

## A FINITE VOLUME METHOD FOR THE CONJUGATE HEAT TRANSFER IN FILM COOLING DEVICES

Francesco Montomoli<sup>\*</sup>, Paolo Adami<sup>\*\*</sup>, Francesco Martelli  
Dipartimento di Energetica, Università di Firenze  
Via di S.Marta 3, 50139 Firenze, Italia

<sup>\*</sup> now at University of Cambridge, UK, <sup>\*\*</sup> now at Rolls-Royce plc, UK

### Abstract

The present work concerns the activity performed to upgrade an in house finite volume CFD solver for computing heat transfer in gas turbine cooling devices. The “conjugate simulation” of fluid heat transfer and metal heat conduction has been considered. To this aim the original code has been coupled to a new one solving the Fourier equation in the solid domain. This modification allows the “conjugate heat transfer” investigation in the fluid and solid domains at the same time.

The approach has been validated through two different test-cases. The first one is a two dimensional laminar flow over a flat plate, the second one is a film-cooled plate. A more complex application is also shown as an example of complete film cooled stage. In this example the complete cooling system of the nozzle has been modeled, including the two plenums and six rows of cooling channels as well.

### 1 INTRODUCTION

A reliable and accurate numerical prediction of temperature field in the metal of the hot components is becoming more and more important for the design of modern gas turbines. In this regard the first stage of the turbine and the combustor basket are two typical examples of concern because subjected to high heat transfer rates with hot gases. The turbine gases have temperatures above the melting point of alloys used and hot components can survive only using advanced cooling systems and metal coatings. An accurate prediction of metal temperature is essential to estimate the life of these components.

Focusing on the first NGV blades, the need for an accurate heat transfer prediction is highly required due to the constant increase of the turbine entry temperatures (TET). These have risen considerably from the beginning of the 1970's, around 1500 K, to the current values of modern turbines, over 2000 K. Several cooling techniques have come into use with the aim to maintain the integrity of the blades and to allow an acceptable engine life as well. Advanced

cooling techniques usually employed in the first stages of the HP turbine make use of internal convection, jet impingement and external film cooling at the same time. To predict the performances of such a complex system a numerical tool should be able to manage three dimensional geometries coupling the heat conduction in the metal to the heat transfer from the external fluid. The need for a coupled approach is justified by the evidence that the conduction in the solid alters the coolant flow temperature (Papanicolau et al., 2001[1]) while very few data are available in practical applications for guiding the detailed thermal design of these components.

In literature the conjugate heat transfer process is realized with two different strategies:

- by using an existing CFD solver that calculates the energy equation in the solid domain where zero velocity and constant density are imposed;
- by incorporating a solver for the temperature field in the solid domain in an existing CFD code.

The first technique does not demand strong modifications of original code and the management of the geometry is straightforward. The solver calculates the same set of equations in both the domains and uses the energy equation in the solid region to obtain the temperature distribution. The solid and fluid regions may be considered as single domain. The drawback of this procedure is that all the Navier-Stokes equations have to be solved in the solid region where only a simplified energy equation is needed. Moreover this method requires a careful attention to define the heat flux across the fluid-solid interface. In this region the diffusion coefficient changes sharply leading to a discontinuous temperature gradient. In order to overcome this problem Patankar (1980)[2] recommends to use harmonic averaging for cell face conductivity rather than arithmetic average.

In the second method the conjugate heat transfer is accomplished coupling an existing CFD solver with a code that solves the Fourier equation in the solid domain. This procedure can use meshes with the same interface or completely independent

meshes. The second choice demands an interpolating procedure to exchange the information through the fluid-solid interface. The development of the coupled approach requires some more implementation, but can allow the use of more appropriate schemes for the solution of the metal conduction and flow problem.

Luff and McGuirk (2001)[3] report an application based on the first method. To study the heat flux in the fluid-solid interface they use the numerical scheme proposed by Luff (2001)[3]. The unknown face temperature is obtained with a weighted-average formulation that takes into account the thermal conductivities of neighboring cells. This procedure gives good results in the test case proposed by Kelkar et al.(1991)[4] with heat conduction in a composite slab characterized by internal heat generation.

Rigby D.L. and Lepicovsky J. (2001) [5] have used the Glenn code to perform conjugate heat transfer simulations. Their approach extends the solver inside the metal imposing in that region a constant density and a nil velocity. In the interface between solid and gas the code set the same wall temperature to produce a consistent heat flux.

The second method has been used by Bock (2001)[6]. In his work Bock introduces a new object termed “virtual boundary conditions”. This new approach aims to detach the boundary condition problem from the computational mesh.

Croce (2000) [7] uses a coupling procedure between a research CFD code and a commercial software for the solid domain. He uses independent meshes and to transfer the information across the solid-fluid interface an interpolation procedure is applied.

