Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 82 (2015) 222 – 229

Energy Procedia

ATI 2015 - 70th Conference of the ATI Engineering Association

Heat transfer investigation on an internal cooling system of a gas turbine leading edge model

Antonio Andreini[a], Emanuele Burberi[a], Lorenzo Cocchi[a], Bruno Facchini[a], Daniele Massini[a]*, Marco Pievaroli[a]

[a]*Department of Industrial Engineering, University of Florence, Via Santa Marta 3, 50139, Florence, Italy*

Abstract

A scaled up test model simulating a realistic leading edge cooling system of a high pressure gas turbine blade was designed with the aim of performing heat transfer measurements in static and rotating conditions.
The test model is composed by a trapezoidal supply channel which feeds three large racetrack holes, generating coolant impingement on the internal concave leading edge surface. Four big fins allow to confine the impingement jets impact zones. Air is then extracted through 4 rows of 6 holes each, two of showerhead (SH) and two of film cooling (FC). The test model is installed on a rotating test rig, which allows to reach jet Reynolds numbers (Re_j) up to 40000 and rotation numbers (Ro_j) up to 0.05. The effect of cross-flow in the supply channel is also considered.
The heat transfer coefficient (HTC) distribution on the internal concave surface was evaluated by means of a steady state technique, using wide band thermochromic liquid crystals (TLCs) to measure the wall temperature and an electrically heated Inconel sheet to provide a constant heat flux to the investigated surface.
This paper reports experimental results obtained in static conditions for Re_j 10000 and 30000 and for two cross-flow cases representative of blade tip and hub sections. The effects of different mass flow extraction between pressure and suction side is also investigated by varying the mass flow rate through FC and SH holes.
The effect of the coolant extraction holes geometry on the Nusselt number distribution is analyzed by comparing the experimental results, reported as 2D Nusselt number (Nu) maps, with a previous investigation on an analogous test model with similar impingement geometry. A CFD campaign was also carried out on the present test rig, exploiting a previously validated computational model. Both numerical and experimental results reveal that the effects of differentiated mass flow extraction and extraction area modification are secondary with respect to the effects of Re_j and crossflow variation.

Peer-review under responsibility of the Scientific Committee of ATI 2015

Keywords: Heat transfer measurement ; thermochromic liquid crystals ; CFD investigation ; Heat transfer numerical analysis ; Rotating leading edge ; Jet impingement cooling

* Corresponding author. Tel.: +39 055 2758 713
E-mail address: daniele.massini@htc.de.unifi.it

Peer-review under responsibility of the Scientific Committee of ATI 2015
doi:10.1016/j.egypro.2015.12.026

1. Introduction

Present day gas turbine performance levels have been mainly reached increasing the Turbine Inlet Temperature (TIT). This trend, however, leads to higher thermal loads for the turbine components, which need to be managed through highly efficient cooling systems, in order to ensure their safety.

The blade leading edge (LE) is one of the most critical regions, given its unfavorable geometry and the presence of the hot gas stagnation point. Particularly effective cooling techniques are required, one of which consists in multiple coolant jets impinging in the LE internal surface. The flow field generated by a similar system can be characterized through two main non-dimensional parameters: the jet Reynolds number ($Re_j = (\rho u_j D_{h,j})/\mu$) and, in rotating blades, the Rotation number ($Ro_j = (\Omega D_h)/u_j$).

Heat exchange performances are determined by the complex interaction between the impinging jets, the coolant extraction configuration and the LE geometry itself. As a consequence, an experimental investigation on a similar system needs to replicate both actual geometrical and boundary conditions.

Jet impingement on a curved target surface was firstly studied by Chupp et al [1], Metzger et al. [2] and Hrycak [3], achieving a good similarity with a real LE cooling system, performing tests for different nozzle-to-plate distances, Reynolds numbers and temperature differences.

In more recent years, the effect of coolant extraction was studied by Metzger and Bunker [4]: such work showed that film cooling holes arrangement can significantly influence heat exchange, but also that an asymmetry in flow extraction has a negligible effect. Such study also indicated that jet Reynolds number is the main performance drive parameter for impingement systems.

Taslim et al. ([5], [6], [7], [8]) and Andrei et al.[9] studied the effects of leading edge internal geometry and boundary conditions, such as coolant extraction and internal and external cross-flow: these studies revealed that cross-flow caused heat transfer to be higher on the side walls than on the nose.

