Methodology of numerical coupling for transient conjugate heat transfer
Emmanuel Radenac, Jérémie Gressier, Pierre Millan

To cite this version:

HAL Id: hal-01002811
https://hal.archives-ouvertes.fr/hal-01002811
Submitted on 6 Jun 2014

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L’archive ouverte pluridisciplinaire HAL, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d’enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.
Open Archive Toulouse Archive Ouverte (OATAO)
OATAO is an open access repository that collects the work of Toulouse researchers and makes it freely available over the web where possible.

This is an author-deposited version published in:  http://oatao.univ-toulouse.fr/
Eprints ID: 11744

To link to this article: DOI: 10.1016/j.compfluid.2014.05.006
URL:  http://dx.doi.org/10.1016/j.compfluid.2014.05.006

To cite this version: Radenac, Emmanuel and Gressier, Jérémie and Millan, Pierre Methodology of numerical coupling for transient conjugate heat transfer. (2014) Computers & Fluids, vol. 100. pp. 95-107. ISSN 0045-7930

Any correspondence concerning this service should be sent to the repository administrator:  staff-oatao@inp-toulouse.fr
Methodology of numerical coupling for transient conjugate heat transfer

E. Radenac\textsuperscript{a,}* , J. Gressier\textsuperscript{b} , P. Millan\textsuperscript{a}

\textsuperscript{a} Onera, The French Aerospace Lab, F-31055 Toulouse, France
\textsuperscript{b} Institut Supérieur de l’Aéronautique et de l’Espace, 10 av. Edouard Belin, BP 54032, 31055 Toulouse, France

\begin{abstract}
This paper deals with the construction of a conservative method for coupling a fluid mechanics solver and a heat diffusion code. This method has been designed for unsteady applications.

Fluid and solid computational domains are simultaneously integrated by dedicated solvers. A coupling procedure is periodically called to compute and update the boundary conditions at the solid/fluid interface. First, the issue of general constraints for coupling methods is addressed. The concept of interpolation scheme is introduced to define the way to compute the interface conditions. Then, the case of the Finite Volume Method is thoroughly studied. The properties of stability and accuracy have been optimized to define the best coupling boundary conditions: the most robust method consists in assigning a Dirichlet condition on the fluid side of the interface and a Robin condition on the solid side. The accuracy is very dependent on the interpolation scheme. Moreover, conservativity has been specifically addressed in our methodology. This numerical property is made possible by the use of both the Finite Volume Method and the corrective method proposed in the current paper. The corrective method allows the cancellation of the possible difference between heat fluxes on the two sides of the interface.

This method significantly improves accuracy in transient phases. The corrective process has also been designed to be as robust as possible. The verification of our coupling method is extensively discussed in this article: the numerical results are compared with the analytical solution of an infinite thick plate in a suddenly accelerated flow (and with the results of other coupling approaches).
\end{abstract}

1. Introduction

Many industrial applications are subject to strong thermal interactions between fluids and solids. Severe configurations exhibit intense aerothermal phenomena, involving complex flows and geometries, unusual thermal conditions at their boundaries and multiple fluids and solids.

Conjugate heat transfer is thus a key industrial issue which can be investigated in several ways. Analytical approaches produce interesting information to identify the leading parameters of a problem and to verify codes. But their applicability is restricted to very simple configurations (such as forced convection over an infinite or semi-infinite thick plate [1,2] or in a channel with thick walls [3]). Alternatively, experiments are essential at least for validation (for instance, the study reported in paper [4]). But they are expensive. The development of new technologies thus cannot rely only on experiments.

That is why numerical simulations are absolutely necessary. For a long time, they have consisted of decoupled studies. For instance, numerical flow simulations were performed with given thermal boundary conditions (often adiabatic or isothermal). This approximate method does not account for the fact that convection affects the fields of temperature and heat fluxes in the solid. There is an essential condition for computing both fluid and solid parts accurately. At their interface, the boundary conditions must be properly calculated.

Coupled methods are therefore necessary for conjugate heat transfer. There is a particular need for unsteady studies. For example, in the aeronautical industry, several transient phases occur in a jet engine during a flight. The safety margins are usually set by steady-state approaches. They could be reduced by a better characterization of transient phenomena.

Sondak et al. [5] have noticed that many studies mainly produce steady-state results for aerothermal fields in turbines [6–10]. Several solver couplings are also dedicated to very specific configurations, with specific geometries or solvers (studies in the fields of space [11], combustion [12,13]). Besides, more and more unsteady applications can be found in the literature [5,12–15,21].