He et al. (1995) [8] make use of a FDM/BEM coupled procedure, Finite Difference and Boundary Elements Methods. In their procedure the code initially solves the flow field along with an adiabatic condition on the fluid-solid interface. The actual conjugate heat transfer starts hereafter. The conduction equation in the solid region is solved using BEM with a boundary temperature obtained from the CFD calculation. The interface temperature is updated at each step on the solid/fluid interface faces by means of a weighted average of heat fluxes. Bohn et al. (1999)[9] followed a similar approach coupling a Navier-Stokes solver for the fluid zone to a thermal code solving the Fourier equation in the solid. In their paper, the author studied the thermal barrier coatings effect over an internal cooled blade.

In the present paper the conjugate approach implemented follows the second method of coupling.

The unstructured CFD solver HybFlow is therefore matched to a specific developed thermal solver Solid\_CHT. To keep a high accuracy, the approach requires that the two solvers share the same faces at the solid boundaries: in this way no approximation is added by the interpolation procedure. The HybFlow code has been selected because it can manage unstructured hybrid meshes. This kind of grids is ideal to model very complex 3D geometries maintaining at the same time a good discretization of boundary layer region. The HybFlow code has been previously validated for heat transfer prediction in film cooled blades or combustors chambers. The fluid solver can also deal with real flows, such as steam, or reactive gases but in this work the fluid used is air standard without chemical reactions. The thermal solver Solid\_CHT has been developed using some of the characteristics of the flow solver such as the parallel implementation in order to achieve a simpler coupling.

## NOMENCLATURE

T	Temperature
k	Thermal conductivity
Vol	Cell volume
n	Face normal
A	Cell Area
x, y, z	Cartesian Coordinates
R	Residual
$\epsilon$	Smoothing Coefficient
p	Lateral pitch
s	Axial pitch
M	Blowing ratio
u	Axial velocity
d	Hole diameter
	Subscripts
s	solid domain
f	fluid domain
w	wall of interface
b	solid wall
0	stagnation quantities

## 2 COMPUTATIONAL DETAILS

A brief description of the basic numerical scheme follows, while more details can be found in Adami et al. 1998[10]. The spatial discretization is based on a finite volume approach for hybrid unstructured grids. The Roe's approximate method or alternatively the AUSM+ flux splitting scheme may be used for the up-winding of convective fluxes. A linear reconstruction of the solution inside the elements provides a second order discretisation while and monotonicity is ensured through the TVD slope

limiter basically proposed by Barth, (1991)[11]. Turbulent viscous flows are represented by the conventional mass-averaged system of Navier-Stokes equations with the eddy viscosity determined according the two equations k- $\omega$  turbulent model proposed by Wilcox, (1993)[12] with realizability constraint, Durbin [17]. This correction helps solving the problem of extra turbulence production providing better results especially for accurate heat transfer predictions. Extra turbulence production is caused by the unbounded growth of turbulent kinetic energy in the presence of moderate initial levels and large rates of strain, such as near stagnation points: “stagnation point anomaly”, or in the presence of a developing boundary layer. This correction consists of a control of the turbulent time scale (Durbin [25]).

The addition of the conjugate heat transfer has been accomplished with a new solver integrated in the code, Solid\_CHT. This routine solves the heat transfer Fourier’s equation inside the solid zone through an explicit time marching scheme. The spatial discretisation follows a finite volume approach and can manage indifferently structured and unstructured grids. Concerning the spatial derivatives they are shown in eq 1 and 2:

$$\overline{k\nabla^2 T} = \frac{1}{V_{ol}} \int_{V_{ol}} k\nabla^2 T dV = 0 \quad (1)$$

$$\int_{V_{ol}} k\nabla^2 T dV = \iint_A k \left( \frac{\partial T}{\partial x} n_x + \frac{\partial T}{\partial y} n_y + \frac{\partial T}{\partial z} n_z \right) dA \quad (2)$$

Therefore the Laplace equation is integrated over each element of the grid. As usual, the surface integral is split into a summation over the plane faces of the elements. On each plane a mid-point quadrature formula is applied with the first derivatives of the temperature computed on the face mid-point by a least square second order method. An explicit time marching approach is finally used to update the solution, eq 3:

$$T_i^{n+1} = T_i^n + R_i^n$$

$$R_i^n = \frac{\Delta\tau}{V} \sum_{j=1}^m k_{i-j} A_{i-j} \frac{T_j^n - T_i^n}{\Delta X_{i-j}} \quad (3)$$

The convergence rate of the time-marching method is enhanced by implicit residual smoothing through the following Poisson equation:

$$\overline{R}_i = R_i + \varepsilon \cdot \Delta^2 \overline{R}_i \quad (4)$$

Here  $\overline{R}_i$  is the smoothed residual,  $\Delta^2 \overline{R}_i$  is its laplacian and  $\varepsilon$  a smoothing coefficient to be chosen according both to the grid the solver Courant number and the features of the thermal field.

