CONJUGATE HEAT TRANSFER ANALYSIS OF A TRI-DIMENSIONAL TURBINE BLADE INTERNAL CAVITY

Guillaume Lassaux, Stéphan Daux and Laurent Descamps
Snecma Moteurs
Turbine Aero and Mechanical Design Department
Centre de Villaroche,
77550 Moissy-Cramayel (France)

keywords :*Computational Fluid Dynamics, High Temperature Materials and Structures*

Abstract

An interaction model between the fluid and solid media is generally required in turbine configurations but remains a difficult issue. A coupling procedure between a Navier-Stokes code and a conduction solver is therefore the only way to achieve heat transfer prediction in all flow situation. The objective of this work is to present such a procedure , which has been developed by Snecma and based on a Finite Volume Navier-Stokes code and a commercial Finite Element solver. To demonstrate the quality of the procedure, a conjugate heat transfer computation in a turbine blade internal cavity is described in detail.

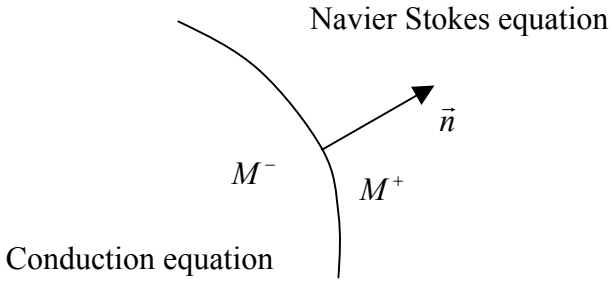
Nomenclature

C^s	solid calorific capacity
C_p	fluid specific heat
L	geometric reference scale
q_w	wall heat flux
T	wall temperature
$\mathrm{Pr} = \frac{\mu C_p}{\lambda}$	Prandtl number
$\mathrm{Re} = \frac{\overline{\rho}\overline{U}L}{\overline{\mu}}$	Reynolds number
ρ	density
λ	conductivity
μ	viscosity

Superscripts ^{s} and ^{f} refer to *solid* and *fluid* quantities respectively.
Subscript $_w$ refers to wall quantities.

1. Introduction

On dealing with heat transfer in aircraft engines, all thermal interaction effects between the solid and fluid media should be written in terms of temperature and heat transfer continuity at the fluid-solid interface. Boundary conditions to be applied to the fluid and solid domains are summarized in Figure 1.



$$\begin{cases} q_w^f = -\lambda^f \left.\frac{dT}{dn}\right|_{M^+} = q_w^s = -k^s \left.\frac{dT}{dn}\right|_{M^-} \\ T_w^f = T_w^s \end{cases}$$

Figure 1 – Conjugate heat transfer problem

In the case of turbine blade, heat transfer between solid and fluid media is usually predicted by the following procedure:

1. perform an uncoupled **Navier-Stokes** (NS) calculation with fixed thermal boundary conditions,
2. derive a convective **heat transfer coefficient** α from the previous NS results, as follows :

$$\alpha = \frac{q_w^f}{T_w^f - T^*}$$

where T^* is a reference temperature field such that wall heat flux $q_w^f = 0$ if $T_w^f = T^*$,

3. perform a **conduction calculation** using the (α, T^*) wall field as a convective heat transfer boundary condition :

$$q_w^f = \alpha(T_w^f - T^*) \qquad (1)$$

This procedure can be iterated until convergence is obtained :

$$q_w^f = q_w^s \qquad (2)$$

$$T_w^f = T_w^s \qquad (3)$$

In most cases, the convective heat transfer coefficient is weakly dependent on wall temperature. Convective heat transfer coefficient â and reference temperature T^* can be estimated with correlation based on Re and Pr numbers. The key issue is finding a systematic way of computing, at each iteration, a reference temperature field T^* which ensures positive α values and satisfies $\alpha(T_w^f - T^*) = 0$ when the wall heat flux q_w effectively narrows to zero and changes sign.

To circumvent this problem, Montenay [1] suggested to perform simultaneously both solid conduction and Navier-Stokes calculations and force wall temperature and (α, T^*) exchanges regularly during their iterative processes. This coupled calculation is driven by (α, T^*), which are trimmed to ensure stability and high convergence rate while satisfying Equation (1). Provided that the iterative procedure converges, the physical meaning of couple (α, T^*) is of no importance since equalities (2) and (3) are satisfied by both fluid and solid temperature fields at the coupled interface.

