GENERIC GRID INTERFACE DEVELOPMENT AND APPLICATION TO CONJUGATE HEAT TRANSFER ANALYSIS

C. Bianchini\textsuperscript{1} – B. Facchini\textsuperscript{1} – L. Mangani\textsuperscript{1} – M. Maritano\textsuperscript{2}

\textsuperscript{1}Dipartimento di Energetica "Sergio Stecco" Via Santa Marta, 3 - 50139 Firenze - Italy
\textsuperscript{2}Ansaldo Energia Via Nicola Lorenzi, 8 - 16152 Genova - Italy

Email: luca.mangani@htc.de.unifi.it, WWW: http://www.htc.de.unifi.it

ABSTRACT

This paper presents the developments introduced in OpenFOAM® libraries, to perform conjugate heat transfer simulations and validations for an internally cooled turbine blade. The solver uses a SIMPLE-like algorithm with a special treatment for the pressure corrector equation to deal with highly compressible flows.

To speed up convergence of the conjugate temperature equation a fully implicit approach has been adopted to solve the coupling for energy between solid and fluid region. Moreover an implicit treatment of generic interface has been developed and applied in this test, allowing treatment of non conformal cyclic patches.

The validation cases to be shown are extracted from experimental tests over the NASA C3X blade, cooled with ten internal ducts. Both subsonic and transonic regimes were investigated. Numerical and experimental results are compared in terms of pressure and temperature distribution on the blade wall at midspan, as well as heat transfer coefficient profiles.

NOMENCLATURE

\begin{tabular}{ll}
Ch\textsubscript{A} & axial chord [m] \\
c\textsubscript{p} & constant pressure specific heat [J/(kg K)] \\
HTC & heat transfer coefficient [W/(m\textsuperscript{2} K)] \\
HTC\textsubscript{0} & reference HTC value = 1135 [W/(m\textsuperscript{2} K)] \\
FV & finite volume [-] \\
k & thermal conductivity [W/(m K)] \\
L\textsubscript{d} & turbulence dissipation length [m] \\
P.S. & pressure side \\
PS\textsubscript{c} & static pressure on cooling outlet [kPa] \\
PS\textsubscript{out} & static pressure on main outlet [Pa] \\
PS\textsubscript{w} & static pressure on blade wall [Pa] \\
PT\textsubscript{in} & total pressure on main inlet [Pa] \\
S.S. & suction side \\
T & static temperature [K] \\
T\textsubscript{ref} & reference temperature = 811 [K] \\
T\textsubscript{Sc} & static temperature on cooling inlet [K] \\
TS\textsubscript{w} & static temperature blade wall [K] \\
TT\textsubscript{in} & total temperature on main inlet [K] \\
Tu & turbulence intensity [-] \\
x & axial distance from leading edge [m] \\
y & dimensional wall distance [m] \\
y+ & non dimensional wall distance [-]
\end{tabular}
INTRODUCTION

The current paper aims at confirming CFD CHT simulations as a viable tool in gas turbine design: however, while the benefits and the current trend on CHT based simulations is favorable in general (as evident from the previously discussed papers), specific and additional contribution of the present investigation is to warn the reader on the effect of correct transition prediction when evaluating metal temperature field on a first stage airfoil through a CHT analysis.

The present work deals with the same geometry analyzed by York et al. 2003: the testcase is described in detail in the experimental study of Hylton et al. 1983. Briefly, it consists of the NASA-C3X linear cascade which has ten radial cooling channels almost located along the chamber line: the experimental activity conducted by Hylton et al. 1983, focused on measuring an accurate set of mid span data (pressure, solid temperature, and HTC profiles). These sets of data are used for open-source CFD code validation purposes for heat transfer predictions after the addiction of specific modules.