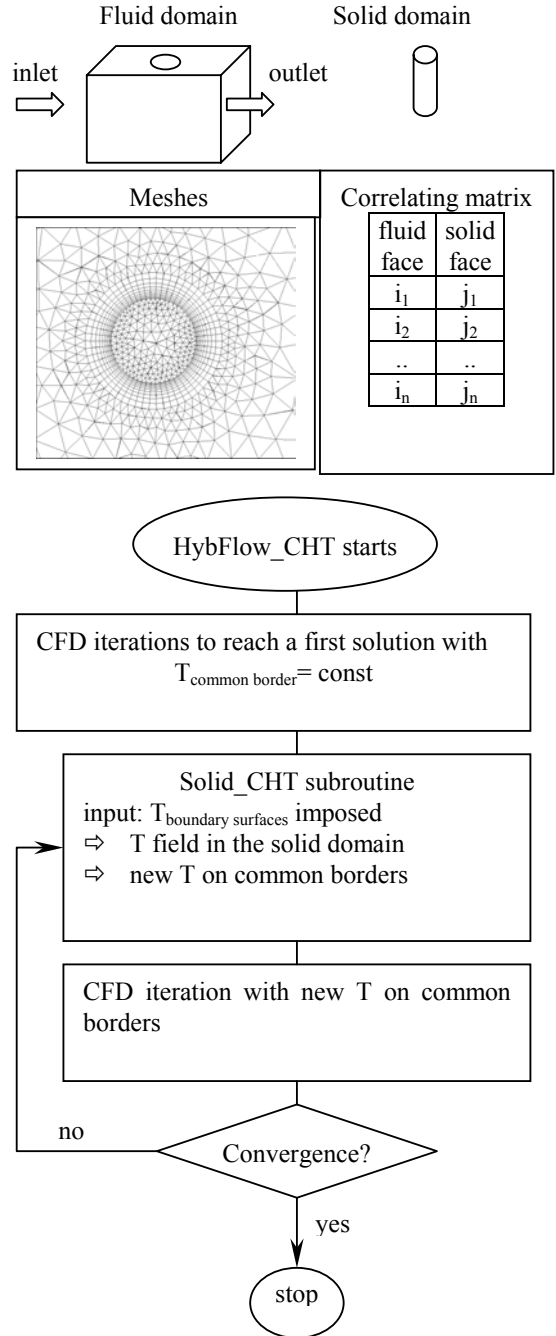


Fig.1- The CHT procedure

The iterative Jacobi method is used as linear solver in the smoothing scheme. The conjugate heat transfer problem starts with the generation of two solid models for the fluid and for the solid domains. To ensure a simple and accurate matching, the same grid generator (Centaur<sup>TM</sup>) has been considered to model the two physical domains. Concerning the fluid zone the core of the flow is tessellated using tetrahedrons, while regular prismatic layers cover solid walls to improve the solution accuracy in the viscous boundary layer. For the solid zone only tetrahedrons are used with the constraint to have an exact matching with the flow interface elements.

Accordingly the two domains are directly coupled imposing the same temperature as boundary condition on the common faces. Some pre-processing routines are also used to prepare all metrics information needed by the solving routines. At the end of this preprocessing phase all the geometrical data are available to run a scalar multi-blocks solver or a parallel job. The correlation array needed to link the common faces in the solid and fluid grids are also provided by the pre-processing routines. It is worth mentioning that the grids handled by the two solvers do not follow the same numeration. The main steps required may be synthesized in Fig.1. As already mentioned a first flow solution is achieved using an isothermal temperature distribution on the solid boundaries, reducing the overall computational time. Then the subroutine Solid\_CHT solves the temperature distribution inside the solid with the same wall temperature imposed at the interface.

Using the fluid and metal temperatures computed in the adjacent elements of the interface, a new wall temperature distribution is obtained imposing the balance of heat fluxes. The updated wall temperature  $T_w$  is retrieved from the following relation, eq 5:

$$k_s \frac{T_s - T_w}{\Delta n_s} = k_f \frac{T_w - T_f}{\Delta n_f} \quad (5)$$

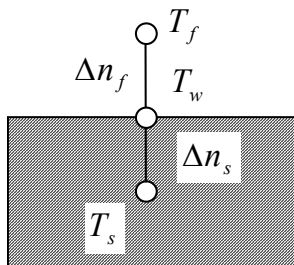


Fig. 2- Heat fluxes at interface surface

where  $T_s$  and  $T_f$  are the temperatures computed in the adjacent cell at the wall interface for the solid and fluid domain respectively.

The approach proceeds iteratively until the convergence by solving both the fluid domain and the solid one with the new wall temperature at the interface. For the present work the thermal conductivity was assumed constant although it could be a more complex distribution to model the presence of thermal barrier coatings.

