Transactions on Engineering Sciences vol 5, © 1994 WIT Press, www.witpress.com, ISSN 1743-3533

A numerical approach for thermally coupled fluid and solid problems in complex geometries

C. Péniguel[a] & I. Rupp[b]

[a]*EDF/LNH, 6 quai Watier BP 49 F-78401 Chatou, France*

[b]*SIMULOG, 1 rue James Joule F-78286 Guyancourt Cedex, France*

ABSTRACT

Thermal coupling between fluid and solid domains is widely present in industrial applications. The approach implemented relies upon the existing CFD code N3S (see Chabard [1]), which handles the averaged Navier-Stokes system with a finite element technique, and a module named Syrthes which takes care of the heat equation inside the solid using finite elements technique.

An explicit scheme, has proven to be very stable to exchange information (temperature or flux) between solid and fluid domains within each time step.

The numerical and geometrical de-coupling of solid and fluid domains has numerous advantages and provides good flexibility when handling complex cases.

INTRODUCTION

In many industrial applications, a thermal coupling exists between a fluid and the solid body by which it is surrounded. Among topics of interest, studied at EDF, thermal shocks arising in piping systems of nuclear plants can be pointed out. These shocks originate from a quick variation of the flow temperature. This may lead to mechanical damages (like cracks). Of course, other fields, like heat exchangers, electrical devices, etc... have to address the same kind of problem.

Experimental approaches have been used extensively in the past, but they may become very costly (when dealing with a very hot fluid under high pressure for example). Moreover, they often lack the flexibility needed when a parametric study is desired. On the other hand, with powerful computer facilities now available at affordable cost, numerical approaches become more and more promising to accurately predict thermal phenomena and their effects.

Tackling a full size problem requires to handle thermal phenomena in the fluid part, in the solid part and the strong interaction between the two regions. At EDF, sophisticated numerical tools have been developped in the past years to handle fluid problems. The software N3S relies on finite element techniques to solve the averaged Navier-Stokes equations.

Unfortunately, wall effects (from the thermal point of view) could only be taken into account in N3S by imposing a wall temperature or a flux. Therefore, thermal fields within solids were not available and thermal inertia caused by the solid wall could not be accounted for.

A general purpose conduction module, Syrthes, is now handling the heat equation inside the solid and the thermal coupling between fluid and solid regions.

Transactions on Engineering Sciences vol 5, © 1994 WIT Press, www.witpress.com, ISSN 1743-3533

THE PROCEDURE IMPLEMENTED

The main characteristics of the two numerical tools used to solve the temperature equations in the solid and fluid region will not be discussed in this paper. Details can be found in Chabard, Pot [1] [2], for the fluid, and in Peniguel and Rupp [3] for the solid. Problems left to be solved are related to the exchange taking place between the two domains. Several numerical schemes can be proposed to couple the two regions (implicit, half implicit, explicit). The simplest of them is explicit. Once the boundary conditions known, solving the heat equation can provide the temperature in each domain. Temperatures or fluxes are therefore updated inside each domain and of course at the interface. Then, these updated values can be used as boundary conditions for the other domain and so on. Even if one can show that very degenerated cases (like 1D pure conduction) may encounter stability problems, for general cases, involving three-dimensional diffusion and non linear convective terms (in the flow), no problems arose in the numerous cases treated so far by the authors. Moreover the choice of an explicit procedure provides possibilities to extend this development to parallelism when dealing with very complex cases.

Description of the algorithm

Let T_s be the temperature of an internal solid node, T_w the temperature at a node which belongs to the interface, and T_f the temperature of a fluid point (located generally in the log layer).

