMRES* coupled to an incomplete *ILU(0)* factorization as suggested in Saad *et al*. (1986). This formulation has been used in the *RANS* case. The implicit formulation with a dual-time stepping approach is applied to get the convergence to the physical unsteady time solution from the implicit steady solver. This kind of formulation allows a quite arbitrarily choice of the time step, not yielding any constraint on the Courant (*CFL*) number. Time integration can also be performed explicitly either with a 2-stages predictor corrector or with a 6-stages 4th order Runge Kutta scheme. While the former is due to Heen the latter is an an extension of the low dissipation and dispersion explicit Runge-Kutta scheme by Calvo *et al*. (2004).

For the *RANS* simulation, turbulence is modelled using the classical eddy-viscosity assumption through the two equation $k - \omega$ *SST* model proposed by Menter (1992) with the transition model of Menter *et al*. (2006). The transition model has been validated by Salvadori *et al*. (2009) in heat transfer problems. Concerning the computational capabilities, the parallel solver balances the computational load partitioning the grid in blocks that are evenly distributed over to the CPUs. The communications between processors are managed by the standard *MPI* message passing libraries. Accurate studies have been performed to establish the code scalability and performance when increasing the number of CPUs (Belardini *et al*. (2001)).

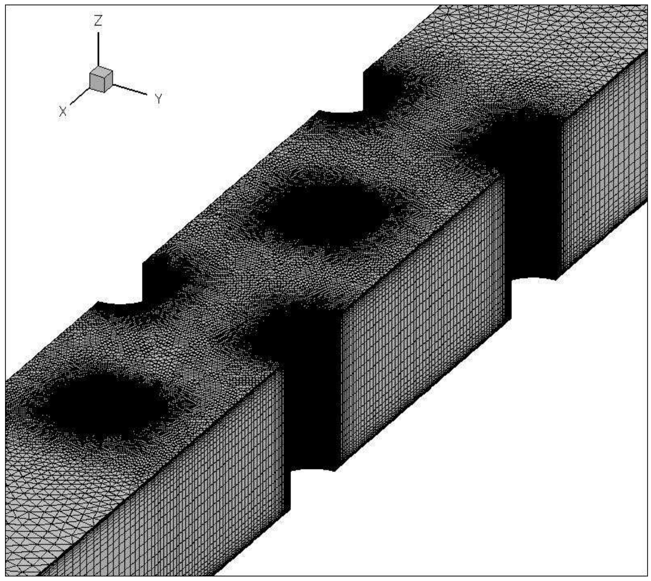
TEST-CASE AND BOUNDARY CONDITION

The computational domain representing the experimental configuration detailed in Ames *et al*. (2005) is shown in Figure 1a. The diameter of the pins is $D = 2.54cm$, the distance between two parallel array is $Lp = 2.5D$ (the distance is calculate considering the center of two adjacent pin. The channel height is twice the diameter ($H = 2D$). Bottom and top walls, as well as pins surfaces are isothermal solid walls. The inlet section of the channel is located at a distance $Lu = 10D$ upstream