Sometimes, the whole computation uses a single fluid solver which treats solid parts as rigid bodies. But it is usual to couple distinct fluid and solid solvers. The fluid and solid domains are...
Thus, heat transfer involves convection (due to viscous effects (included in divergence of energy source).  

In terms of accuracy, the coupling methodology affects unsteady calculations more than steady-state simulations. Most coupling methods do not ensure the proper instantaneous conditions at the solid/fluid interface. This issue will be extensively discussed in the current article. The innovative concept of interpolation scheme will be introduced. A novel interpolation scheme will be proposed and compared to more usual approaches of the literature.

Moreover, our coupling methodology is intended to be conservative. This concept is poorly discussed in the literature of coupling strategies. Conservativity ensures that no energy is lost during the computation. This requires the use of the original method described in this paper. It will be demonstrated that a conservative method improves accuracy even though the codes exchange data at small frequencies, which leads to gains in computational cost. A verification case will show the efficiency of our method.

2. Physical problem

This section presents the main equations describing heat transfer in a solid, in a fluid and at their interface. In the whole paper, the subscripts I, S and F will denote the properties at the interface, in the solid and in the fluid, respectively.

2.1. Conservation laws and heat transfer in solids and fluids

Solids and fluids are governed by conservation laws in the general form

$$\frac{\partial U}{\partial t} + \nabla (\vec{F}(U)) = \dot{S}.$$  \hspace{1cm} (1)

where $U$ is a vector of conserved quantities, $\vec{F}$ is a flux tensor and $\dot{S}$ is a vector of source terms.

2.1.1. Heat transfer in solids

For solids, $U = e_s$, $\vec{F} = \Phi_s$, and $S = \rho_s C_s \dot{T}_s$, where the internal energy $e_s$ depends on the temperature $T_s$ : $e_s = \rho_s C_s T_s$. The heat flux density is given by Fourier’s law $\Phi_s = -\dot{\lambda}_s \nabla T_s$. $\Phi_s$ is a heat source, $\rho_s$, $C_s$ and $\dot{\lambda}_s$ are the mass density, the specific heat capacity and the thermal conductivity of the material, respectively.

2.1.2. Heat transfer in fluids

For fluids, Eq. (1) represents the Navier–Stokes equations [16]. Heat transfer is more specifically described by the law of conservation of energy

$$\frac{\partial \rho \phi}{\partial t} + \nabla (\rho \phi \vec{V} - \frac{\dot{\lambda}_f}{\gamma - 1} \vec{V} \cdot \vec{V}) - \rho \phi \dot{\lambda}_f \frac{\dot{\lambda}_f}{\gamma - 1} \vec{V} = 0,$$ \hspace{1cm} (2)

where the total energy of the fluid is given by $e_f = \frac{r}{\gamma - 1} T_f + \frac{\dot{\lambda}_f}{\gamma - 1} \vec{V}$. Thus, heat transfer involves convection (divergence of flux), a heat source due to viscous effects (included in div$(-\frac{\dot{\lambda}_f}{\gamma - 1} \vec{V})$), heat diffusion (div$(\Phi_f)$ = div$(\dot{\lambda}_f \nabla T_f)$) and other heat sources ($\rho \phi \dot{\lambda}_f$, $\vec{V}$, $\phi_f$ and $T_f$) are the flow velocity, the fluid density and the temperature, respectively. $\dot{\lambda}_f$ is the thermal conductivity, $r$ is the specific gas constant and $\gamma$ is the heat capacity ratio. $\nabla T_f$ represents the body force (for instance gravity), $\nabla T_f$ is the stress tensor.

2.2. Solid/fluid interface

Under the assumption of perfect thermal contact, normal heat flux and temperature are continuous across the interface between a solid and a fluid. The heat flux condition $\Phi_I = \Phi_F$ and the temperature condition $T_I = T_F$ at the interface should thus be such that

$$\Phi_I = \Phi_{IS} = -\Phi_{IF},$$
$$T_I = T_{IS} = T_{IF},$$  \hspace{1cm} (3)

where the subscripts $IS$ and $IF$ have been introduced to indicate conditions right at the interface on the solid and fluid sides, respectively (see Fig. 1 in the case of the Finite Volume Method). Here, the heat flux vector $\Phi_{IS}$ is oriented from the solid to the fluid (whereas conversely, $\Phi_{IF}$ leaves the fluid).