We will first describe both fluid and conduction solvers involved in the coupling procedure based on Montenay's work and used at Snecma Moteurs. Then we will present an example of conjugate heat transfer analysis in a 3D turbine blade cavity.

2. Coupled solvers

Navier-Stokes solver

The Navier-Stokes code, MSD, has been developed by ONERA. Snecma uses this fluid solver for the prediction of compressible turbulent flows and heat transfer phenomena in engine cavities or blade internal cooling systems.

The standard turbulence model of the code is a two equations Boussinesq Viscosity Model.

The global system treated by the solver is of the usual form :

$$\frac{\partial u}{\partial t} + div\left(\vec{F}_e(u) + \vec{F}_v(u)\right) = S(u) \qquad (4)$$

where u is the flow variables vector, which includes two entries for both kinetic turbulent energy and turbulent characteristic length. $\vec{F}_e(u)$ is the Eulerian flux tensor, $\vec{F}_e(u)$ the viscous flux tensor, which includes turbulent viscosity effects, and $S(u)$ the source terms vector.

This system is discretized with a structured cell-centered finite volume technique. The spatial discretization uses second order MUSCL extrapolations on the variables u with TVD Flux Difference Splitting for the Eulerian fluxes. The viscous fluxes are computed at the center of each control volume. Those fluxes are then interpolated at cell interfaces. The time integration scheme is a backward implicit Euler scheme. Details can be found in [2].

Conduction solver

The conduction solver used at Snecma is the commercial finite element code ABAQUS. For our 3D configurations, we use hexahedral finite elements. ABAQUS can solve steady and transient problems via a backward Euler implicit scheme. The numerical algorithm is a Newton method for non-linear problems, which converges in one iteration when the response of the discretized system is linear. Details can be found in [3].

3. Coupling algorithm

Conforming to the fluid solver numerical scheme, transient coupled heat transfer computation seems a possible direction to solve the coupled heat transfer problem. However it can be shown that the convection time scale t_{fv} and the conduction time scale t_{fc} of the Navier-Stokes solver satisfy :

$$\frac{t_{fv}}{t_{fc}} = \frac{\rho^f C_p l^2}{\lambda}\frac{U}{L} = \frac{\mu C_p}{\lambda} = \Pr \qquad (5)$$

where $l = \mathrm{Re}^{-\frac{1}{2}} L$ is the fluid conduction length scale. t_{fv} is the same order of magnitude as t_{fc}. Besides, the conduction time scale of the Navier-Stokes solver and the conduction time scale of the solid solver satisfy the following equation :

$$\frac{t_{sc}}{t_{fc}} = \frac{\rho^s C^s L^2}{\lambda^s}\frac{\lambda^f}{\rho^f C_p l^2} = \frac{\rho^s C^s \lambda^f}{\rho^f C_p \lambda^s}\mathrm{Re} \qquad (6)$$

Typical values for the properties of steel ($\lambda^s \approx 20\ W.m^{-1}.K^{-1}$, $C^s \approx 500\ J.kg^{-1}.K^{-1}$ and $\rho^s \approx 10000\ kg.m^{-3}$) and air ($\lambda^{fs} \approx 0.03\ W.m^{-1}.K^{-1}$, $C_p \approx 1000\ J.kg^{-1}.K^{-1}$ and $\rho^s \approx 1\ kg.m^{-3}$) lead to :

$$\frac{t_{sc}}{t_{fc}} = 7\,\mathrm{Re} \qquad (7)$$

The characteristic time scale of the Navier-Stokes solver is much smaller than the time scale of the conduction solver. Each temperature variation requires an important effort from the fluid solver, which would drive the time decomposition to small steps. As a consequence, transient explicit fluid/solid coupling, which performs a FEM calculation at each time step, would be prohibitively CPU time-consuming.

Montenay developed a loosely coupled procedure that allows the fluid solver to perform a fixed number of iterations N^f between two exchanges, while the solid conduction solver only computes one steady state step.

Montenay also showed that transmitting heat flux from the fluid to the solid domain leads to unstable models.