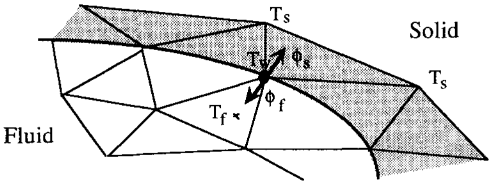


Figure 1 : Data transfer

At time t^n, T_s^n, T_w^n, T_f^n are known

- From the local fluid conditions and the wall temperature, the CFD code N3S provide after calculation :
  h^{n+1} : the local heat exhange coefficient at time t^{n+1}
  T_f^{n+1} : the local inside fluid temperature at time t^{n+1}
- using these data, the flux to be applied to the solid is :
  $\phi_s^{n+1} = -h^{n+1}\,(T_w^n - T_f^{n+1})$
- Using this flux (or the exchange conditions (h^{n+1}, T_f^{n+1})) one can solve the heat conduction equation inside the solid. This gives an updated T_s over all the solid region. These new values named T_s^{n+1} are also updated on the boundary. Therefore T_w^{n+1} is also known.

Thus T_s^{n+1}, T_w^{n+1}, T_f^{n+1} are known at time t^{n+1}

The calculation of the turbulent heat exchange coefficient, noted h above, takes different forms according the modelling retained, or the position, with respect to the boundary layer thickness, of the point where T_f is estimated. A detailed description of the wall law used inside N3S is given in Delenne and Pot [4].

Transactions on Engineering Sciences vol 5, © 1994 WIT Press, www.witpress.com, ISSN 1743-3533

Geometrical treatment of the interface

As stated before, solid and fluid problems are discretized on independent grids. At the interface, these grids may be coincident or not. In the later case, the two grids only approach the same geometrical boundary.

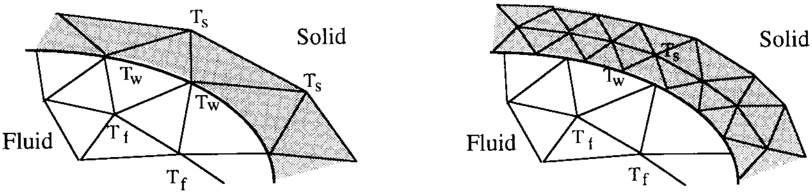


Figure 2 : Coincident and non coincident options

The second option is much more general and has numerous advantages. For example, the solid thermal field can be used afterwards for mechanical stresses calculations, since the solid has its own independent grid. Moreover grid refinements can be located at different places than those required for the fluid. This allows to capture high gradients in the solid and in the fluid at a much lower cost. Finally one can imagine that fluid and solid meshes be generated by independent engineers or teams, once the common geometry defined.

On the other hand, an interpolation procedure has to be performed each time that updated boundary conditions are required when solving fluid and solid thermal equations.

A very robust and efficient method has been implemented to handle this problem. Boundaries to be coupled are identified and boundary meshes are built, (a surface formed by triangle embedded in R^3 for three dimensional problems, or a line for two dimensional problems).

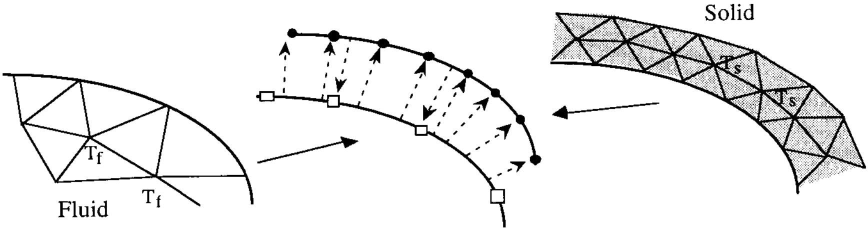


Figure 3 : Boundary meshes extracted

Then for all nodes of each boundaries, a procedure determines which element and nodes of the other boundary mesh should be retained for the interpolation. The following sketch explains briefly the procedure used, on a two dimensional example, but the same technique applies in three dimensional space.