the first pin row. The same distance Ld has been considered for the downstream boundary.
Numerical simulations have been carried out applying different turbulence closures. More specifically the algebraic Reynolds stress $k-\omega$ *SST* model of Menter (1992) together with the transition model of Menter *et al*. (2006) is applied in *RANS* and *URANS* formulations. These results are compared with those obtained by means of the Numerical LES (*NLES*).
It is well known that for industry-oriented applications unstructured solvers are necessary to tackle complex geometries. However, fully unstructured codes tend to be more dissipative compared with the corresponding structured formulation with equal order of the variable reconstruction. This is mostly due to difficulties associated with tetrahedral element shape distortion in the wall region and the impossibility in preserving the grid quality during the grid generation process. These shortcomings could be exploited in a *LES* approach. Tucker (2011) recently suggested adopting a *NLES* formulation, which consists in omitting the sub grid-scale model at all, responsible for further dissipation compared to the one coming from the discretization. In fact under many circumstances the effects of the sub-grid scale model, particularly considering the Smagorinsky one, may be negligible. Also Garnier *et al*. (1999) recommended the same approach and suggested a *NLES* approach without sub-grid scale model if the approximate Riemann solver of Roe is used in conjunction with a linear variable reconstruction (second order scheme). This approach is followed herein. Moreover preliminary tests adopting the Smagorinsky sub-grid model in the HybFlow code, gave disappointing results, due to the excessive numerical dissipation of the unstructured solver. For all of these reasons the *NLES* approach has been selected for the present analysis.
Simulations have been performed at a Reynolds number based on pin diameter D and gap bulk velocity V_b of $Re = 10000$. According to the experimental data, the gap bulk velocity is computed averaging the local velocities between two adjacent pins of the same row. Taking V_0 and V_b as the inlet and gap mean velocities respectively, and considering mass conservation, one obtains $V_b = H/(H-D)*V_0$. The main flow temperature and wall temperature have been set to $T = 290.15K$ and $Twall = 315.15K$, respectively. The free stream Mach number at the channel inlet is 0.15. No turbulence inflow boundary conditions have been considered for the *NLES* approach. The low turbulence level measured in the experiments, $Tu = 1.3\%$, has been imposed at inlet section in the *RANS* and *URANS* simulations.
The computational domain has been discretized with an unstructured mesh with 5.3×10^6 elements. The mesh has been generated as a 2D grid and then extruded in the z-direction. This procedure improved the discretization in the span-wise direction, since the extrusion of parallel planes allows a better control of the spatial distribution along the extrusion z-axes. At the top and bottom surfaces (see Figure 1b) the procedure yields a y^+ value below 1 over all the wall faces. An hyperbolic law has been considered to cluster the grid in the wall regions (z-direction), where the grid spacing is close to the typical *DNS* setup. Aspect ratios of the final mesh can be summarized as follow: $\delta z/\delta x$ in the base region in correspondence of pins is 0.15, and considering the normal to the wall direction of pins $\delta x/\delta y$ is 0.025. The typical cell volume in the midspan region, characterized by the coarsest exahedral distribution, is $1.7 \times 10^{-4} mm^3$ in near wall region and $1.2 \times 10^{-2} mm^3$ in the wake region. The averaged grid skewness computed in the wake region is 0.09 while the maximum value is 0.22. The skewness is defined $(OptimalCellSize - CellSize)/OptimalCellSize$ where, the optimal cell size is the size of an equilateral cell with the same circumradius. The non-dimensional time step Δt^* has been set equal to 1.52×10^{-4}, corresponding to *CFL* < 1 everywhere. The same time step has been considered both for the *URANS* and *NLES*.
A grid independence analysis has been performed (in the *RANS* approach) in order to obtain a proper reconstruction of the shear layer in the near wall regions. Several calculations have been performed considering only the two dimensional (x, y) plane at midspan, analysing the quality of the reconstructed velocity profile in the near wall region around the first pin. Moreover a different set of



(a) **Test Case**



(b) **Mesh details**

Figure 1: **Domain scheme and grid features**

parameters of the hyperbolic grid distribution law, have been tested in order to determine the best configuration for the reconstruction of the velocity profile near the end walls (z-direction). The final grid is comparable with the grid adopted for the same calculation by other authors Delibra *et al*., 2010 (b). The same grid has been employed for both *URANS* and *NLES* computations.

RESULTS AND DISCUSSION

In the present section the numerical time averaged results are compared with the experimental data of Ames *et al*. (2005) at Reynolds number equal to 10000, which, as recognised by Simoneau *et al*. (1984), is representative of the operating conditions of a real turbine blade cooling system.

Aerodynamic

In Figure 2 the numerical results obtained with *RANS*, *URANS* and *NLES* are compared in terms of surface pressure coefficient distribution at midspan (Cp), defined as:

$$Cp = \frac{p - p_0\left(\theta_0\right)}{1/2\rho V_{max}^2} \tag{1}$$

and plotted versus the angular location θ around the cylinder, the origin being located at the pin leading edge. The experiments indicate the occurrence of an extensive separation around the first pin, whose starting point is located at about $80°$ and it is well reproduced by all calculations. The agreement is also good for the 2^{nd} pin, while some differences are observed for the 3^{rd} pin. These discrepancies may be due to the insufficient spanwise extension of the computational domain and possibly to the constrain enforced through the lateral boundary conditions applied over a reduced channel width.
By comparing the different results, it can be observed that *NLES* is overall in better accordance with experimental data, although some discrepancies appear in the accelerating part where *URANS* seems to perform better. *RANS* approach gives acceptable results for the first pin, and the agreement gets worse and worse for the downstream pins.
A similar trend is observed monitoring the velocity profiles at midspan in the normal to the streamwise direction at different axial locations, namely $x/D = 10$, $x/D = 12.5$ and $x/D = 15$ presented in