During the independent integrations of the solid and fluid domains, the solvers do not enforce both temperature and heat flux at their interface. Therefore, Eq. (3) cannot always be satisfied. Coupling a fluid solver and a solid solver consists of periodically updating their interface boundary condition so that the heat flux $\Phi_I$, and the temperature $T_I$, fulfill condition (3) at each face of the interface. The numerical evaluations of $\Phi_I$ and $T_I$ actually depend on temperature and heat flux in the solid and fluid domains ($T_S$, $T_F$, $\Phi_S$ and $\Phi_F$, respectively) and on other parameters such as the solid and fluid heat conductances ($h_S$ and $h_F$, which will be more precisely defined in Section 4.1.1):

$$ \begin{cases} \Phi_I = \mathcal{F}_S (\Phi_S, \Phi_F, T_S, T_F), \\ T_I = \mathcal{F}_T (\Phi_S, \Phi_F, T_S, T_F). \end{cases} \hspace{1cm} (4) $$

$\mathcal{F} = (\mathcal{F}_S, \mathcal{F}_T)$ will be called an interpolation scheme. At this stage, it is not specified where exactly $T_S$, $T_F$, $\Phi_S$ and $\Phi_F$ are assessed. For temperature specifically, $T_S$ and $T_F$ may be assessed either at the interface or at the neighboring node (or at the centroid of the neighboring cell).

3. General coupling strategy

In this section, the main properties of our coupling strategy will be described independently of the discretization technique used in the coupled solvers. Section 4 will deal with the application of this strategy to the Finite Volume Method.

3.1. Coupling process

Several strategies can be employed to couple fluid and solid solvers. Duchaine [15] has proposed to distinguish between sequential (Fig. 2) and parallel coupling strategies (Fig. 3).

Fig. 1. Adjacent cells in the vicinity of a coupling interface.
In the sequential approach, the solid and fluid solvers are run alternately, each of them providing the boundary condition of the other. In the example of Fig. 2, at coupling step \( N \), the fluid code provides a boundary condition to the solid solver. The solid domain is then integrated during \( N_i \) iterations. At the following synchronization, the fluid solver receives data from the solid solver, allowing to solve the fluid domain. The sequential coupling strategy is quite easy to implement, because the solvers are successively loaded into all the required processors of the parallel machine. But it is mainly expected to be accurate for steady-state computations (actually, it is often used with steady-state codes\([10]\)). Obtaining an accurate solution during the whole unsteady integration would require a local convergence (therefore an iterative process) at each time step.

In the parallel approach (Fig. 3), the two solvers are run together (in parallel) once the boundary conditions have been updated at their interface (at a coupling step). The fluid and solid domains are independently solved during a cycle of duration \( \Delta t \) (which is the coupling period, Fig. 3(a)). The time steps \( \delta t_F \) and \( \delta t_S \) are not necessarily equal. A cycle involves a coupling step and the subsequent integration of the fluid and solid domains (until the following exchange). \( \Delta t \) is thus a global time step linked to the coupled calculation, whereas \( \delta t_F \) and \( \delta t_S \) are related to the independent solver computations. \( \Delta t \) is chosen after the physics of the problem in order to reach a satisfactory trade-off between accuracy and time resolution. This point will be addressed in Section 5. The stability of coupling processes is a particular problem discussed in reference \([17]\). For instance, the condition that \( \Delta t \) must fulfill is different from the usual CFL or Fourier conditions.

In the general approach investigated in the current paper, the coupling step has two components. First, the interface conditions are computed from data provided by both solvers at the end of each cycle. Second, boundary conditions are assigned to the solvers. In Duchaine’s parallel method, the first step is skipped and the solvers directly exchange their temperature and flux conditions at the boundary.

As discussed in Duchaine’s article \([15]\), the parallel coupling strategy is recommended for unsteady problems. It is all the more efficient in terms of computational cost as \( \Delta t \) is large (which reduces the frequency of exchanges between the solvers). The frequency of coupling steps influences the stability, the accuracy and the restitution time of the computation.