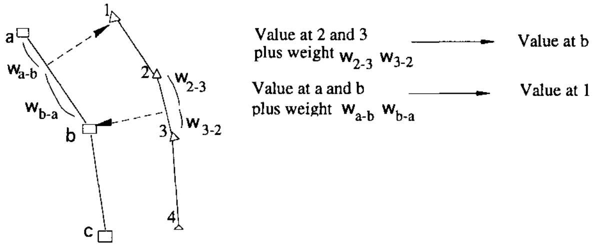


Figure 4 : Data transfer

Even if the procedure implemented has been very optimised, this geometrical search may become costly if the number of nodes to be treated is high. However in practice, it is done only once, and information like the corresponding element and the weight to apply for interpolation is stored. Then the data transfer from one grid to the other is performed at very low cost.

APPLICATIONS

The authors have applied this development to numerous cases. Three of them are presented here. The first one (axisymmetrical case) is concerned with the thermal shock experienced by a component similar to those found in French power plant. A second one deals with a three dimensional case. Finally the last one shows the flexibility of the approach retained by the authors on a complex case.

The CLAPET experiment - axisymmetrical case

EDF is studying the consequences of thermal shock on components, to determine potential long term damage. Typically in the following component, water is flowing at constant flow rate (80 $m^3/hour$ which leads to about 6.5m/s), at a temperature of $60°C$ and a pressure of 85 bars. Suddenly the temperature of the incoming fluid rises to $280°C$. A very sudden heating of the wall occurs, and propagates through the metal. This heating can be quite complex due to the flow pattern.

The following figure shows the grids used for the fluid and the solid domains. The fillet regions, where cracks are likely to appear, have been discretized carefully.

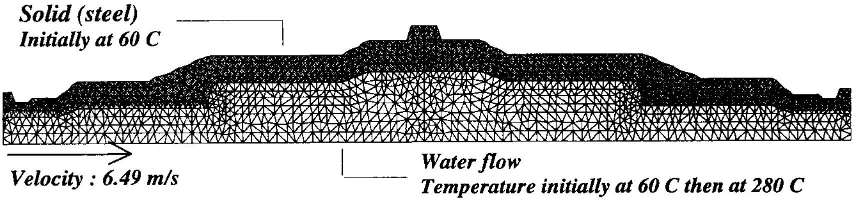


Figure 5 : Grids used for the fluid and the solid regions.

The next figure shows the temperature field inside the two domains at $t = 9.75s$. The non regular thermal field found in the solid originates firstly from the delay occasionned by the recirculating flow, secondly because the local heat exchange coefficient varies widely from one place to another. Highest values (where local velocities are high) will correspond to high heat fluxes leading to a quick heating. On the contrary low velocities will give way to slow heating due to small heat exchange coefficients.

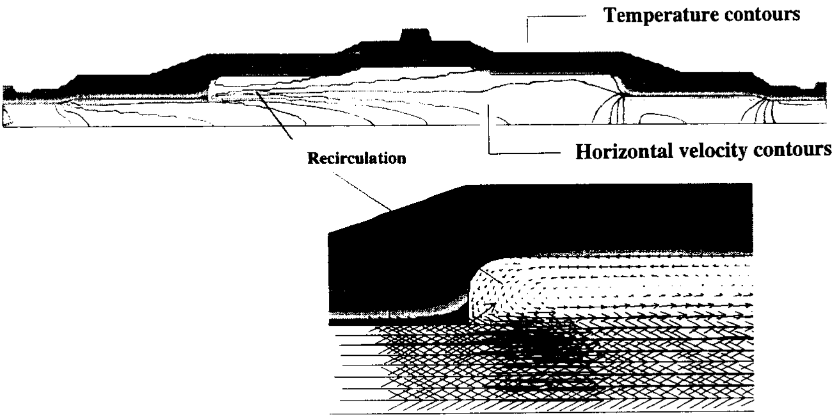


Figure 6 : Thermal contours obtained with a coupled calculation

To emphasize this point figure 7 gives the solid thermal contours obtained when performing a purely conductive calculation, with a constant (guessed!) heat exchange condition. The very regular thermal contours obtained are much less physical, although this constitutes a very classical approach.