Figure 2. Again the major features of the mean flow are correctly reproduced. The comparison of the velocity profiles between experimental and numerical data are shown in Figure 3 at mid-span in the normal to the mainstream direction (line *a*). The agreement among the closures is good for rows 1-2 and satisfactory for row 3.

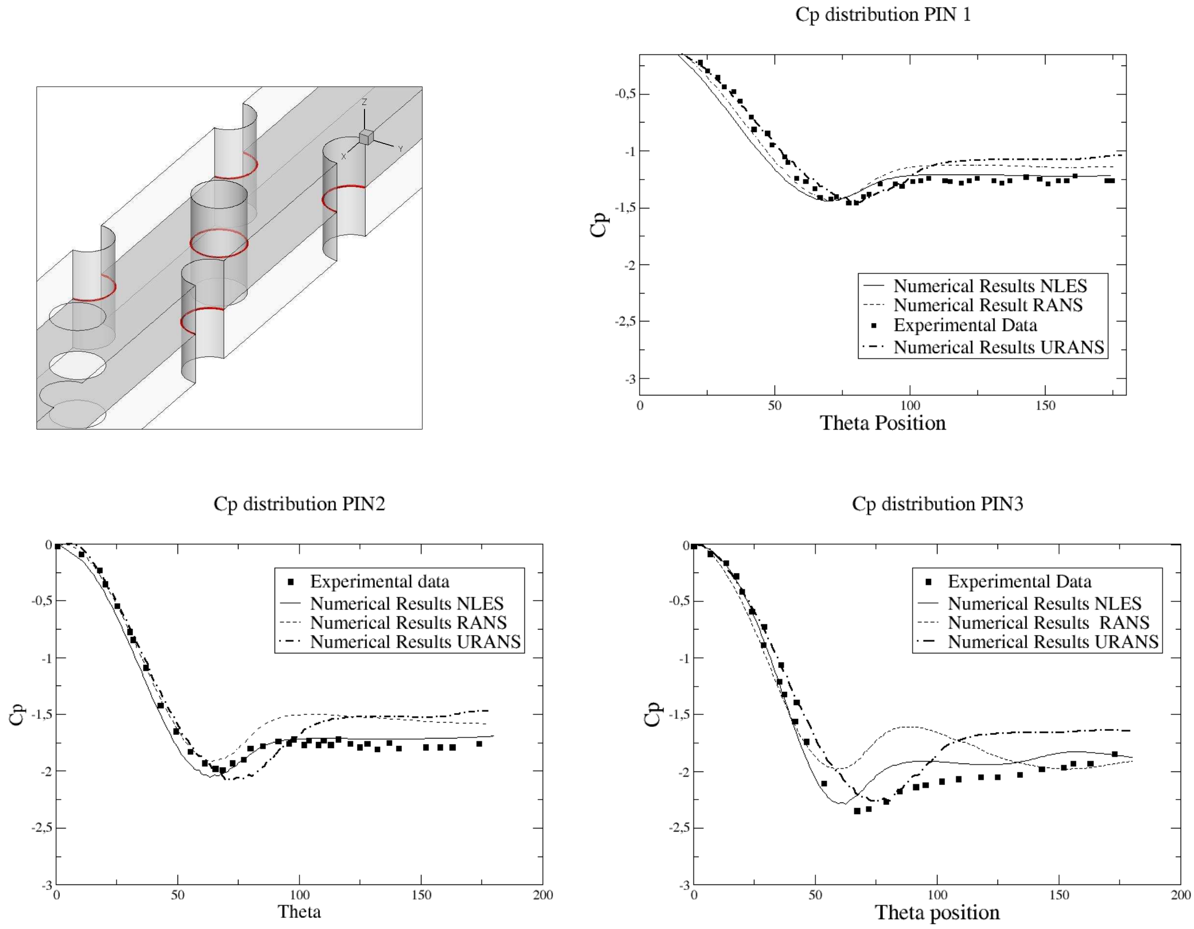


Figure 2: Pressure Coefficient for pin at midspan

The vortex shedding frequency as predicted by *URANS* and *NLES* led to a Stroual number of $St = 0.243$ which is very close to the experimental one, corresponding to $St_{exp} = 0.234$.
Concerning the quality of *NLES* in terms of resolution and the position of the filter cut-off, it can be appreciated from the streamwise velocity fluctuation power spectrum given in Figure 5, that the inertial sub-range development with a $-5/3$ slope is correctly reproduced.