The solver takes advantage of the same multi-block parallel strategy of the CFD code that allows a significant reduction of the overall CPU costs. The parallel solver is based on the standard MPI message passing libraries to ensure the portability.

### 3 VALIDATION

In order to assess and validate the scheme two basic applications are herein reported. The first is the laminar flow over a flat plate with conjugate heat transfer between the gas and the plate itself. The second is a film cooled flat plate with seven rows of cylindrical holes. The first test is intentionally laminar to decouple the validation of the scheme from the turbulence modeling problem.

#### Flow field over a flat plate

The flow over the plate is obtained considering air with a total temperature of  $T_0=1400\text{K}$  and a Mach= 0.13. The non-dimensional temperature  $T_0$  used in the graphs is the total temperature. The plate is 100mm thick cooled from below. The wall temperature of the cooling surface is imposed  $T_b=600\text{K}$  and the thermal conductivity of the plate is  $k_s=4\text{W/mK}$ . The velocity profile on the inlet surface is flat. To validate the code a comparison of CFD results to analytical solution of Luikov[13] has been performed. This analytical solution has been chosen because it gives the temperature profile normal to the wall in both fluid and solid field.

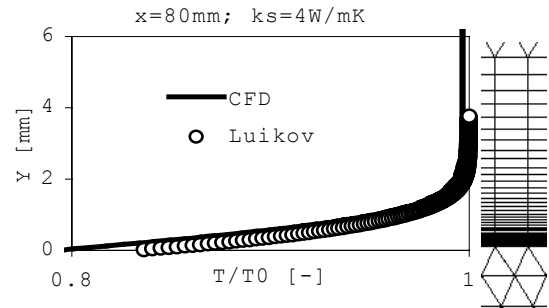


Fig.3- T distribution at x=80mm

Part of the grid used in this case is shown in fig.3. The hybrid mesh uses 50 prismatic layers with the first node within  $y^+ < 1$ . The boundary layer edge is inside the prismatic layers until the domain exit.

A grid sensitivity analysis has been carried out with three grids for fluid and solid but the results do not show any appreciable variations. The agreement between the analytical and numerical solution is satisfactory.

### Flow field over a film cooled flat plate

The second test case used is a flat plate with seven staggered rows of coolant holes. This configuration is a representative test of a full coverage film cooling systems used in modern combustor. Moreover the present case has been computed by other authors (see Papanicolau et al. 2001 [1]) and therefore represents a valuable reference for the assessment of a CHT solver. The experimental investigation has been carried out by Martiny et al.(1998)[14]. The flat plate is 12 mm thick with an injection angle for the coolant channels of  $17^\circ$ . The diameter of the hole is 4mm and the pitch is  $p=8.96$ mm in the z direction and  $s=29.84$ mm in the x direction. The solid model is shown in Fig. 4.

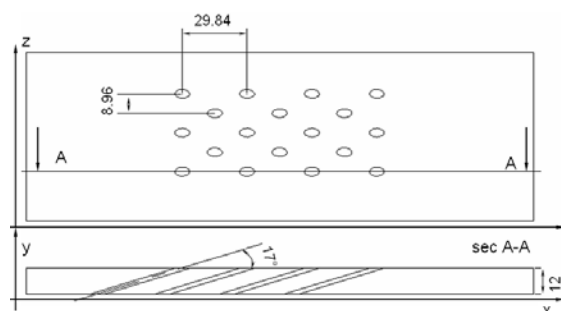


Fig. 4- Film cooled flat plat, dimensions in mm

From experimental data, inlet conditions for hot gas are:

- constant velocity  $u=28$ m/s at 12 mm distance from the test plate, y-wise;
- the total temperature distribution along y follows a power law from 475K, at the plate surface, to an asymptotic value of 550K, far away from the plate;
- turbulence level 4%.

The inlet conditions for coolant flow are:

- temperature 315K;
- turbulence level 5%.

Furthermore, the scale length is set 1% of the inlet height for inflow boundaries. As demonstrated by the present application, as well as in A.A. Chernobrovkin and B. Lakshminarayana [17], the effect of turbulence length scale of the main-flow has a major

influence on the flow development and heat transfer downstream the cross-flow ejection. The solid conductivity coefficient is  $k_s=12.2$ W/mK (Incoloy(800H)). The simulation is carried out for a blowing ratio  $M=1.2$ .

In Fig.5 the lateral averaged cooling effectiveness  $\eta$  is compared to the experiments. Its definition is:

$$\eta = \frac{T_0 - T_w}{T_0 - T_c}$$

where  $T_0$ =main temperature,  $T_c$ =coolant temperature,  $T_w$ =wall temperature. In fig. 5 there is also the lateral average value of  $Y^+$ , dashed line. The value of  $Y^+$  allows the integration of turbulent quantities at wall without using wall functions.

A grid sensitivity analysis has been performed.