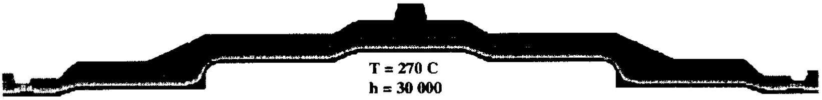


Figure 7 : Thermal contours obtained with a purely conductive approach

Thermal shock in a three dimensional geometry

The same type of study has been performed for three dimensional geometries. Here it is clear that the solid part has been carrefully discretized near the fillet region where cracks may appear, while the grid used for the fluid is much coarser. Of course both grids approach the same geometry at the interface.

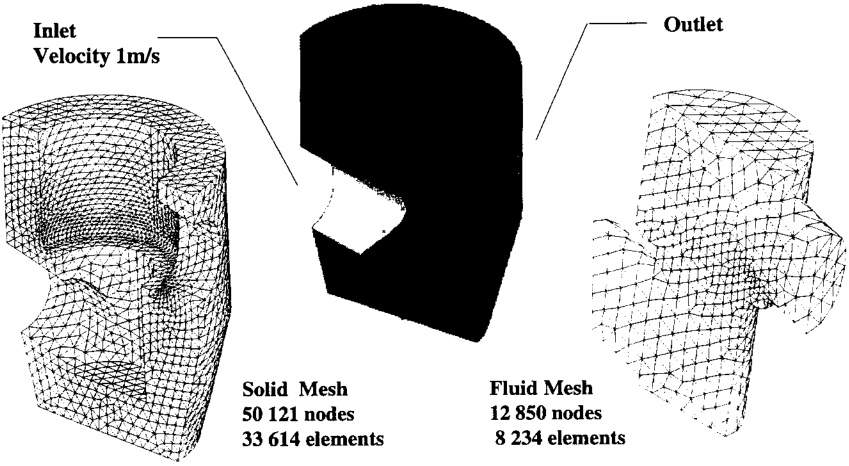
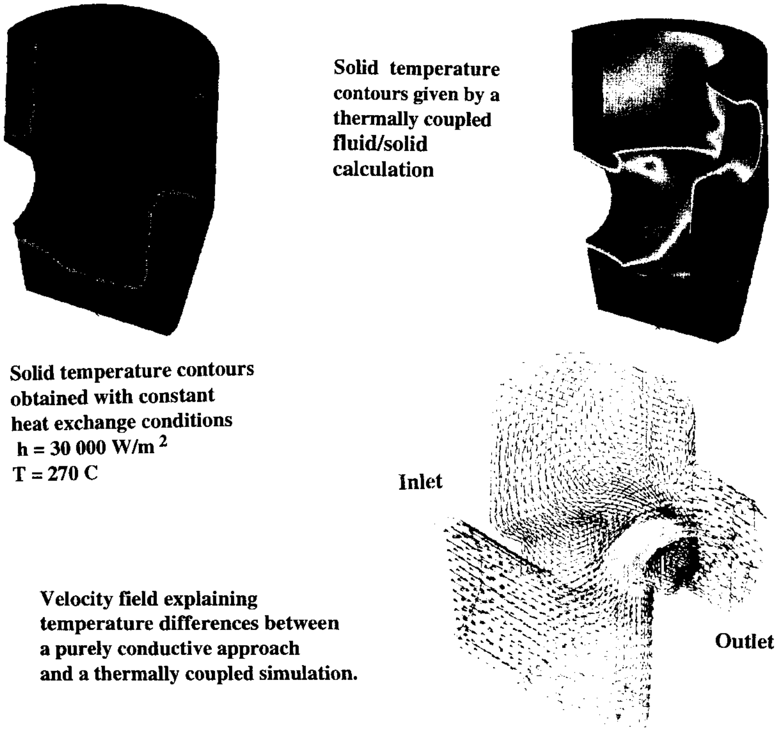


Figure 8 : Geometry and grids used in solid and fluid regions

Transactions on Engineering Sciences vol 5, © 1994 WIT Press, www.witpress.com, ISSN 1743-3533

Once again the thermal field obtained when performing purely conductive calculations with a guessed heat exchange coefficient differs widely from a fully coupled calculation. The main explanation is related to the very complex velocity pattern existing in the mock-up.