A stable alternative consists in imposing, at coupling step *n*, $(\hat{a}, T^{*n})$ at the solid wall, while prescribing $T_w^{f\,n} = T_w^{s\,n-1}$ at the fluid interface, where $T_w^{s\,n-1}$ is the wall temperature computed by the conduction solver during step *n-1*.

To circumvent the difficulty of defining an ambient temperature field, an a priori heat transfer coefficient $\hat{a}$ is used to compute the reference temperature T^{*n} at step *n*, as shown below :

$$q_w^{f\,n-1} = \alpha(T_w^{f\,n-1} - T^{*n}) \qquad (8)$$

where $q_w^{f\,n-1}$ is the heat flux computed by the fluid solver at step *n-1*, and $T_w^{fn-1} = T_w^{s\,n-2}$ the wall temperature previously imposed to the fluid, at the beginning of coupling step *n-1*.

The actual value chosen for $\hat{a}$ and T^* has no influence on the solution obtained once the procedure has converged. Moreover this method can be viewed as a relaxation approach :

$$q_w^{s,n} = q_w^{f,n-1} + \alpha\left(T_w^{s,n} - T_w^{f,n-1}\right) \qquad (9)$$

The second term of the right side of the above equation prevents the solid temperature $T_w^{s,n}$ from deviating too much from the temperature $T_w^{f,n-1}$ prescribed at the fluid interface.

The numerical heat transfer coefficient $\hat{a}$ is kept constant during the successive coupling steps and along the whole fluid-solid interface. On the solid side, a steady-state conductivity calculation is performed at each coupling step *n*, using the

couple $(\hat{a},T^{*n})$. Experience shows that greater values for $\hat{a}$ lead to more stable calculations but also to slower convergence rates for the coupled problems. $\hat{a}$ results from a compromise between stability and convergence rate.

The heat transfer coefficient $\hat{a}$ is the first parameter required to drive the coupled problem to convergence. The calculation is also dependent on the number of fluid solver iterations N^f. It has been found that it is not necessary to lead the NS finite volume calculation to convergence, at each coupling step. However, to get convergence of the coupled problem in reasonable time, N^f has to be high enough to allow information to spread out in the fluid during each coupling step.

Montenay's coupling procedure is sketched in Figure 2. From each exchange, the FEM solver receives $(\hat{a},T^{*n})$ and performs a steady state calculation in one Newton iteration. Meanwhile, the fluid solver performs N^f iterations with the prescribed wall temperature condition $T_w^f = T_w^s$.

4. Example : a tri-dimensional turbine blade cavity

To demonstrate the potential of the coupling procedure described above, a coupled computation was performed on an internal turbine blade cavity at full power conditions.

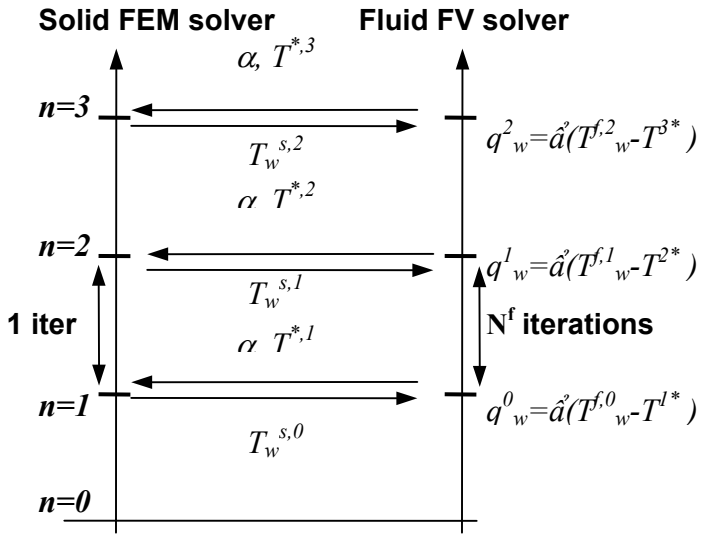


Figure 2 – Montenay's coupling procedure

Two meshes were created to solve this coupled problem. An unstructured first order finite element mesh represented the solid domain. The fluid mesh is split into 4 sub-domains for parallel computing. The two media are coupled in the trailing edge cavity of the turbine blade. Both solid and fluid domains are presented in Figure 3 and Figure 4 respectively.