Heat transfer

In order to provide a more comprehensive quantitative comparison between experiments and calculation, following the procedure adopted in Ames *et al*. (2005), the Nusselt number distribution (Nu) reported in Figure 4b have been normalized with the surface-averaged value on the bottom wall. Experimental data are affected by uncertainties estimated smaller than 11.4%, which correspond to $Nu/Nu_{ave} = 0.1$. Recalling that the driving mechanism controlling the heat transfer enhancement on the plate is the vortex shedding generated by each pin, which cannot be predicted by any steady *RANS* simulation, it is generally accepted that the Nusselt number distribution is underestimated.

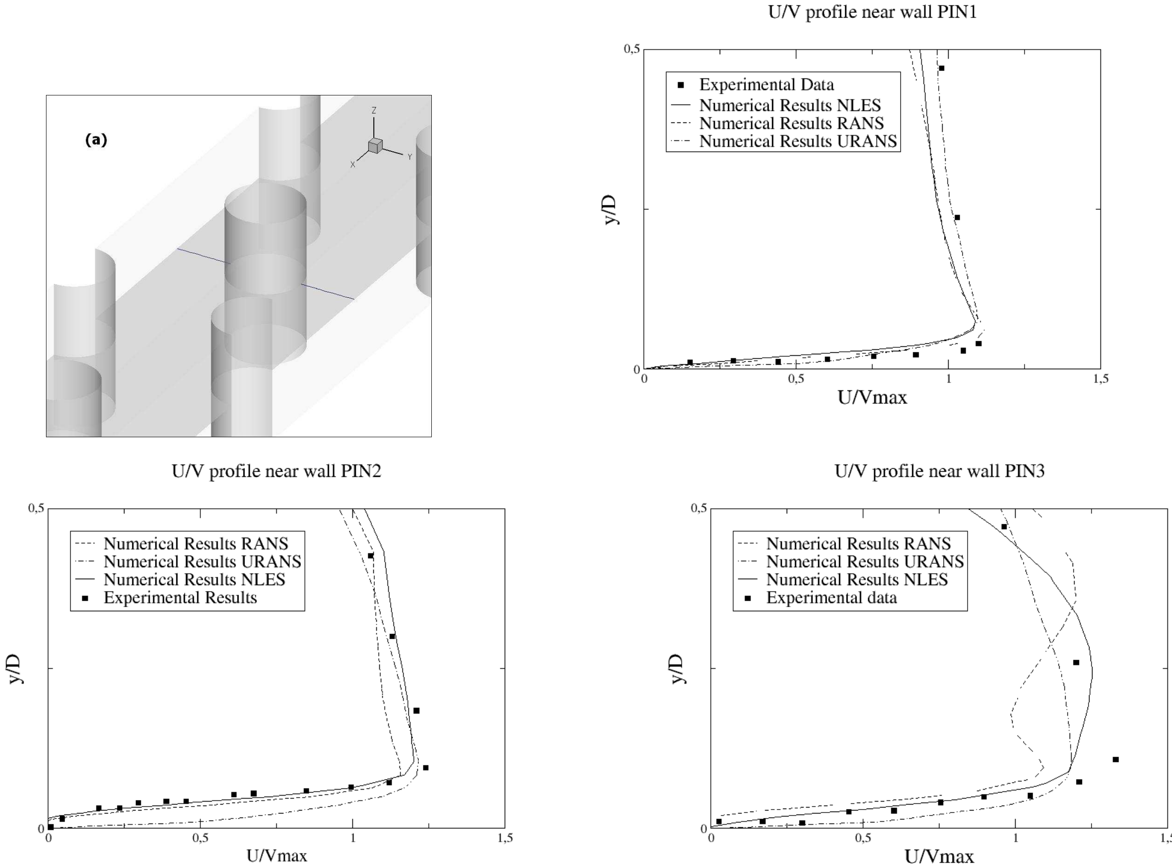
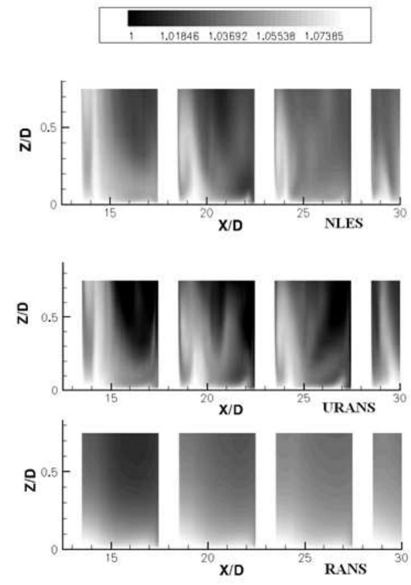
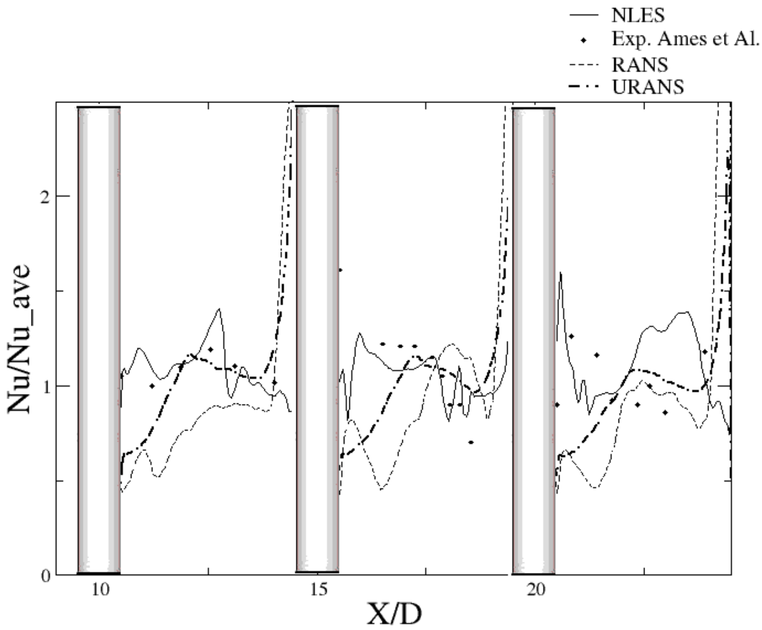