The parallel coupling strategy is thus selected for the current method. But three aspects of the coupling must still be specified:

1. the exact type of boundary conditions used by the two solvers at the interface. This defines a couple of boundary conditions, which will be specified in Section 3.2.2;
2. the way the interface conditions are computed, or, in other words, the selection of the interpolation scheme. This point will be addressed in Sections 3.2.1 and 3.2.3 (as well as in Section 4.1, in the case of the Finite Volume Method);
3. the method that ensures conservativity (Fig. 3(b)), which will be explained in Section 3.3 (and in Section 4.2 in the specific case of the Finite Volume Method).

The coupling strategy will have to be accurate, stable and conservative while using as large \( \Delta t \) as possible. The stress will be put on the constraints that will ensure these properties.

3.2. Constraints and purpose

3.2.1. Consistency

Consistency is an important property of numerical methods for conservation laws such as Eq. (1), \( \nabla \cdot \hat{F}(U) \) is computed by using a numerical scheme. For instance, in a 1D form for illustration, at point \( i \) \([18]\),

\[
\frac{\partial \hat{F}(U)}{\partial x} = \hat{F}_{i+1/2}(U_{i+1}, U_{i+1-1}, k \in \{1, \ldots, N\}) - \hat{F}_{i-1/2}(U_{i-1}, U_{i+1}, k \in \{1, \ldots, N\})
\]

where \( \Delta x \) is the grid cell size, \( N \in \mathbb{N}^* \) and \( \hat{F} \) is called the numerical flux (whereas \( F \) is the physical flux). \( \hat{F} \) is a function of the values of \( U \) at the neighboring points. The consistency condition states that \([18]\)
\[ \hat{F}_{i+1/2}(U_{i,k}, U_{i,k+1}), k \in \{1, \ldots, N\} = \hat{F}(U(x_{i+1/2})), \quad \text{when } \forall k, \quad U_{i,k} = U_{i+1,k} = U. \] (6)

This condition ensures the consistency of Eq. (5) with the original Eq. (1).

Likewise, if \( \Phi_0 \) and \( T_0 \) are arbitrary values of heat flux and temperature, a consistency condition can be designed for the interpolation scheme as:

\[
\begin{align*}
\{ T_0(\Phi_0, \Phi_f, T_f, T_s) \} &= \{ \Phi_0 \} \quad \text{when } \Phi_s = -\Phi_f = \Phi_0 \text{ and } T_k = T_f = T_0, \\
\{ T_0(\Phi_0, \Phi_f, T_f, T_s) \} &= T_0. 
\end{align*}
\] (7)

It ensures that an infinitely small discretization produces correct results. It also ensures that the state of equilibrium is stable once it has been reached, because the equilibrium temperature and heat flux remain unaffected at the interface. The interpolation schemes which satisfy condition (7) must be recommended.

### 3.2.2. Coupling boundary conditions

Once the conditions \( \Phi_0 \) and \( T_0 \) have been computed using Eq. (4), the boundary condition can be applied at the interface as a Neumann condition (\( \Phi_0 \) is enforced), as a Dirichlet condition (\( T_0 \) is enforced) or even as a Robin condition (with a heat transfer coefficient \( h_f \), a reference temperature \( T_{ref} \) and a reference flux \( \Phi_{ref} \)). \( h_f, T_{ref} \) and \( \Phi_{ref} \) must fulfill the condition

\[ \Phi_0 = h_f(T_0 - T_{ref}) + \Phi_{ref}. \] (8)

More specifically, a couple of boundary conditions must be enforced (one condition per domain). Some authors [5,19] have selected the couple of boundary conditions at the solid/liquid interface as follows: the heat flux of the fluid is assigned to the solid and conversely the temperature of the solid is assigned to the fluid. Hence, the boundary conditions are a Neumann condition for the solid and a Dirichlet condition for the fluid. This option proves to be the most robust one in Giles’ stability analysis [20]. To further improve the stability of the coupling process, a Robin condition is often preferred to the Neumann condition [7,9,15,21].

In the current paper, the notation for the two coupling boundary conditions will refer to the solid/liquid conditions, in that order: for instance, Robin/Dirichlet for a Robin condition assigned to the solid and a Dirichlet condition assigned to the fluid. It must be noted that the couple of boundary conditions is mainly selected on the basis of stability criteria [17].

### 3.2.3. Characteristics of unsteady coupling, influence of interpolation schemes

Steady coupling methods must at least ensure convergence to equilibrium of temperature and heat flux. For that purpose, the interpolation scheme must fulfill condition (7).

Extension to unsteady methods is not straightforward. Unsteady methods are expected to be accurate at each time of the computation. But a poor interpolation scheme induces errors which may vanish only once steady-state has been achieved. These errors may not even decrease over time if the computed case is purely unsteady.