A new rotating test rig has been designed in order to investigate rotation effects on a LE geometry with three impingement jets and two rows of showerhead (SH) and two of film cooling (FC) holes.

A preliminary experimental campaign was performed by imposing symmetrical coolant extraction in static conditions, thus allowing a comparison with a CFD RANS investigation as explained in [10].

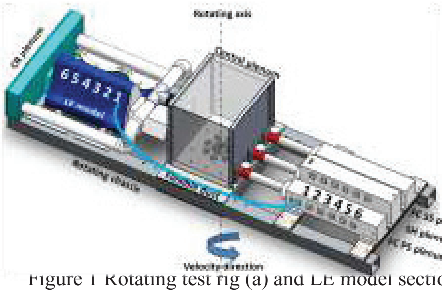
In this paper further experimental results obtained from static HTC measurements will be presented and used to evaluate the effect of asymmetric mass flow extraction, i.e. of pressure and suction side. Moreover, results of Andrei et al. [9] have been rearranged in order to have a more consistent comparison with the present tests, thus allowing to identify the effect of extraction holes area and position.

A RANS CFD investigation has been carried out in order to support the experimental evidence.

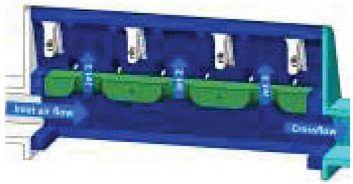
Nomenclature

D_h	Jet hydraulic diameter	[m]	y	Radial direction	[m]
HTC	Heat Transfer Coefficient	[W/m^{2}K]	ω	Turbulent kinetic specific dissipation	[s^{-1}]
k	Turbulent kinetic energy	[m^2/s^2]	Ω	Angular velocity	[rad/s]
k_{air}	Air thermal conductivity	[W/mK]	**Acronyms**		
μ	Air dynamic viscosity	[kg/ms]	CFD	Computational Fluid Dynamics	
Nu	Nusselt number	[-]	Cr	Cross-flow	
Nu_{ave}	Averaged Nusselt number	[-]	FC	Film Cooling	
P_y	Impingement jets pitch	[m]	LE	Leading Edge	
$\dot{q}$	Heat flux	[W/m^2]	PS	Pressure Side	
ρ	Air density	[kg/m^3]	RANS	Reynolds Averaged Navier Stokes	
Re_j	Jet Reynolds number	[-]	SH	Shower-Head	
Ro_j	Jet rotation number	[-]	SS	Suction Side	

T_c	Coolant temperature	[K]	SST	Shear Stress Transport
T_w	Wall temperature	[K]	TIT	Turbine Inlet Temperature
u_j	Jet flow velocity	[m/s]	TLC	Thermochromic Liquid Crystals
x	Circumferential direction	[m]		



(a)



(b)

Figure 1 Rotating test rig (a) and LE model sectional view (b)

2. Experimental facility and post-processing

Figure 1a shows a sketch of the test rig, which is an open loop suction type wind tunnel located at the Industrial Engineering Department of the University of Florence. The rig is installed on a rotating chassis designed and first tested by Bonanni et al [11].

In Figure 1b a sectional view of the leading edge (LE) model is reported. Cooling air enters the rig at ambient pressure and temperature and passes into a trapezoidal supply channel. Part of such flow is supplied to an impingement plate, which generates three jets by means of three racetrack-shaped holes, while the remaining part leaves the channel at the other side obtaining the cross-flow (Cr). Four ribs are located on the impingement target surface and allow to confine the jet impact zones. Coolant air is then extracted by film cooling (FC) and shower head (SH) holes. In order to reproduce a realistic internal flow field a fine regulation of the mass flow through each outlet is required. A system of four plenum, equipped with gate valves, allows to independently set the mass flow through the cross-flow outlet, the two showerhead rows together and the film cooling rows on suction and pressure side separately. In order guarantee the same mass flow rate from every hole also in rotating tests, where a centrifugal pressure gradient is present, each hole and the corresponding inlet on the plenum are located at the same radius.

The overall mass flow rate measurement is allowed by a calibrated orifice positioned downstream the test section and by three other orifices, experimentally calibrated, connected to the three plenum on board.