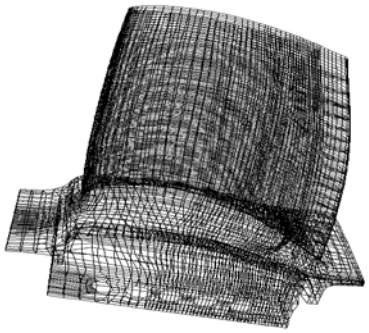


Figure 3 – Solid domain : unstructured finite element mesh.

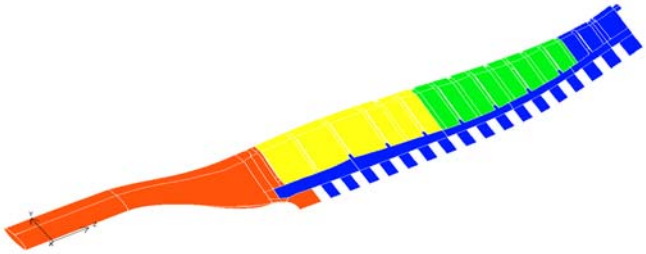


Figure 4 – Fluid domain : 4-sub-domain structured mesh

Constant $(\hat{a},T^*)$ boundary conditions were imposed on all the uncoupled surfaces of the solid

domain. These conditions were chosen to be close to full power conditions.

The following constants parameters were given to the fluid solver :

- Rotating speed,
- Inlet and outlet prescribed pressure,
- Turbulence rate and turbulence characteristic length scale.

A total of 5 instances dispatched over 5 processors were created to solve the coupled problems, using the parallel computing capabilities of the fluid solver MSD together with the coupling library MpCCI, version 1.3. The entire calculation was run on 5 five SGI/Irix 6.5 processors.

The coupled calculations required the fluid fields to be initialized. 4000 initial iterations were performed by the fluid solver with a uniform prescribed temperature condition at the coupled interface.

All along the coupled computation, the heat transfer coefficient â was kept high and the number of fluid iterations between two exchanges was equal to 100.

We followed the convergence history by plotting the average solid temperature increment on the coupled surface, as a function of the number of exchanges.

The temperature increment decreases below the acceptable level of 0.1 *K* in about 40 coupling steps, namely 4000 fluid iterations. A coupled fluid-solid temperature field is therefore obtained in twice the time of a non-coupled fluid calculation. Experience shows that 8 to 9 iterations would be necessary to reach the same result with a classical iterative procedure, which would require full convergence by the fluid solver to allow fluid/solid exchanges.

Figure 5 – Average solid temperature increment, as a function of the coupling step number. Logarithmic scale.

From Figure 5 we can also observe that temperature convergence conforms to the following logarithmic law :

$$(\Delta T)_{\max} = a(0.9)^n \qquad (10)$$

where *n* is the coupling step number.

Heat flux convergence at the coupled interface is presented in Figure 6. Violent oscillations can be observed shortly after initialization, as both fluid and solid temperature fields are far from their converged values. These oscillations decrease as the calculation gets closer to convergence. These phenomena justify the high value prescribed for the heat transfer coefficient. As shown by Montenay, temperature oscillations need to be constrained at the beginning of the coupled calculation to allow convergence. However, as the coupled problem converges, the temperature gap between the solid and the fluid wall temperature can be less constrained.

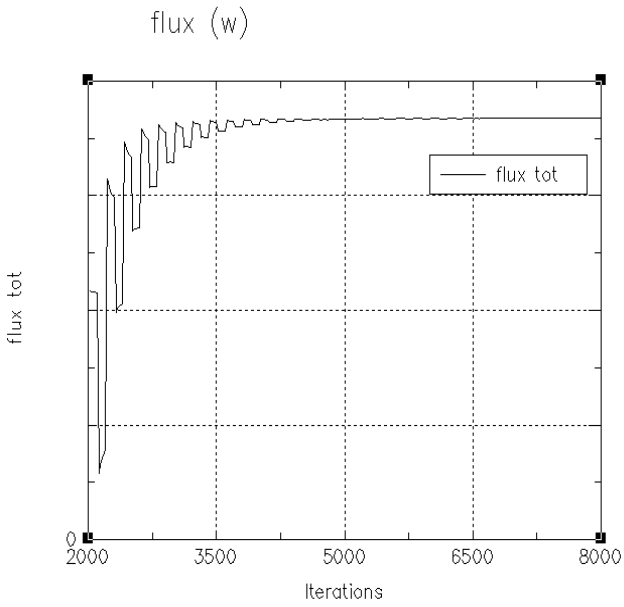


Figure 6 – Fluid heat flux convergence at coupled interface.