The final mesh consists of 1.100.000 of elements for the fluid region and 340.000 for the solid one. About 20 prismatic layers have been used to model the fluid boundary layer. Only tetrahedral cells have been used for the solid domain.

The numerical and experimental results are very similar in the first half of the plate. In the rear part, as shown also by Papanicolau [1], the difference is generated by the boundary conditions imposed on the numerical model. More precisely, the adiabatic condition at the extreme surfaces of the blade does not represent the real conditions of the test. The effect of this approximation is particularly felt in the  $\eta$  distribution shown in Fig. 5 for  $x/d > 40$ . Unfortunately the actual boundary condition to be imposed is not available.

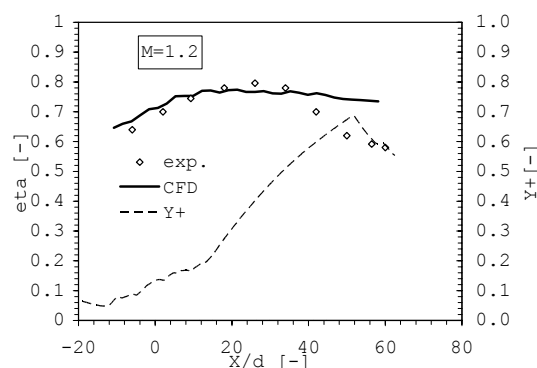


Fig.5- Eta laterally averaged with  $Y^+$  distribution

The temperature traverses are reported in Fig. 6 along with experimental data. Three axial ( $x/d$ ) positions are considered for  $z=0$  mm where  $d$  is the cooling hole diameter. As mentioned the solution close to the plate leading edge shows some inaccuracy in predicting the axial development of the

boundary layer that it is also felt in the temperature traverse for  $x/d=0.75$ . The shape of the thermal boundary layer is correct although the height is somewhat under predicted. Proceeding downstream the agreement improves until  $x/d \approx 40$ . From here the effect of the wall adiabatic boundary condition becomes dominant.

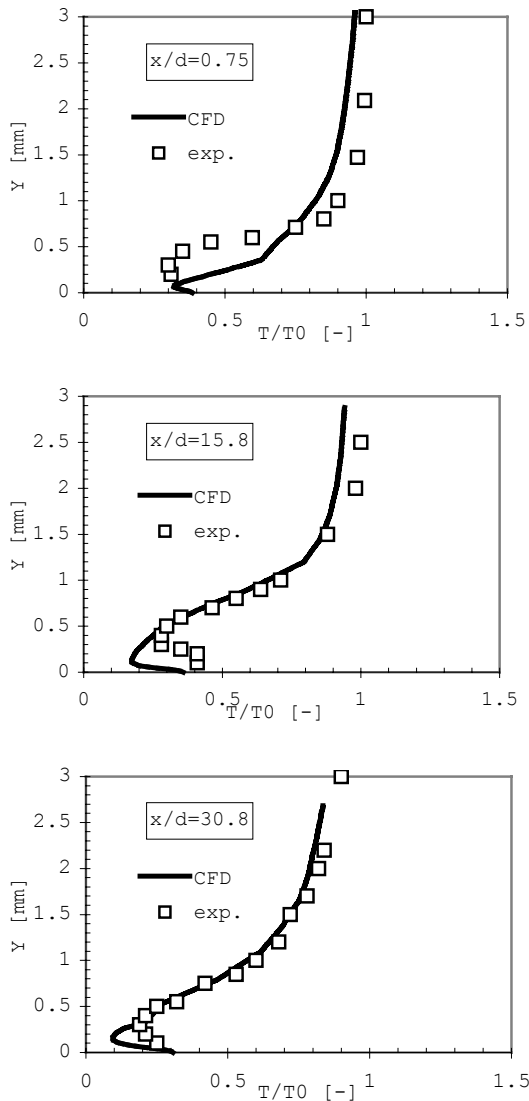


Fig.6 -  $T/T_0$  normal to wall

In Fig. 7 the cooling effectiveness on the upper surface is shown. As expected the convergence of the CHT approach needs several iterations on both the fluid flow and metal side. The high number of iterations required to convergence has been attributed to the presence of very small fluid elements at the solid interface. These small layers of elements are

dominated by viscous effects and have a slow convergence rate.

This effect actually decouples the two domains explaining the “ill-conditioning” observed for the whole system. The basic strategy used in the computation consists in starting the flow field with an isothermal or adiabatic wall condition. The thermal computation inside the metal is started is a second step just before convergence of the external flow domain.