Figure 3: Comparison of mean Velocity U/V_{max} between numerical and experimental data in the midplane along the line (a) for rows 1-3

Conversely in *URANS* the major unsteady characteristic are accounted for, and consequently the heat transfer prediction is improved. As expected the *NLES* simulation shows the best agreement in terms of Nusselt distribution especially in the zone between the first and the second pins and the zone between the second and third pins.
These results are confirmed by the temperature contours plotted in Figure 4a where the effects of vortical structures controlling the heat transfer mechanism, are evident.
The discrepancies between the numerical and the experimental results in the rear part of the channel are most likely due to the local coarsening of the mesh. Moreover present results, confirmed by other computational works on the same test case (see Delibra *et al.*, 2010 (b) and Benhamadouche *et al.* (2012)), seems to suggest that the choice of the distance between the periodic lateral boundaries is not adequate. For a better prediction of the local flow features a greater number of pin arrays in the spanwise direction should probably be considered. The size of the coherent structures nearby each pin rows in the downstream area is underestimated in the calculations. This is most likely due to the already mentioned effect of the insufficient channel width of each periodic boundaries. This effect is confirmed both from the heat transfer distribution, the surface wall pressure coefficient, and the velocity profiles respectively given Figure 4b in Figure 2 and velocity profile of Figure 3, where all numerical predictions show discrepancies with respect to the experiments in the far downstream region.