As was explained earlier, a lot of methods found in the literature directly exchange their temperature and heat flux at the interface (Neumann/Dirichlet [5,19] or Robin/Dirichlet [7,9,15,21] boundary conditions), hence using a very basic “interpolation scheme”. The following example will illustrate that such an inappropriate interpolation scheme induces a non-physical transient. Let us consider two coupled 1D domains whose initial temperatures \( T_1 \) and \( T_2 \) are uniform, in the solid and in the fluid, respectively (Fig. 4). At \( t = 0s \), the uniform temperature in the fluid domain implies

\[ (\nabla T)_{-n} = 0 \text{ and } \Phi_2 = 0. \]

Hence, using the Neumann/Dirichlet approach of the literature, the boundary condition is \( \Phi_1 = \Phi_2(t=0s) = \Phi_2 = 0 \) on the solid side of the interface.

On the fluid side, it is \( T_1 = T_2(t=0s) = T_1 \). \( \Phi_2 \) is thus largely underestimated (the real value is infinite). But it is also very different from the heat flux computed during the subsequent fluid integration, which is for instance with a first-order scheme: \( 2T_2 - T_1 \neq 0 \) (where \( d \) is the size of the first cell at the interface on the fluid side, as shown in Fig. 4). Thus, evaluating interface conditions in that way can introduce major errors which at least affect unsteady accuracy.

Actually, the interpolation scheme \( \mathcal{F} \) must be as consistent as possible with the spatial schemes used by the solid and fluid solvers. It is now assumed that \( \mathcal{F} \) is exactly inferred from the schemes of the solvers. If the solvers have equal time steps and are coupled at each time step (\( \Delta t = \Delta t_f = \Delta t_2 \)), there is no mismatch between the heat fluxes at the surfaces of the two domains. Consequently, the heat flux is conserved and perfect equilibrium is achieved during the whole integration. But if \( \Delta t_f \neq \Delta t_2 \), the two domains are independently solved between two consecutive coupling steps (Fig. 3(a)). Therefore, regardless of the selected couple of boundary conditions, temperature and heat flux usually cannot be continuous at the interface all along the integration.

In the coupling method described hereafter, \( \mathcal{F} \) is expected to be accurate by simultaneously and instantaneously balancing both temperature and heat flux at the interface. Derivation of \( \mathcal{F} \) will be extensively discussed in the framework of the Finite Volume Method (Section 4.1.2, Eq. (16)).

### 3.3. A method for conservative coupling

Reference [17] has demonstrated that a non-conservative approach could result in significant errors in the steady-state computation of cases where Neumann conditions are assigned at all external boundaries and where at least one of the coupling boundary conditions is not a Neumann condition (which is often the case, at least for ensuring stability). It will be shown that using a conservative coupling method can also improve the accuracy of unsteady simulations (Section 5).

Let us assume that one of the coupling boundary conditions is not a Neumann condition. During the independent integrations of the two domains, the heat fluxes may be different on the two sides of the solid/fluid interface. This will produce energy losses. Indeed, the heat flux is enforced at the interface (and therefore equal on both sides) during the whole cycle only if Neumann conditions are assigned to both solvers. A corrective method is constructed to prevent such energy losses. This method, depicted in Fig. 3(b), consists of three steps:

\begin{figure}[h]
\centering
\includegraphics[width=0.5\textwidth]{Fig_4.png}
\caption{Fluid and solid domains with uniform initial temperatures.}
\end{figure}
1. the computation of the transferred energies: the solvers compute the thermal energy transferred across each face of the interface during a cycle (solid: \(E_s\), and fluid: \(E_f\)):

\[
E_s = \int_{t}^{t+\Delta t} \int_{S} \dot{\Phi}_s(t) \cdot dS \, dt,
\]

\[
E_f = \int_{t}^{t+\Delta t} \int_{S} \dot{\Phi}_f(t) \cdot dS \, dt.
\]

\(\Delta t\) is the time step, \(S\) is the surface area, \(t\) is the time, \(\dot{\Phi}\) is the flux, and \(\Delta E = E_f - E_s\) is the energy transferred between the solid and the fluid. \(n\) is the surface normal.