Solid temperature contours given by a thermally coupled fluid/solid calculation

Solid temperature contours obtained with constant heat exchange conditions
$h = 30\,000$ W/m ^{2}
$T = 270$ C

Velocity field explaining temperature differences between a purely conductive approach and a thermally coupled simulation.

Figure 9 : Temperature and velocity fields

A complex case - Towards the parallelism

In many industrial processes, there may be more than one fluid taking part. One may think of heat exchanger devices, but this may also arise in "simpler" problems. For example, lets take the case of a fluid flowing inside a pipe, if the pipe is not insulated, natural convection may take place on the external side. Then the solid is related not only to one fluid calculation but with two of them.

The option retained by the authors to tackle this kind of problems is to run several N3S computations simultaneously (one N3S for each independent fluid domain) and one Syrthes module to solve the heat conduction problem and to synchronize all data exchanges (wall temperature and fluxes) accross the interface between fluid domains and solid domains within each time step.

The data exchange is performed this time between several computer processes, running on the same computer or on a cluster of computers, with the help of PVM (Parallel Virtual Machine). This software package (see reference [5]) allows a network of computers to appear as a single computation ressource and allows computer tasks to communicate and synchronize with each other.

One case illustrating this capability is presented below.

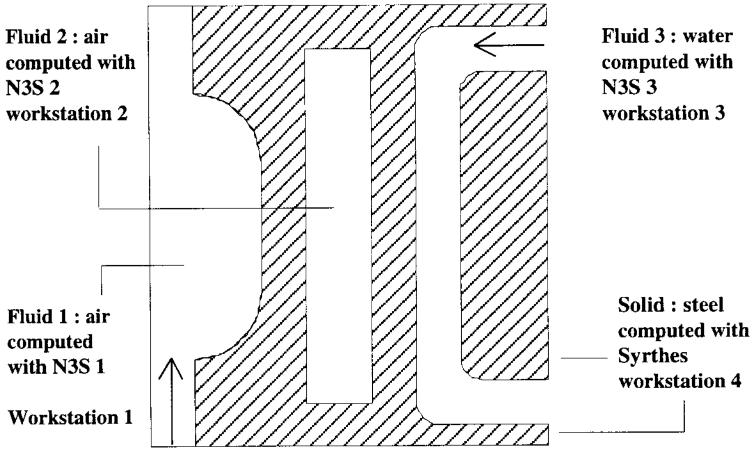


Figure 10 : A complex case with several fluid domains

Three flows are taking place simultaneously. The left one is air flowing at a speed of $2m/s$, initially at a temperature of $20°C$, the temperature increases suddenly at $200°C$. Another one, located on the right, is water flowing at a speed of $1m/s$ with an inlet temperature of $20°C$. Finally, the middle domain is composed of air, initially at rest, at a temperature of $20°C$.

Initially, all domains are at a temperature of $20°C$. When the left flow experiences a thermal shock, it heats the right wall with which it is in contact. Heat propagates through the wall and heats the air inside the closed cavity. Natural convection starts due to gravity effects. On the contrary, the wall on the right side of the rectangular cavity is maintained at a low temperature, indeed the cold water flowing the other side of the wall cools it down.