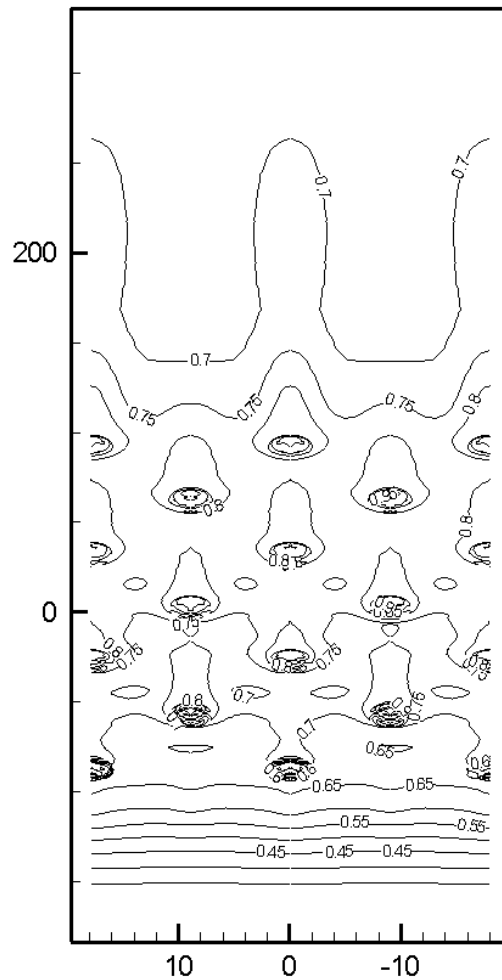


Fig.7 – Eta distribution

Concerning the Solid\_CHT solver, at each fluid iterations, the advancement in time is carried out through a quite high number of explicit iterations in order to allow a nearly converged solution of the metal thermal field. This approach resulted in a stable convergence rate of the whole approach without the appearance of un-damped oscillations of the interface temperature. No relaxation has been needed in the wall temperature update, while the high number of

iterations for the Solid\_CHT solver has produced an acceptable CPU time requirement in comparison to the single fluid iteration cost. The overall computational time is approximately 36 hours on a DS20 Workstation.

#### **Example of application to a film cooled Stage-QinetiQ**

The test configuration used is the transonic MT1 HP stage investigated at QinetiQ, UK, Chana and Mole, 2002 [15]. The rotor rotational speed is 9500 RPM, with a pressure ratio of about 2.8. The stage has 32 NGV and 60 rotor blades. The stage calculation has been carried out using a steady mixing plane approach. The geometry and the surface meshes are shown in fig. 8.

The nozzle is film-cooled with six rows of cylindrical holes with two double rows placed on the pressure side and two single rows on the suction side. The internal diameter of the channels is 0.6mm with 1.8mm pitch and 50% of inclination angle. Two plenum channels inside the blade with a different total pressure deliver the coolant flow, the front of 10 mm of diameter, the rear of 7 mm. The same higher-pressure supply plenum feeds the two rows on the suction and pressure side that are closer to the leading edge. The rear plenum has a lower total pressure and feeds the two successive rows encountered on the pressure and suction side of the blade. In fig.9 the model of the solid domain is shown. Concerning the working conditions the following data have been used in the simulation of the vane:

- ✓ upstream  $P_0 = 4.6$  bar
- ✓ upstream  $T_0 = 444$  K
- ✓ front channels  $P_0 = 6.28$  bar
- ✓ rear channels  $P_0 = 4.88$  bar
- ✓ coolant  $T_0 = 286.5$  K
- ✓ exit average  $P = 2.48$  bar
- ✓ exit: radial equilibrium;
- ✓ wall temperature = 288 K;
- ✓ inlet Tu level = 5.5 % ;
- ✓ metal conductivity  $k=12.2$ W/mK ;

The simulation includes the whole fluid domain with the coolant interior channels. The objective of this application is twofold: to verify the flow field changes when conjugate heat transfer is accounted for; to determine the hottest metal parts of a typical transonic HP stage. The comparisons with the experiments have been carried out for the isentropic Mach number distribution and the Nusselt number over the external surface. The usual solver for the external flow has already successfully applied to this blade (Adami 2003 [16]) and these results have been used as comparison in the present work.

A grid sensitivity analysis has been carried out for the fluid region. Three different meshes for the nozzle have been compared with 1,000,000, 1,800,000 and 2,200,000 elements. Using a previous set of results obtained on the same geometry 20 prismatic layers has been used to solve the boundary layer over all solid surfaces. The solid region of the nozzle counts approximately 400,000 elements.

For the rotor, the solid mesh consists of 250,000 elements, and about 700,000 for the fluid. The interaction between NGV and rotor has been solved with a mixing plane approach.