According to that, a complete CHT simulation of NASA-C3X turbine vane has been performed using variants of the k-ε and k-ω Shear Stress Transport turbulence models as well as the Spalart-Allmaras one-equation model for the subsonic case. The transonic case was tested only with the a single turbulence model, namely the Two-Layer one. Detailed comparisons between experimental and numerical HTC, pressure, and solid temperature profiles are shown.

CODE DEVELOPMENT

To make OpenFOAM® robust, fast and reliable for RANS heat transfer predictions it was indeed necessary to implement additional submodules. The package coded by the authors within the environment includes a suitable algorithm for compressible steady-state analysis. A SIMPLE like algorithm was specifically developed to extend the operability field to a wider range of Mach numbers. A set of Low-Reynolds number eddy-viscosity turbulence models, chosen amongst the best performing in wall bounded flows, was developed.

After a first validation effort, see Mangani et al. 2007, Mangani et al. 2008 and Innocenti et al. 2008, based on internal thermo-aerodynamic, this article presents an attempt to enhance external aero-thermal prediction capability. Several issues still remain to be addressed within this task, especially in highly compressible regimes.

Numerical methods and physical modeling

The solver implements a SIMPLE like algorithm for steady-state flows modified to solve the Navier-Stokes equation in the compressible form. Extension to high Mach number flows requires
the effects of density derivatives to be taken into account, thus introducing an extra convective term
in the standard Fourier pressure equation which results from the mass continuity equation:
\[
\nabla \cdot \left( \rho \cdot U \cdot \nabla \rho \right) - \nabla \cdot \left( \rho \cdot H \cdot \nabla p \right) = -\nabla \left( \phi_i \right) \tag{1}
\]

With no possibility to implicitly relax such equation in order not to corrupt mass conservation,
inlet boundary conditions become a major issue in solving such an equation. An inlet boundary
condition to respect the total pressure constraint was implemented implicitly correcting mass
imbalance in first cell layer to account for density variations at the boundary face.

Turbulence is taken into account in terms of RANS modeling with a wide selection of
turbulence models specifically suited for heat transfer analysis. The option to test different
turbulence models is crucial in case fluid nearwall behavior is of some interest. It is known for
example that the choice of the turbulence model strongly influences heat transfer prediction,
especially in proximity of recirculation zones, counter-pressure gradients, detached flows and many
other effects quite common in turbo-machinery flows. Moreover heat transfer predictions along
the blade are strongly dependant on inlet turbulence level. These are however data that experiments
often do not report with sufficient certainty. That is why a sensitivity analysis to the turbulence
model and the inlet turbulence dissipation rate length scale was conducted in order to establish the
most suitable values for such variable at the inlet.

In particular, due to the very detailed spatial discretization in the near wall region ($y^+<1$), it
was possible to use Low-Reynolds number turbulence models such as the one-equation Spalart-
Allmaras (SpAll in short), see Spalart et al. 1992, the two-layer k-ε (TL in short), see Rodi et al.
1991 also in the RNG form (TL-RNG in short), see Yakhot et al. 1992 and the k-ω SST (k-w in
short), see Menter 1993. For further details on solver and turbulence models implemented refer to

**Conjugate Heat Transfer Coupling**

The energy equation for the two domains is written in same form (static temperature) and
discretized in the same matrix. The full Navier-Stokes energy equation in fact reduces to the simpler
conduction equation in case the fluxes are null as they are in the solid domain.

\[
\nabla (\phi_i \cdot c_p \cdot T) - \nabla \left( k \nabla T \right) = \nabla (\phi_i / \rho_0 \cdot p) - p \cdot \nabla \left( \phi_i / \rho_0 \right) \xrightarrow{\phi_i \to 0} \nabla \left( k \nabla T \right) = 0 \tag{2}
\]

In this manner the coupling between the two sides of the interfaces can be easily treated in an
implicit way just discretizing the energy flux in terms of both solid and fluid adjacent cell thus
respecting at each iteration both the continuity of temperature profiles and the equality of thermal
fluxes across the interface. The coefficients of mutual interaction are directly calculated from the
conservation of energy across the boundary face weighting temperature at cell center with
conductivity and distance from the wall.