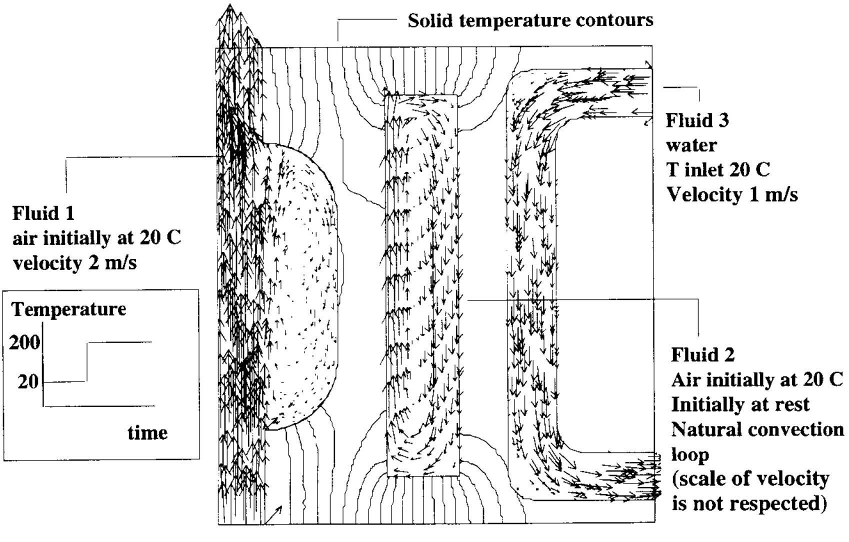


Figure 11 : Velocity fields in 3 fluid domains, and solid temperature

Transactions on Engineering Sciences vol 5, © 1994 WIT Press, www.witpress.com, ISSN 1743-3533

This calculation has been performed on four workstations, fluid and solid problems being solved on independent computers. PVM allows the four tasks (three N3S codes and one Syrthes module) to communicate data (temperature or fluxes at the interface) and synchronize each other.

It is clear that very complex processes can be simulated using such an approach, without increasing dramatically the difficulty of a calculation. Indeed work can be dispatched between computer ressources available (mainframe or cluster of workstations) but also between different teams. Each team being responsible only for one part of the global problem. Moreover this makes possible the use of different modelling (turbulence for example) in each separate fluid domain. One can also imagine to simulate a compressible flow outside of a body and an incompressible flow inside the same body (e.g. turbine blade cooling), the balance of heat across (and inside) the wall being updated within each time step.

CONCLUSION

A numerical method to handle thermally coupled fluid and solid domains has been presented.

One advantage of treating the solid with an independent unstructured mesh is that the solid geometry is well handled at low cost. Another advantage is that thermal results may be used directly by mechanical codes to calculate thermally induced stresses in structures or components.

Numerous cases have been studied, and so far the explicit numerical scheme chosen for the data transfer seems to behave very well. Relying on the sophisticated CFD code N3S for fluid problems and the independent module Syrthes for solid thermal problems allows great flexibility when handling complex cases. This numerical decoupling allows also to extend the development towards parallelism which seems quite promising as has been demonstrated in the last application shown in this paper.

REFERENCES

1 CHABARD J.P., METIVET B., POT G., THOMAS B.
An efficient finite element method for the computation of 3D turbulent incompressible flows, *Finite Elements in Fluids, Vol 8, 1992*

2 POT G., BONNIN O., MOULIN V., THOMAS B.
Improvement of finite element algorithms implemented in CFD code N3S for industrial turbulent applications, Proceedings of *Numerical Methods for Laminar and Turbulent Flow* Swansea, U.K., July 1993

3 PENIGUEL C., RUPP I.
A Numerical Method for Thermally Coupled Fluid and Solid Problems Proceedings of *Numerical Methods in Thermal Problems*
Swansea, U.K., July 1993

4 DELENNE B., POT G.
Modelling of the Turbulent Thermal Boundary Layer Implemented In the CFD code N3S and Application to flows in heated rooms. Proceedings of *Numerical Methods in Thermal Problems* Swansea, U.K., July 1993

5 GEIST A., BEGUELIN A., DONGARRA J., JIANG W., MANCHEK R.
PVM 3 USER'S GUIDE AND REFERENCE MANUAL
Oak Ridge National Laboratory, Oak Ridge, Tennesse 37831