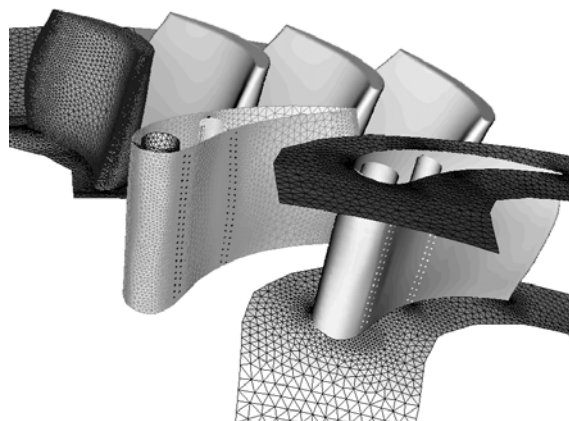


Fig.8 surface mesh of MT1 HP stage

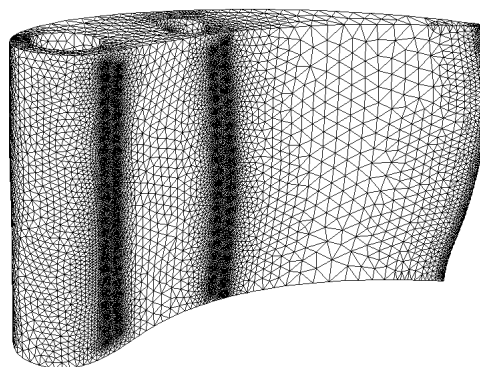


Fig.9- Mesh of the Solid Model

The isentropic Mach number distribution at mid span for the NGV cooled blade is shown in Fig.11. The agreement with experiments is satisfactory and no appreciable difference has been detected comparing the results from CHT with conventional external CFD. The isentropic Mach number distribution shows a weak shock wave on the rear part of the suction side ( $x=30$ mm). The Mach number field shown in Fig.10 confirms the presence of this flow structure.

The coupling between solid and fluid domain can be appreciated in fig.11 where the non-dimensional temperature distribution is shown for both the domains at midspan.

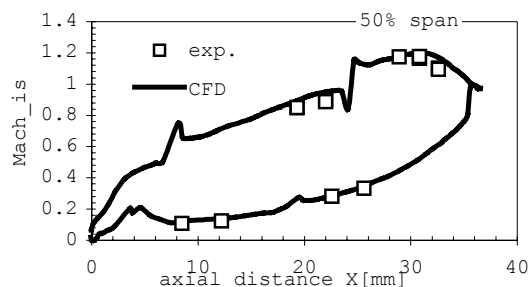


Fig.10- Isentropic Mach number

The iso-contours lines are continuous at the interface. The coolant presence can be clearly detected in fig. 11. The dark region indicates low temperature zone and consequently the coolant presence. It is possible to observe that around the two plenums the metal temperature is quite high and that the temperature of the coolant increases moving from the plenum to the blade surface. This mitigation effect induced by the metal conduction has a beneficial effect on hot components life: by reducing the thermal gradients there are fewer problems of thermal shocks and/or thermal stresses.

A simulation of the same components using a standard CFD code tends to underpredict the component life overpredicting the thermal gradients.

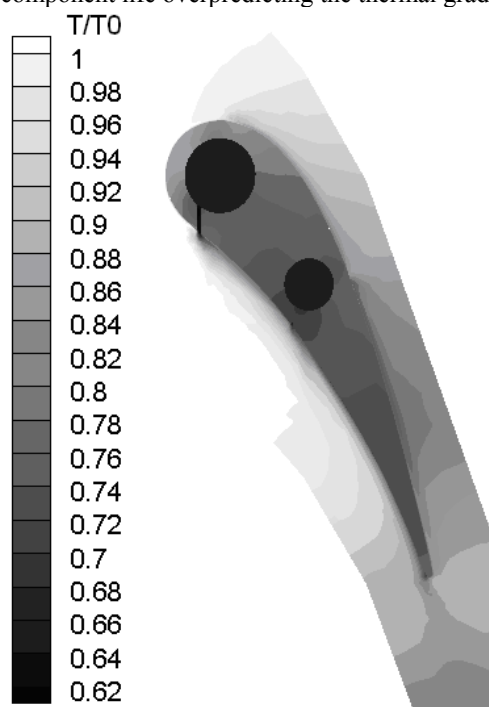


Fig.11-  $T/T_0$  on fluid and solid domain

The Nusselt number of CHT simulation has been compared against experiments and the CFD simulation with the dense and medium mesh. The comparison of CFD on dense and coarse mesh shows that the solution is almost grid independent except on the rear side of the pressure side where a denser mesh maybe has to be used. A general agreement is achieved on the whole profile. More precisely on the pressure side the CHT gives a better accuracy while on the suction the heat transfer rate appears somewhat lower. Both the approaches suffer from a high turbulence overproduction near the blade nose which is felt on the crown as a Nusselt over prediction, the zone in the circle of figure 17.