\[
\frac{k_f}{\Delta y_f} (T_f - T_w) = - \frac{k_s}{\Delta y_s} (T_s - T_w) \Rightarrow T_w = \frac{k_f \cdot T_f + k_s \cdot T_s}{k_f + k_s} \frac{\Delta y_f}{\Delta y_s}
\]

\[
(\nabla T)_{ws} = \frac{1}{\Delta y_s} \left( 1 - \frac{k_s}{\Delta y_s} \left( \frac{k_f}{\Delta y_f} + \frac{k_f}{\Delta y_f} \right) \right) \cdot T_s - \frac{1}{\Delta y_s} \left( \frac{k_f}{\Delta y_f} \left( \frac{k_f}{\Delta y_f} + \frac{k_f}{\Delta y_f} \right) \right) T_f \tag{3}
\]
This implicit technique allows faster convergence rates compared to standard explicit coupling due to the fact that energy balance is strictly respected at each iteration step, meaning that the temperature residuals in the solid only follow the implicit relaxation and not the update of the boundary conditions, basically not overloading non-conjugate calculations convergence rates. Parallelization is achieved decomposing the interface to maintain the two cells across the boundary face on the same processor. This special boundary condition can be applied only to a Low-Reynolds-number mesh on the fluid part because of a linear approximation can be considered valid in the thermal boundary layer only if the first fluid cell node is close enough to the wall. With such an assumption total and static temperature coincide close to the wall implying that this boundary condition can be extended also to total temperature equation.

Validation was performed on purely conductive tests as well as on a turbulent flat plate case (Xue 2005) with very good agreement with both correlative and theoretical results.

**Generic Grid Interfaces GGI**

In order to ease the mesh generation process, to allow a fully independent meshing of the different domains and to increase mesh quality in highly curved periodic interfaces a very general connection algorithm has been developed. Such an algorithm deals with multiple implicit coupling of unstructured mesh. A ghost cell is used to store the contributions of all the neighbor cells weighted on the face overlapping area. The ghost cell is updated at every iteration of the linear system solver therefore achieving pure implicit coupling between the two sides of the interface.

\[ \phi_r = W_p \cdot \phi_p + (1 - W_p) \cdot \phi_g \Rightarrow \phi_g = \sum_n \alpha_n \phi_n \quad (4) \]

A pre-processing tool is used to calculate and store both the addressing of the cells of mutual interaction and the weighting factors. Basically this tool calculates the overlapping area of the faces using an algorithm based on the definition of the winding number, valid for all polygons unless they are self-intersecting and have different orientation. If these two requirements are satisfied, and this is always the case if volumes of cells are positive, the integral of the product of the winding number of the two polygons is the intersection area. Obviously two polygons in space must be coplanar in order to share a finite portion of area, indeed this can be problematic in case of curved boundaries such as blade to blade periodic boundary. To prevent such failure of the algorithm the faces are first projected onto a common plane and so the projected overlapping areas are calculated. This does not invalidate the algorithm bearing in mind that the relative weight of the overlapping areas is actually the fundamental quantity to be calculated. If the overlapping area goes beyond a very tight tolerance the address of the cell is stored in the set of neighbor cells.

\[ \alpha_n = \frac{A^\text{overlap}_{\text{m-fp}}}{A_p} = \left( \frac{A^\text{overlap}_{\text{m-fp}}}{A_p} \right)_{\text{projected}} \quad (5) \]

This coupling can be applied to both coincident interfaces as well as periodic boundaries like rotational or translational cyclic but at the moment such treatment is not implemented for fluid-solid interfaces. It works in parallel exploiting MPI communication, the send and receive mechanism is built up in order to include only the nodes actually sharing a GGI boundary thus not overloading any of the processors with collecting and sending tasks. Multiple definition of GGI is also possible inside the same domain.
CONJUGATE CALCULATIONS

Simulation Setup and Computational Mesh

The numerical model simulates experimental setup of Hylton et al 1983 on a ten internal ducts cooled blade, see Figure 1 for a sketch of the geometry and the boundary conditions imposed.