(a) Temperature contouring in the pin to pin domain



(b) Nusselt distribution

Figure 4: Heat Transfer prediction

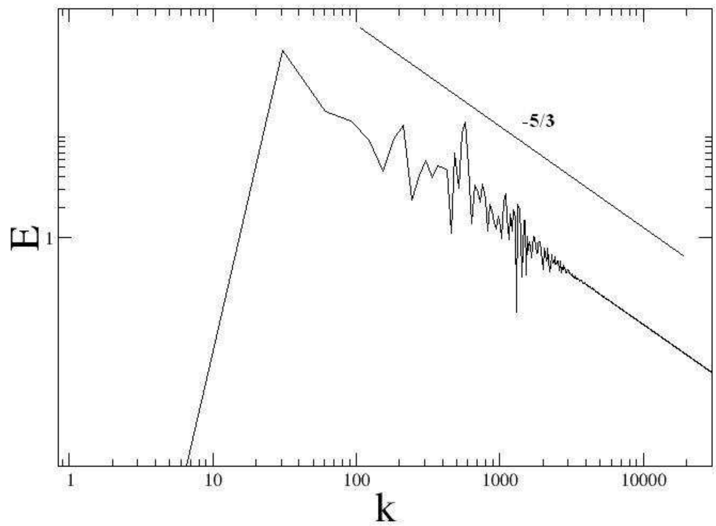


Figure 5: Streamwise velocity fluctuation power spectrum *(NLES)*

CONCLUSIONS

The paper presents a critical analyses of three widely used turbulence models, as applied to a currently open problem in fluid mechanics represented by the flow over realistic geometry typical of turbomachinery cooling device. The selected test case consisting of a pin fin cooling channel config-

uration allows to investigated the effects of the aerodynamic field on the heat transfer characteristics of the device with very complicated flow structures.
The present results are expected to be of favour to industrial oriented designers providing quantitative information on the device aero-thermal fields, as predicted by the different turbulence closures. The quality of the final results must also be considered in the light of the core hours needed to perform the calculation. On a IBM cluster ($2.40GHz$ processors),*RANS* calculation has been performed in 400 cpu hours. On the same machine *URANS* and *NLES* calculations took about 20000 cpu hours each. Even if in order to more easily compare the results, grid and time steps considered herein are identical, *RANS* and *URANS* calculations could be performed with a much small computational cost. Numerical predictions related to the aerodynamic features of the presented test case are in general in good agreement among each other, for all performed calculations. Results are in better agreement with experimental data for the time-averaged solution obtained from the unsteady calculation, particularly for the downstream pins. The analyses of the aerodynamic characteristics of the flow as predicted by the different closures suggests that the *NLES* better reproduce the flow in the rear part of the devices, while the *URANS* approach ($k - \omega$ SST with Menter transition model) provides better prediction on the separation on the pin surface.
Considering the heat exchange, the capability of *RANS* to reproduce such large separation region is questionable.

REFERENCES

Adami, P., Martelli, F., Michelassi, V., *Three Dimensional Investigations for Axial Turbines by an Implicit Unstructured Multi-Block Flow Solver.* In Proc. ASME IGTI Turbo Expo ASME 2000-GT-0636, Munich, Germany. 2000.

Ames, F.E., Dvorak, L.A., Morrow, M. J. *Turbulent Augumentation of Internal Convection Over Pins in Staggered Pin-Fin Arrays.* ASME J. turbomachinery. 2005, 127, pp. 183-190.

Anderson, W. K., Bonhaus, D. L. *An Implicit Upwind algorithm for Computing Turbulent Flow on Unstructured Grids.* Computer and fluids. 1994, Vol. 23, 1, pp. 1-21

Belardini, E., Adami, P., Martelli, F. *Development of an Unsteady Parallel Approach for 3D Stator-Rotor Interaction.* In: Science and Supercomputing at CINECA. In: Science and Supercomputing at CINECA. 2001, pp. 338-355.

Benhamadouche, S., Afgan, I., Dehoux, F., Manceau, R. *Unsteady RANS and large-eddy simulation of the flow and heat transfer in a wall bounded pin matrix* Proc. of THMT Conference, Palermo 7th 24-27 September, ICHMT DIGITAL LIBRARY ONLINE (2012)

Bernardini, C., Carnevale, M., Salvadori, S., Martelli, F. *On the Assessment of an Unstructured Finite-Volume DES/LES Solver for Turbomachinery Application.* WSEAS TRANSACTIONS ON FLUID MECHANICS. 2011, Vol6, Issue 1 (Print ISSN:1790-5087, E-ISSN 2224-347X).