2. the assignment of the conservative correction: at the coupling step, transferred energies \(E_s\) and \(E_f\) are compared face by face. The real amount of energy transferred across the interface (which is most probably neither \(E_s\) nor \(E_f\)) is not known and must be estimated. As shown in Fig. 5, the estimator \(E_i\) is interpolated as follows: \(E_i = kE_s + (1-k)E_f\). The weighting coefficient \(k\) takes values within the range \([0, 1]\). Corrective energies are inferred:

\[
\Delta E_s = E_i - E_s = (k-1)(E_s + E_f),
\]

\[
\Delta E_f = -\Delta E_s = E_i - (1-E_s) = k(E_s + E_f);
\]

3. the assignment of the corrective energies \(\Delta E_s\) and \(\Delta E_f\) can be assigned either as flux conditions at the interface or as increments of temperature like \(\Delta T_s = -\frac{\Delta E_i}{\rho_s c_s V_s}\) (for instance in the adjacent cells of volume \(V_s\) in the solid, in the framework of the Finite Volume Method). Besides, \(\Delta E_s\) and \(\Delta E_f\) can be applied before or after updating the coupling boundary conditions. They can even be distributed over a given number of iterations of the following cycle. These options are called pre-update, post-update or distributed corrections, respectively (Fig. 6).

Consequently, the corrective energy is spatially (through the use of \(k\)) and temporally distributed. There are several ways to make these distributions. All of them ensure conservativity and improved accuracy but they are discriminated by their impact on stability.

Step 1 requires that the solvers compute the thermal energy transferred across each face of the interface during a cycle. It may be an intrusive part of the the coupling method: integration of \(E\) (Eq. (9)) must be implemented into the solvers. Step 3 can also be intrusive, as will be seen in the case of the Finite Volume Method (Section 4.2).

4. The case of the Finite Volume Method

The solvers used in this study are based on the Finite Volume Method. The details of an efficient coupling scheme will be given in this framework.

The Finite Volume Method is a broadly used discretization technique. It is designed to solve conservation equations such as Eq. (1) on structured or unstructured meshes in a conservative way. For any further information about this method, the interested reader will refer to [18] for example.

4.1. Interface boundary condition

Let us consider a face \(f\) at the interface, in the case of matching meshes (Fig. 1). \(S_s\) and \(S_f\) are the centroids of the adjacent cells in the solid and fluid domains, respectively. \(I\) is the centroid of face \(f\).

The interface boundary conditions are computed at the coupling steps. Preliminary conditions are assessed at the end of the previous cycle, either at the cell centroids \((\bar{T}_s^0, \bar{T}_f^0)\) or at the interface \((T_{I,s}^0, T_{I,f}^0, \Phi_{I,s}, \Phi_{I,f}\), see Fig. 1). In general, \(T_{I,s}^0 \neq T_{I,f}^0\) and \(|\Phi_{I,s}^0| \neq |\Phi_{I,f}^0|\) because of the independent computations of the solid and fluid domains during a cycle. The coupling method calls an interpolation scheme, which will be specified in the current section, to compute a single couple of values for temperature \((T_I)\) and heat flux \((\Phi_I)\) at the interface.

4.1.1. Assumptions

To derive relevant estimators of \(T_I\) and \(\Phi_I\), a simplified typical case is addressed. Thus, the current interpolation scheme is based on two simplifying assumptions:

1. no wall function is used to solve the fluid. It means that either the flow is laminar or the grid is fine enough to ensure that the temperature gradient at the interface is small (for instance in the adjacent cells of volume \(V_s\) in the solid, in the framework of the Finite Volume Method). Besides, \(\Delta E_s\) and \(\Delta E_f\) can be applied before or after updating the coupling boundary conditions. They can even be distributed over a given number of iterations of the following cycle. These options are called pre-update, post-update or distributed corrections, respectively (Fig. 6).

Consequently, the corrective energy is spatially (through the use of \(k\)) and temporally distributed. There are several ways to make these distributions. All of them ensure conservativity and improved accuracy but they are discriminated by their impact on stability.

Step 1 requires that the solvers compute the thermal energy transferred across each face of the interface during a cycle. It may be an intrusive part of the the coupling method: integration of \(E\) (Eq. (9)) must be implemented into the solvers. Step 3 can also be intrusive, as will be seen in the case of the Finite Volume Method (Section 4.2).

The Finite Volume Method is a broadly used discretization technique. It is designed to solve conservation equations such as Eq. (1) on structured or unstructured meshes in a conservative way. For any further information about this method, the interested reader will refer to [18] for example.