Among the wide database available, two runs were chosen for the validation, a subsonic (run 112-4422) and a transonic one (run 111-5422). The subsonic case and the general setup of the computations was already described in Mangani et al 2009 so no further details will be given, regarding the transonic case a summary of the conditions can be found in Tables 1 and 2.

<table>
<thead>
<tr>
<th>$P_{in}$</th>
<th>$T_{in}$</th>
<th>$L_d$</th>
<th>$T_u$</th>
<th>$P_{Scat}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>307995</td>
<td>796</td>
<td>5e-6</td>
<td>0.083</td>
<td>192500</td>
</tr>
</tbody>
</table>

Table 1: Boundary condition imposed for the external gas path.

<table>
<thead>
<tr>
<th>N°</th>
<th>$M$</th>
<th>$T_\infty$</th>
<th>$P_\infty$</th>
<th>$L_d$</th>
<th>$T_u$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>.00771</td>
<td>415.81</td>
<td>100</td>
<td>.001</td>
<td>.05</td>
</tr>
<tr>
<td>2</td>
<td>.00621</td>
<td>417.75</td>
<td>100</td>
<td>.001</td>
<td>.05</td>
</tr>
<tr>
<td>3</td>
<td>.0062</td>
<td>399.15</td>
<td>100</td>
<td>.001</td>
<td>.05</td>
</tr>
<tr>
<td>4</td>
<td>.00658</td>
<td>407.29</td>
<td>100</td>
<td>.001</td>
<td>.05</td>
</tr>
<tr>
<td>5</td>
<td>.00661</td>
<td>404.36</td>
<td>100</td>
<td>.001</td>
<td>.05</td>
</tr>
<tr>
<td>6</td>
<td>.00686</td>
<td>444.68</td>
<td>100</td>
<td>.001</td>
<td>.05</td>
</tr>
<tr>
<td>7</td>
<td>.00636</td>
<td>401.55</td>
<td>100</td>
<td>.001</td>
<td>.05</td>
</tr>
<tr>
<td>8</td>
<td>.00225</td>
<td>486.1</td>
<td>100</td>
<td>.001</td>
<td>.05</td>
</tr>
<tr>
<td>9</td>
<td>.00137</td>
<td>539.68</td>
<td>100</td>
<td>.001</td>
<td>.05</td>
</tr>
<tr>
<td>10</td>
<td>.00077</td>
<td></td>
<td>100</td>
<td>.001</td>
<td>.05</td>
</tr>
</tbody>
</table>

Table 2: Boundary conditions imposed for the internal cooling channels

Regarding the computational mesh used for this work a very refined reconstruction of the boundary layer is necessary to well-capture heat transfer in the fluid domain. O-grid structure is imposed around the airfoil in order to guarantee orthogonal grid clustering, that is pushed down to reach values of $y^+$ below 0.5 all solid boundaries. The solid grid is composed by tetrahedral elements while for both the two fluid domains hexahedral cells were preferred. Total number of elements is around 1.3 million cells.

![Figure 1: Computational domain and boundary conditions used.](image-url)
ANALYSIS OF RESULTS

SUBSONIC CASE

Flow field predictions
In order to establish and validate the correct implementation of the newly introduced feature, namely the Generic Grid Interface algorithm, at first non-conjugate two-dimensional case at a fixed wall temperature was performed reproducing the already described set-up in terms of vane flow field. Continuity and smoothness of contour lines were checked across the cyclic interface together with the global conservation of mass. The non conformal periodic fluid-to-fluid interface strictly conserves mass, reducing imbalance up to the geometrical error induced by geometrical discretization. Such geometrical error is indeed not null only in case of non planar interfaces, in the case to be shown for example the relative difference in the sum of face areas is in the order of $10e^{-4}$ between the two sides of the interface. The smoothness of pressure contour plots across the interface can be appreciated in Figure 4.