Calvo, M., Franco, J.M. and Randez, L. *A New Minimum Storage Runge-Kutta scheme for Computational Acustics.* J. Comput. Phys. 2004, Vol. 201, pp. 1-12.

Carnevale, M., Salvadori S., and Martelli F. *On the use of LES in the aero-thermal analysis of internal cooling systems.* Proc. of THMT Conference, Palermo 7th 24-27 September, ICHMT DIGITAL LIBRARY ONLINE (2012).

Carnevale M., *Development of Large Eddy Simulation Modelling for Turbomachinery Application* PhD Thesis 2012.

Carnevale M., Montomoli F., D'Ammaro A., Salvadori S. *A Stochastic Method for Heat Transfer Prediction Using LES* Proc. of ASME Conference Copenhagen June 11-15 2012.

Dahlquist, G., Bjork, A. *Numerical Methods.* Englewood Cliffs, NJ: Prentice-Hall. 1974.

G. Delibra, D. Borello, K. Hanjalic, F. Rispoli. *URANS of the Flow and Endwall Heat Transfer in a Pinned Passage Relevant to Gas-Turbine Blade Cooling.* International Journ. of Heat and Fluid Flow. 30: 549-560, 2009.

Delibra, G., Borello, D., Hanjalic, K., Rispoli, F., *An Insight into into Convective Mechanism of Heat Transfer in a Wall-Bounded Pin Matrix.* In Proc. of the International Heat Transfer Conference (IHTC14), 8-13 August,Washington DC. 2010.

G. Delibra, K. Hanjalic, D. Borello, F. Rispoli. *Vortex structures and heat transfer in a wall-bounded pin matrix: LES with a RANS wall-treatment.*International Journ. of Heat and Fluid Flow. 31: 740-753, 2010.

Garnier, E., Mossi, M., Sagaut, P. and Davile, M. *On the Use of Shock-Capturing Schemes for Large-Eddy Simulation.* Journ. of Computational Phys. 1999, 153, pp. 272-311.

F Hussain J.Jeong. *On the identification of a vortex journal of fluid mechanics*. Journ. of Fluid Mech. Vol. 285, pp. 69-94, 1995.

Menter, F.R. *Improved Two-Equation k-w Turbulence Models for Aerodynamic flow* Technical Report, NASA Technical Memorandum. 1992, 103975.

Menter, F.R., Langtry, R.B., Likki, S.R., Suzen, Y.B., Huang, P.G, Volker. S., *A Correlation - Based Transition Model Using Local Variables - Part I: Model Formulation.* ASME J. Turbomach. 2006, Vol. 3, 128, pp. 413-422.

Montomoli F., Massini M., Salvadori S., *Geometrical Uncertainty in Turbomachinery: Tip Gap and Fillet Radius.* Computers and Fluids, Vol. 46, Issue 1, July 2011, pp. 362-368, DOI: 10.1016/j.compfluid.2010.11.031.

Roe, P.L. *Characteristic based Scheme for Euler Equations.* Ann. Rev. Fluid. Mech. 1986, Vol. 1.

Saad, Y., Schults, M.H.,Gmres. *Generalized Minimal Residual algorithm for Solving non-Symmetric linear System, SIAM.* Journ. Sci. Stat. Comput. 1986, 7, pp. 4873-4894.

Salvadori, S., Bernardini, C., Adami, P., Martelli, F. *Turbulence and Transition Modelling in Transonic Turbine Stage.* Proc. of the 19th ISABE CONFERENCE 7-11 September 2009, Montreal, Canada. ISABE-2009-1218

Simoneau, R. J., and Van Fossen, G. J. *Effect of Location in array on Heat Transfer to a Short Cylinder in Crossflow.* J HEAT TRANS-T ASME. 1984, Vol. 106, pp. 183-190.

Thornber, B., Modedale, A., Dikakis, D., Youngs, D., Williams, R.J.E. *An Improved Reconstruction Methods for Compressible Flow with Low Mach Number.* Journ. of Comp. Ph. 2008, 227, pp. 4873-4894.

Tucker, P.G. *Computation of Unsteady Turbomachinery Flows: Part 2 - LES and Hybrids.* Progress in Aerospace Sciences. 2011