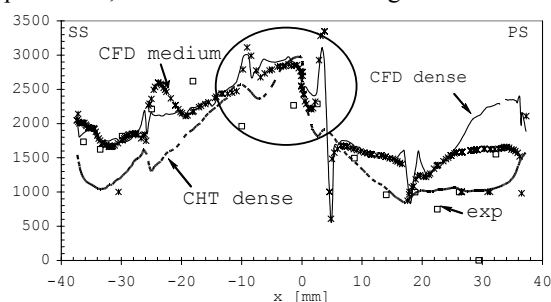


Fig.12- Nusselt at mid span

#### 4 CONCLUSIONS

In this paper a solver for the conjugate heat transfer has been discussed. The approach proposed (the HybFlow\_CHT code) is aiming to handle very complex configurations for the design of the hottest turbomachinery components. In order to validate the solver two basic problems have been successfully considered. The numerical results show a good agreement both with the analytical solution and the experimental data available for each case.

The last application has been applied trying to assess the flexibility, robustness and accuracy of the code for the investigation of a complete high pressure stage including the coolant ejection channels, the plenum supplies and the solid metal. The results in terms of heat transfer distribution are accurate and encourage proceeding with further activity in this direction.

Finally the approach has demonstrated high flexibility, satisfying accuracy and stability. The main drawback consists of the high number of iterations needed to reach the convergence. In this regard few improvements have been presently achieved through the use of different updating strategies of the interface temperature.



## ACKNOWLEDGMENTS

The authors wish to acknowledge for financial support as well as the contribution of the industrial partners the Brite-EuRam TATEF project (BRPR-CT97-0519).

## REFERENCES

1. **Papanicolau E., Giebert D., Koch R., Schulz A.**, "A conservation-based discretization approach for conjugate heat transfer calculations in hot-gas ducting turbomachinery components", International Journal of Heat and Mass Transfer 44, 2001, 3413-3429
2. **Patankar S.V.**, "Numerical heat transfer and fluid flow", Hemisphere, Washington, DC, 1980
3. **Luff I.K., J.J. McGuirk**, Conjugate heat transfer predictions of a combustor heatshield containing pedestals, NATO RTO-AVT Symposium on advanced flow management Norway, 2001
4. **Kelkar, K.M., D. Choudhury, M. Ambrosi**, "Numerical method for the computation of conjugate heat transfer in non-orthogonal boundary-fitted coordinates", Numerical Heat Transfer, Part B, 20, pp. 25-40, 1991
5. **Rigby D.L., J. Lepicovsky**, "Conjugate heat transfer analysis of internally cooled configurations", ASME TURBO EXPO, New Orleans, 2001
6. **Bock S.**, "Approach for coupled heat transfer / heat flux calculations", RTO-Symposium on heat transfer and cooling in propulsion and power system 2001.
7. **Croce G.**, "A Conjugate Heat Transfer Procedure for Gas Turbine Blades", in "Heat Transfer in Gas Turbine Systems", R.J. Goldstein, ed., Annals of NY Academy of Sciences, Vol.934, pp.273-280, 2001
8. **He M., A.J. Kassab, P.J. Bishop, A. Minardi**, "An iterative FDM/BEM method for the conjugate heat transfer problem - parallel plate channel with constant outside temperature", Engineering Analysis with boundary Elements, Vol. 15, 1995
9. **Bohn D., V.J. Becker, J.T. Kortmann**, "Numerical conjugate flow and heat transfer investigation of a transonic convection cooled turbine guide vane with different thermal barrier coatings", IGTC Kobe, 1999
10. **Adami, P., Martelli F. and Michelassi V.**, 2000, "Three-Dimensional Investigations for Axial Turbines by an Implicit Unstructured Multi-block Flow Solver", ASME, IGTI TurboExpo 2000.
11. **Barth T.J.** "Numerical Aspects of Computing Viscous High-Reynolds Number Flows on Unstructured Meshes", AIAA Paper 91-0721, Jan. 1991.
12. **Wilcox D.C.** "Turbulence modeling for CFD" DWC Industries, Inc., La Canada, 1993
13. **Luikov A.V.**, "Conjugate convective heat transfer problems", International Journal of Heat and Mass Transfer 17, 1974, 1207-1214
14. **Martiny M., R. Schiele, M. Gritsch, A.Schulz, S.Kim**, "High efficient cooling concepts for low emission combustors", Collaborative research center 167, High Intensity Combustors- Steady Isobaric Combustion 1998
15. **Chana, K.S., Mole, A.H.**, "Summary of Cooled NGV and Uncooled Rotor Measurements from the MTI Single Stage High Pressure Turbine in the DERA Isentropic Light Piston Facility", Brite-EuRam TATEF project (BRPR-CT97-0519), 2002.
16. **Adami P., F. Martelli, F. Montomoli, K.S. Chana**, "Numerical predictions of film cooled NGV blades", ASME, IGTI TurboExpo 2003.
17. **Durbin, P.A.**, "On the k- $\epsilon$  Stagnation Point Anomaly", Int. J. Heat and Fluid Flow, 1996, 17, 89-90