4.1. Interface boundary condition

Let us consider a face \(f\) at the interface, in the case of matching meshes (Fig. 1). \(S_s\) and \(S_f\) are the centroids of the adjacent cells in the solid and fluid domains, respectively. \(I\) is the centroid of face \(f\).

The interface boundary conditions are computed at the coupling steps. Preliminary conditions are assessed at the end of the previous cycle, either at the cell centroids \((\bar{T}_s^0, \bar{T}_f^0)\) or at the interface \((T_{I,s}^0, T_{I,f}^0, \Phi_{I,s}, \Phi_{I,f}\), see Fig. 1). In general, \(T_{I,s}^0 \neq T_{I,f}^0\) and \(|\Phi_{I,s}^0| \neq |\Phi_{I,f}^0|\) because of the independent computations of the solid and fluid domains during a cycle. The coupling method calls an interpolation scheme, which will be specified in the current section, to compute a single couple of values for temperature \((T_I)\) and heat flux \((\Phi_I)\) at the interface.

4.1.1. Assumptions

To derive relevant estimators of \(T_I\) and \(\Phi_I\), a simplified typical case is addressed. Thus, the current interpolation scheme is based on two simplifying assumptions:

1. no wall function is used to solve the fluid. It means that either the flow is laminar or the grid is fine enough to ensure that the temperature gradient at the interface is small (for instance in the adjacent cells of volume \(V_s\) in the solid, in the framework of the Finite Volume Method). Besides, \(\Delta E_s\) and \(\Delta E_f\) can be applied before or after updating the coupling boundary conditions. They can even be distributed over a given number of iterations of the following cycle. These options are called pre-update, post-update or distributed corrections, respectively (Fig. 6).

Consequently, the corrective energy is spatially (through the use of \(k\)) and temporally distributed. There are several ways to make these distributions. All of them ensure conservativity and improved accuracy but they are discriminated by their impact on stability.

Step 1 requires that the solvers compute the thermal energy transferred across each face of the interface during a cycle. It may be an intrusive part of the the coupling method: integration of \(E\) (Eq. (9)) must be implemented into the solvers. Step 3 can also be intrusive, as will be seen in the case of the Finite Volume Method (Section 4.2).

4. The case of the Finite Volume Method

The solvers used in this study are based on the Finite Volume Method. The details of an efficient coupling scheme will be given in this framework.

The Finite Volume Method is a broadly used discretization technique. It is designed to solve conservation equations such as Eq. (1) on structured or unstructured meshes in a conservative way. For any further information about this method, the interested reader will refer to [18] for example.
4.1.2. Proposed interpolation scheme

The purpose of the interpolation scheme is to enforce Eq. (3), hence to update the values of temperature $T_i$ and heat flux $\Phi_i$ at the interface so that, at the coupling step $n = 1$ (which is the beginning of the first iteration of the cycle), $T_{l_i} = T_{l_f} = T_j$ and $\Phi_{l_i} = -\Phi_{l_f} = \Phi_i$. Thus at $n = 1$, given preliminary conditions provided by the coupled solvers $T_{l_i}^{n-1} - T_{l_f}^{n-1}$ and $T_{l_i}^{n-1} - T_{l_i}^{n-1}$, these updated interface values still satisfy Eq. (11) if

$$
\Phi_i = -h_s \left( T_i - T_{l_f}^0 \right) = h_f \left( T_i - T_{l_f}^0 \right),
$$

which implies that

$$
T_i = h_f T_{l_f}^0 + h_s T_{l_i}^0.
$$

Additionally, by eliminating $T_{l_i}^0$ and $T_{l_i}^0$ thanks to Eq. (11), Eq. (13) yields

$$
T_i = \frac{h_f T_{l_f}^0 + h_s T_{l_i}^0}{h_f + h_s}.
$$

It must be noted that $\Phi_{l_f}^0$ and $\Phi_{l_i}^0$ are produced by the coupled solvers. Thus, they account for the complete temperature gradient directly result from Eq. (8) with $\Phi_{l_f} = 0$ and $T_{l_f} = T_{l_i}^0$, in the first case and with $\Phi_{l_f} = -\Phi_{l_i}^0$ and $T_{l_f} = T_{l_i}^0$, in the second case.