The heat transfer coefficient (HTC) distribution was measured exploiting a steady state technique which allows to perform tests without placing cameras on-board: a constant Joule heat flux is imposed by means of an electrically heated 25.4 μm thick Inconel Alloy 600 sheet applied on the LE internal surface and supplied by two copper bus bars. Temperature measurement on the impingement target surface is achieved through Hallcrest 30C20W wide band thermochromic liquid crystals (TLCs) with an activation band comprised between 30 and 50 °C. TLC activation range was chosen to ensure TLCs to be colored, and thus measurement to be performable, on the whole measurement surface in every test condition.

Before the experiments, a robust TLC color temperature response calibration was performed to achieve high accuracy measurements, according to the steady state gradient method [12]. During the tests all the conditions were set identical to the calibration, with the aim to avoid color variation problems due to the lighting/viewing arrangement.

Energy conservation provides the following definition of convective heat transfer coefficient:

$$HTC = \frac{\dot{q}_{conv}}{T_w - T_c} = \frac{\dot{q}_{Joule} - \dot{q}_{loss}}{T_w - T_c} \quad (1)$$

where $\dot{q}_{conv}$ is the convective heat flux, $\dot{q}_{Joule}$ is the imposed generated heat flux, $\dot{q}_{loss}$ is the portion of heat flux not exchanged through convection, T_w is the wall temperature and T_c the coolant temperature measured by T type thermocouples. Considering the complex geometry investigated, the Joule heat flux does not have a constant distribution, for this reason a thermal-electric FEM analysis was performed exploiting ANSYS Mechanical APDL® v15: the measured temperatures have been used as boundary conditions, a first HTC distribution was calculated considering a constant heat flux, the result of the simulation was then a more realistic Joule heat flux distribution. The whole measurement technique and post-processing is deeply described in [10] together with the uncertainty quantification. A maximum error of 30% in Nu evaluation was found in the central part of the jet impingement area, where the temperature difference between coolant and wall temperature is smaller than any other point on the investigated area.

3. Computational analysis

A computational analysis of the test rig was performed using a steady RANS approach, in order to deepen the comprehension of the fluid phenomena and of the effects of different extraction areas. A previous work [10] demonstrated this approach to be suitable for the present geometry and flow conditions: in particular, an accurate reproduction of Nusselt distribution shape and of the effects of Re_j and crossflow variation were achieved.

In order to underline the effect of different coolant extraction areas, computations were performed considering uniform mass flow extraction. This choice, together with the symmetry of the test model, allowed to reduce the computational domain to only half of the real geometry.

Following a mesh sensitivity analysis [10], grids were built using ANSYS ICEM® v15, consisting in hybrid unstructured meshes with tetrahedral elements in the freestream region and 20 layers of prisms in proximity of the physical walls. This resulted in $6.1 \cdot 10^6$ elements for the full extraction area case and in $4.0 \cdot 10^6$ elements for the reduced area case. The computations were performed using the solver ANSYS CFX® v15, using the implemented k-ω SST formulation for turbulence modeling [15]. Further details of the computational setup and of the boundary conditions are identical to the ones adopted for previous simulations on the same rig, and can be found in [10].

4. Results

In this section, the results will be shown in terms of Nusselt number based on the hydraulic diameter of the impingement holes:

$$Nu = \frac{HTC \cdot D_h}{k_{air}} \quad (2)$$

In the following, cross-flow and leading edge direction will be referred as radial, while the lateral extension of the profile will be defined as circumferential, in line with the real blade reference system. For a better representation, Nu distribution on the inner curved target surface has been reported to a flat plane rolling out the LE geometry. On such flattened surface, horizontal coordinate x represents the circumferential distance from the LE, and it is scaled with respect to the maximum lateral extension x_{max}; instead, vertical coordinate y indicates the radial distance from the inner point of the measurement surface, and it is scaled with the impingement holes radial pitch P_y. The area corresponding to the ribs has been removed from the analysis, because of the constraints implied by both the geometry and the measurement technique; at the same time, even the extraction holes positions have been excluded from the analysis. In order to clarify the relative positions of impingement and extraction holes, the projection of the former is also represented on the maps.

In the present work four test conditions will be compared, resulting by the combination of two values of Re_j (10000 and 30000) and two crossflow configurations, corresponding to 70% (blade hub section) and 10% (blade tip section) of such total cooling flow.

4.1. Non uniform coolant extraction effects

In order to deepen the effect of the differential coolant extraction and to evaluate the effects of the external pressure distribution on the internal cooling system, two different configurations were tested. In the first, hereinafter referred to as *uniform*, each extraction row drew 25% of the coolant flow entering the impingement cavity. In the second, hereinafter referred to as *non uniform*, 40% of the impingement flow was extracted by the FC row on the suction side, 10% by the FC row on the pressure side and the remaining 50% by the two SH rows.