Figure 4: Isolines and pressure contour plot: no discontinuities across the GGI interface.

Regarding the three dimensional calculation blade load predictions are tested against both experimental and numerical results obtained with the commercial CFD code STAR-CD v3.15 in a similar configuration, see Facchini et al. 2004. In particular non-dimensional pressure contour at mid span was available to compare the different models’ behaviour.

![Comparison between EXPERIMENTAL and CFD 3D results](chart.png)
As it is shown in Figure 5 the dependency of blade load on turbulence model is quite low and all models show a general good level of agreement with the experimental data, practically confining the only critical zone in the first half of the suction side where the highest pressure gradients are located. This can be ascribed to the excessive turbulent kinetic energy production around the leading edge which deeply affects viscous loss mechanism thus resulting in a weaker acceleration around the nose. Star-CD predictions in fact, due to a realizability constraint that limit turbulent kinetic energy production do not show such behaviour.

In order to give an overview of the predicted flow field in blade channel a Mach contour plot at midspan is reported, see Figure 6, visualizing a throat Mach very close to sonic condition. Also in this case agreement on pressure side between the different turbulence models is almost perfect, small differences can be appreciated only on the suction side and close to the trailing edge.

**Heat transfer predictions**

Even though previous section showed how pressure load is almost insensitive to turbulence modeling and the level of agreement obtained with experimental results is quite high, this cannot be considered true for heat transfer predictions. Due to the thin thermal boundary layer and the importance of the wall temperature gradient, good aerodynamic predictions are not sufficient to guarantee correct predictions of heat transfer phenomena. Usually this lack is attributed to boundary condition uncertainty and difficulty in finding a suitable turbulence closure: that is why several turbulence models were tested at the same conditions. The experimental results to match with are reported in terms non-dimensional wall temperature, moreover such temperature distribution was used to extract via an FEM solver a heat transfer coefficient profile.

As it may be noticed in Figure 7, the Two Layer models do not predict a local maximum in the values for heat transfer coefficient at the leading edge but with a typical fully turbulent behavior increase heat exchange on the suction side. While for the standard model the HTC is overestimated along the entire profile, the ReNormalization Group variant, reducing the level of turbulence near the wall, results in line with experimental data both at leading edge and downstream on the suction side. Better matching can be obtained using k-ω SST and Spalart Allmaras models.
Even though the two models seem to overlap on the pressure side with a very good agreement with the experimental investigation, the different nature of the two models is shown in the suction side. The k-ω SST in fact is following the turbulent trend to increase heat transfer coefficient downstream the leading edge but with a weaker level of enhancement compared to Two Layer models. This however results in an underestimation of heat transfer coefficient for $x/Ch_a > 0.5$. On the contrary Spalart-Allmaras model correctly predict laminarization on the early suction side but it shifts the transition point downstream compared to experimental evidence. As a consequence the turbulent increase of HTC is not sufficient to rematch experimental curve close to the trailing edge. It is interesting to point out how the HTC growth rate after transition occurs is the same as in the experiments, this is well promising for further development of the code: correctly predicting transition point could really improve the global level of agreement.

Figure 8 shows the profiles of wall temperatures: the trends of the previous graph are confirmed, assessing the standard Two Layer model, both Star-CD and OpenFOAM®, to higher levels of temperature, the k-ω SST and the Two-Layer RNG to an intermediate level and the Spalart Allmaras a little bit in underestimation. The model that globally matches better experimental results is the Spalart Allmaras with major misalignments downstream the suction side. The two intermediate models, basically shifts the standard Two Layer profile up to values that matches estimated temperature near the trailing edge better on suction than on pressure side.
TRANSONIC CASE