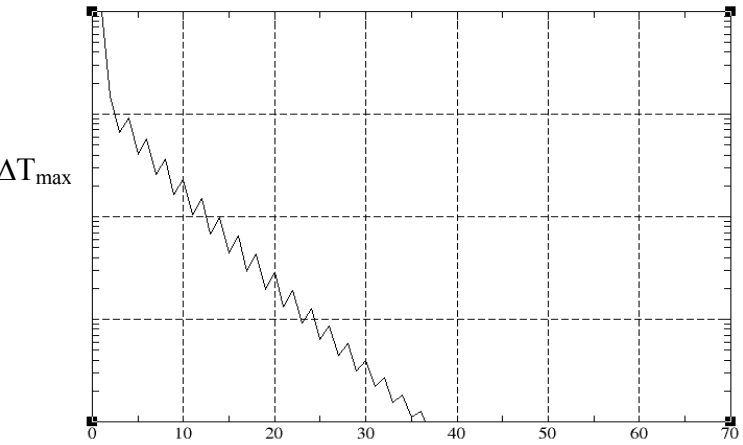


Figure 7 illustrates the continuity of the temperature field at the coupled interface, when convergence is reached. Both solid and fluid temperature fields have been interpolated at the same vein height, from the solid and fluid meshes respectively. Both temperature distribution obtained are plotted in curvilinear coordinates, from suction to pressure side. Only slight discrepancies are observed, at geometric accidents such as sharp angles. They are mainly due to the fact that the fluid and solid meshes are not coincident. Good agreement of both curves is achieved. This demonstrates the ability of Montenay's method to lead both fluid and solid solver to wall temperature convergence.

Results obtained were compared to experimental data. Very good agreement was obtained. This coupled approach offers new perspectives for heat transfer analysis on turbine components.

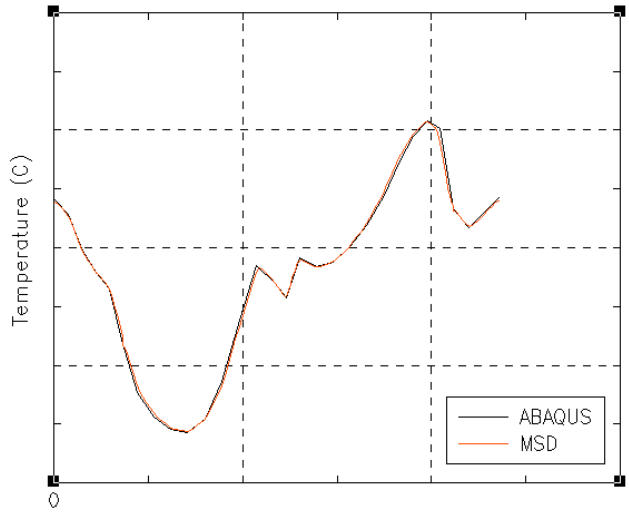


Figure 7 – Temperature continuity at convergence, at the coupled interface.

Conclusion

An iterative steady state method for conjugate heat transfer analysis has been developed at Snecma Moteurs. Its ability to handle general flow situation has been demonstrated through a coupled calculation in a 3D turbine blade cavity.
Convergence has been obtained in twice the time required by an uncoupled Navier-Stokes calculation, which makes the method quite cheap in terms of CPU time.
Future efforts will be dedicated to the study of the numerical heat transfer coefficient and the number of fluid solver iterations, in order to increase convergence speed while preserving stability.

Bibliography :

[1] Montenay, A., « Analyse Thermique de l'Aérothermique d'Aubages et de Cavités de Turboréacteur », Thèse Université paris VI, June 2000
[2] Dutoya D., and M.P. Errera, « Le code de MATHILDA. Modèles physiques, réseau de calcul et méthode numérique », Tech. Report 42/3473 EN, ONERA, 1991
[3] « ABAQUS Theory Manual. Version 5.8 », HKS Inc., 1998