In Fig. 2 the obtained experimental Nu/Nu_{ave} distributions are reported, where Nu_{ave} is the average value for each test. A proper analysis of Nu distribution shape and of Re_j and cross-flow variation effects is reported in [10].

It is possible to note that a non uniform coolant extraction has a minimal effect on the Nusselt

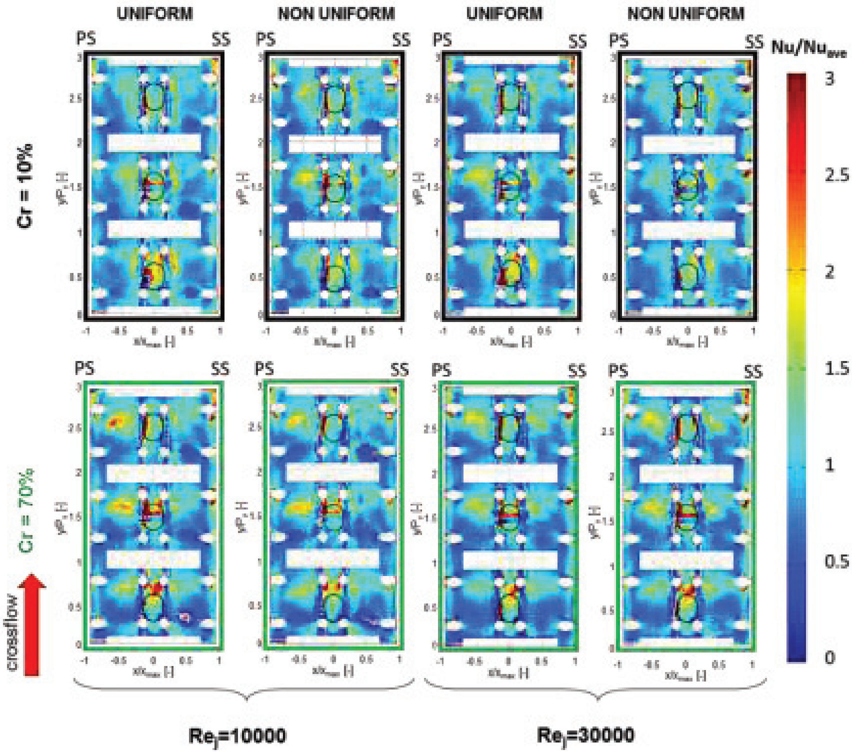


Fig. 2 Non uniform coolant extraction effect - Experimental results

distribution: this can be attributed to the major importance of the jet impact zone, influenced mostly by the feeding channel. As already mentioned in [10], the Reynolds number does not influence the Nu/Nu_{ave}

distribution and all the effects are scaled with the jet velocity. On the contrary, the cross-flow conditions are responsible for a different jet inclination and expansion within the leading edge cavity, which strongly affects the HTC pattern on the target surface. Following these considerations, from now on only results obtained for Re_j=10000 will be compared.

4.2. Overall coolant extraction area effects

In order to evaluate the discharge area effect on heat exchange, results from Andrei et al. [9] on a similar LE geometry having different extraction area have been exploited and compared with the new experimental results and CFD simulations. Two different computational models were developed: the first model, referred as RANS1, closely replicates the real test rig geometry, as showed in [10]; the second one, referred as RANS2, is identical to the first, apart from a reduction in FC and SH holes cross sectional area. Its extraction area was reduced in order to obtain the same value of Andrei et al. [9] tests, and was then scaled down to consider the different scale factors of previous and new geometries.

The comparison between the maps of each test point reported in Fig. 3 reveals that a reduction in extraction area does not significantly alter the Nu distribution shape.

Experimental results show very similar Nu peaks related to the impingement jet shapes, nevertheless a sharper Nu reduction far from them can be observed in Andrei et al. [9] results.