Flow field predictions
For the transonic case a single turbulence model was tested, namely the Two-Layer model. Even though this was not the best performing model in terms of heat transfer, the aim of this simulation was to compare the level of agreement obtained in subsonic regime with the predictions obtained at transonic regime. Since agreement with experiments was difficult to obtain also for subsonic case no improvememts are expected in the more difficult to predict transonic regime. Anyhow regarding the pressure distribution along the blade, the results do match quite well experiments all along the chord, see Figure 9. The agreement is almost perfect on the pressure side, on the suction side critical zones results to be the throath of the blade where the acceleration to supersonic regime is probably a little underestimated resulting in a higher pressure. Even though in terms of flow field analysis these are the only experimental values available, a contour plot of the Mach distribution at midspan is also reported to describe at least qualitatively the flow inside the channel. Two supersonic regions can be seen in Figure 10, the first one almost covering the entire blade throath, the second one developing downstream and laying on the terminal part of the suction side. Peak Mach reaches values around 1.2.

![Figure 9: Non-dimensional pressure profile at midspan.](image-url)
Heat transfer predictions
The conjugate heat transfer procedure is important because it gives a deep insight on heat transfer phenomena enabling the coupled estimation of the heat transfer coefficient and the wall temperature. Even though metal temperature distribution are also available as an output for the CFD calculation, experimental results do not offer any data with respect to the internal distribution of temperature or the heat transfer coefficient into the cooling ducts. That is why also results are reported in terms of heat transfer coefficient and wall temperature.
Looking at Figure 11, in where non-dimensional heat transfer coefficient is reported, the agreement with experiments seems to increase with respect to the subsonic case. The overestimation of the heat transfer coefficient along the suction side is much less pronounced and no shift is noticed on the pressure side.
Looking at wall temperature distributions in Figure 12, it is possible to assess that the profile quite closely reproduce the heat transfer distribution behavior meaning that the heat flux on the wall is more uniform. This is probably due to a more equilibrated internal cooling distribution as well as a less variable heat transfer coefficient.

**CONCLUSIONS**

A turbulence model assessment and the adaptation of OpenFOAM® for conjugate heat transfer of a turbine vane model have been performed both in subsonic and transonic regimes.

OpenFOAM® has been improved to predict heat transfer phenomena in gas turbine blade cooling: in details Generic Grid Interfacing and implicit conjugate heat transfer module have been developed and validated.
The reference test case studied was based on the NASA-C3X linear cascade, cooled by ten radial channels representative of a gas turbine first stage and results were compared with measured data. Loading distributions were found to be in good agreement with experiments for all turbulence models. At the same time the agreement with experimental measurements was good in terms of Heat Transfer Coefficient and wall temperature only for the Spallart-Allmaras turbulence model. The other models can partially reproduce the trends on the pressure side and on the downstream part of the suction side.

For all the models used as evidenced by measurements, both pressure and suction side exhibit a typical transitional behavior. For this reason, HTC profiles obtained without taking into account transition severely overestimate experimental data, especially near the leading edge, where a laminar region is present: results can be improved with the development and the applications of specific transition models.

Future work will be concentrated on expanding the capability of the code to simulate fluid-structure interaction with the new GGI boundaries between fluid and solid domains and to develop algorithms for the agglomeration in the Algebraic Multi Grid linear solvers specific for the GGI boundaries.

ACKNOWLEDGEMENTS

This work was conducted within a collaboration contract between the Energy Engineering Department of the University of Florence and the industrial partner ANSALDO Energia Spa.

REFERENCES


Facchini et al., “Conjugate heat transfer simulation of a radially cooled gas turbine vane”, Asme Turbo Expo, June 2004


Menter, “Zonal two equation k–w turbulence models for aerodynamic flows”, AIAA Paper(93-2906), 1993
Rodi, W., 1991., “Experience with Two-Layer Models Combining the k-\( \varepsilon \) Model with a One-Equation Model near the Wall”, AIAA-91–0216.