The Nu distribution seems to be well replicated by both simulations as well. Computations performed with Cr=70% show a stronger similarity between full and reduced extraction area cases, while slight differences are present for the Cr=10% tests: a smaller extraction area causes a reduction in extension and

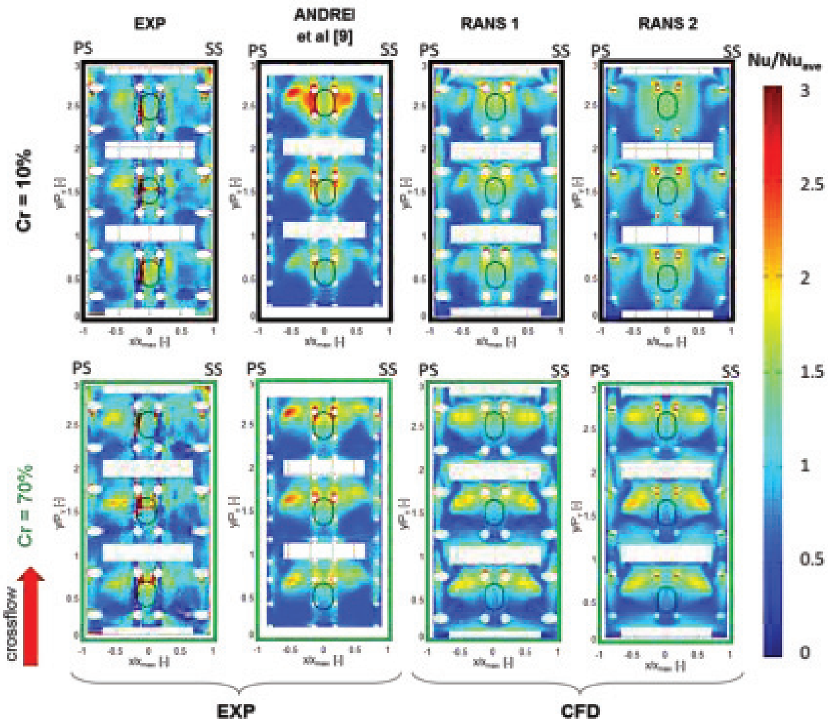


Fig. 3 Coolant extraction area effect - Experimental and numerical results

entity of the lateral Nu peaks, while the central high Nu area appears to be widened and strengthened.

This fact can be justified by considering that a smaller crossflow rate causes the flow to be guided more easily by the impingement hole: this leads to a more coherent and less inclined jet, which impacts more directly with the central curved region of the target surface where the showerhead holes are [10].

The different and smaller SH extraction holes strongly affect the flow field in such zone since they cause a stronger acceleration field, which intercepts a higher jet mass flow and draws it towards the central region. For the tests with Cr=70%, where the jet impacts mainly on the lateral flat surfaces, the effects on heat exchange of the flow field surrounding the extraction holes are small.

Finally the averaged Nu values (Nu_{ave}) of all the performed tests are compared, varying both the Re_j (between 10000 and 40000) and the three cross flow conditions (10%, 40% and 70%), in order to validate both the measurement technique, the new test rig and the CFD RANS analysis.

All these results are reported in Figure 4 plotted against the correlations proposed by Chupp et al. [1] and Taslim et al. [5] and the results obtained by Andrei et al. [9], showing good agreement.

5. Conclusions

Experimental and numerical investigations have been carried out on a test rig simulating a realistic leading edge cooling system of a high pressure gas turbine blade. Results for two different Reynolds number and two cross flow conditions have been obtained imposing a non uniform coolant extraction between suction and pressure side FC rows in order to reproduce actual flow field conditions within the LE cavity. The effect of the differential extraction showed to be not relevant compared with tests performed imposing a uniform extraction. Moreover, in order to assess the extraction holes area and position effects on heat transfer coefficient distribution, a further comparison with experimental results from a previous measurement campaign on a similar LE geometry with different coolant extraction area was performed. This additional analysis showed that also these effects are secondary with respect to Re_j and Cr variations both in terms of averaged Nu values and of Nu distribution on inner target surface. This experimental evidence was further validated by a numerical investigation in which the tested model was reproduced varying the coolant extraction area. The next step is to extend the experimental investigation with rotating tests.

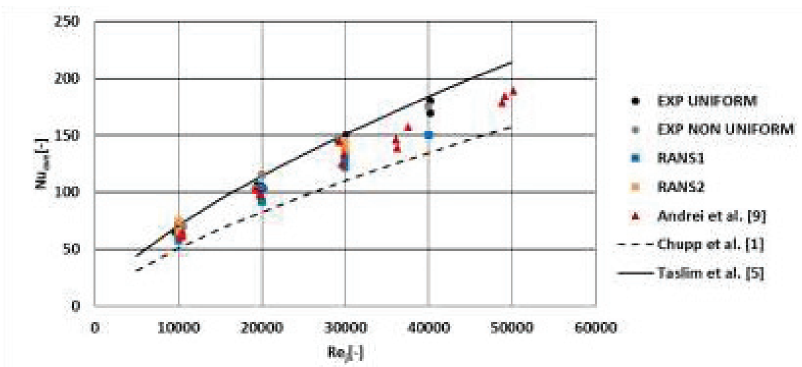


Figure 4 Nu_{ave} vs Re_j

Acknowledgements

The reported work has been conducted within the national research project PRIN 2010-11 INSIDE, supported by the Italian Ministry of Education, University and Research (MIUR).

References

[1] Chupp, R., Helms H, McFadden, P, Brown T. Evaluation of Internal Heat Transfer Coefficients for Impingement Cooled Turbine Blades, *J Aircr* 1969;**6**:203-208.

[2] Metzger D, Baltzer R, Jenkins C. Impingement Cooling Performance in Gas Turbine Airfoils Including Effects of Leading Edge Sharpness. *ASME J Eng Power* 1972;**94**:219-225.

[3] Hrycak P. Heat Transfer From a Row of Impinging Jets to Concave Cylindrical Surfaces. *Int J Heat Mass Transfer* 1981;**24**:407-419.

[4] Metzger D, Bunker R. Local Heat Transfer in Internally Cooled Turbine Airfoil Leading Edge Regions: Part I Impingement Cooling Without Film Coolant Extraction. *ASME J Turbomach* 1990;**112**:451-458.

[5] Taslim M E, Pan Y, Spring S D. An Experimental Study of Impingement on Roughened Airfoil Leading-Edge Walls With Film Holes. *ASME J Turbomach* 2001;**123**:766-773.

[6] Taslim M, Bakhtari K, Liu H., Experimental and Numerical Investigation of Impingement on a Rib-Roughened Leading-Edge Wall. *ASME Paper No. GT2003-38118* 2003.

[7] Taslim M, Bethka, D. Experimental and Numerical Impingement Heat Transfer in an Airfoil Leading-Edge Cooling Channel With Crossflow. *ASME Paper No. GT2007-28212* 2007.

[8] Elebiary K., Taslim M. Experimental/Numerical Crossover Jet Impingement in an Airfoil Leading-Edge Cooling Channel. *ASME Paper No. GT2011-46004* 2011.

[9] Andrei L, Carcasci C, Da Soghe R, Facchini B, Maiuolo F, Tarchi L, Zecchi S. Heat Transfer Measurements in a Leading Edge Geometry With Racetrack Holes and Film Cooling Extraction. *J Turbomach* 2013;**135(3)**:031020.

[10] Bianchini C, Burberi E, Cocchi L, Facchini B, Massini D, Pievaroli M. Numerical analysis and preliminary experimental heat transfer measurements on a novel rotating leading edge model. *ISAIF12_092*, under review.

[11] Bonanni L, Carcasci C, Facchini B, Tarchi L. Experimental Survey on Heat Transfer in a Trailing Edge Cooling System: Effects of Rotation in Internal Cooling Ducts. *ASME Turbo Expo 2012: Turbine Technical Conference and Exposition: 633-644*

[12] Chan T L, Ashforth-Frost S, Jambunathan K. Calibrating for viewing angle effect during heat transfer measurements on a curved surface. *International Journal of Heat and Mass Transfer* 2001:**44**:2209–2223.

[13] Wilks G. External natural convection about two dimensional bodies with constant heat flux. *International Journal of Heat and Mass Transfer* 1972;**15(2)**:351-354.

[14] Facchini B, Maiuolo F, Tarchi L, Ohlendorf N. Experimental Investigation On The Heat Transfer In A Turbine Airfoil Leading Edge Region: Effects Of The Wedge Angle And Jet Impingement Geometries. *European Turbomachinery Conference, Lappeenranta, Finland, 15-19 April 2013*, 130.

[15] Menter F, Thomas E, Vieser W. Heat transfer predictions based on two-equation turbulence models. *ASME-JSME 2003 Thermal Engineering Joint Conference* 2003.

Biography

Daniele Massini is a Phd student at the University of Florence
2012: Master Degree; 2013/2014: Research Engineer in Industrial Engineering Department, University of Florence; 2014/Ongoing: PhD student in Industrial Engineering Department, University of Florence. Main interests: aerothermal and heat transfer experimental investigations.