Let $N_i$ denote the number of iterations per cycle in the solid domain (which means that $\Delta t = N_i \Delta t_s$), the boundary condition at the interface is ($\forall n \in (1, N_i)$)

$$
\begin{align*}
\left\{ \begin{array}{ll}
T_{l_i} = T_i &= T_{l_f} = T_{l_f}^0, & \Phi_{l_f} = 0, \\
T_{l_i} = T_i &= T_{l_f}^0, & \Phi_{l_f} = -\Phi_{l_i}^0.
\end{array} \right.
\end{align*}
$$

These expressions (which both have the same value for $h_i$) directly result from Eq. (8) with $\Phi_{l_f} = 0$ and $T_{l_f} = T_{l_i}^0$, in the first case and with $\Phi_{l_f} = -\Phi_{l_i}^0$ and $T_{l_f} = T_{l_i}^0$, in the second case.

Let $N_i$ denote the number of iterations per cycle in the solid domain (which means that $\Delta t = N_i \Delta t_s$), the boundary condition at the interface is ($\forall n \in (1, N_i)$)

$$
\begin{align*}
\left\{ \begin{array}{ll}
T_{l_i} = T_i &= T_{l_f} = T_{l_f}^0, & \Phi_{l_f} = 0, \\
T_{l_i} = T_i &= T_{l_f}^0, & \Phi_{l_f} = -\Phi_{l_i}^0.
\end{array} \right.
\end{align*}
$$

The same kind of relations can be obtained in the fluid domain.

4.1.3. Other possible interpolation schemes

**Scheme based on the literature.** ⊖ denotes here the scheme used by most authors, which is not based on a genuine interpolation. Indeed, the temperature of the solid domain is enforced at the fluid boundary and the heat flux of the fluid is assigned to the solid [5,19];

$$
\left\{ \begin{array}{ll}
S_F \left( \Phi_{l_f}^0, \Phi_{l_f}^0, T_{l_f}^0, T_{l_f}^0 \right) &= \Phi_{l_f}^0, \\
S_F \left( \Phi_{l_f}^0, \Phi_{l_f}^0, T_{l_f}^0, T_{l_f}^0 \right) &= T_{l_f}^0.
\end{array} \right.
$$

Thus, the Robin/Dirichlet version of this usual method, which is most robust [7,9,15,21], reads

$$
\left\{ \begin{array}{ll}
T_{l_f}^0 = T_{l_f}^0 &= \Phi_{l_f}^0, \\
T_{l_f}^0 = T_{l_f}^0 &= T_{l_f}^0.
\end{array} \right.
$$

**Scheme based on physical considerations.** Another scheme can be designed according to the following physical law. Let $h_f$ and $h_i$ be the fluid and solid effusivities ($\beta = \sqrt{2 \rho c}$, where, for the fluid, $c$ is the specific heat capacity at constant pressure, $c_{fp} = \frac{\gamma T}{\gamma - 1}$).

$$
\left\{ \begin{array}{ll}
\forall n \in (1, N_i), & h_f \left( T_{l_i}^0 - T_{l_i}^0 \right) = \Phi_{l_f}^0, \\
\forall n \in (1, N_i), & h_i \left( T_{l_f}^0 - T_{l_i}^0 \right) = \Phi_{l_i}^0.
\end{array} \right.
$$

Scheme ⊖ fulfills condition (7).
\[ T_i = \beta_T T_{i,n}^0 + \beta_s T_{s,n}^0 \]

is the initial temperature at the interface of two semi-infinite walls of initial temperatures \( T_{i,n}^0 \) and \( T_{s,n}^0 \). This physical consideration does not allow to define a heat flux at the interface (whose initial value is infinite). Consequently, the interpolation scheme \( \mathcal{H} \)

\[
\mathcal{H}_T (\Phi_{i,1S}, \Phi_{s,1F}, T_{i,1S}, T_{s,1F}) \]  

undefined.

\[
\mathcal{H}_1 (\Phi_{i,1S}, \Phi_{s,1F}, T_{s,1S}, T_{i,1S}) = \frac{\beta_T T_{i,1S}^0 + \beta_s T_{s,1S}^0}{\rho_s c_s \nu_s},
\]

can only be used with a Dirichlet/Dirichlet couple of boundary conditions:

\[
\begin{align*}
\forall n \in \{1, N_s\}, & \ T_{1F} = \mathcal{H}_1 (\Phi_{i,1S}, \Phi_{s,1F}, T_{s,1S}, T_{i,1S}), \\
\forall n \in \{1, N_s\}, & \ T_{1S} = \mathcal{H}_1 (\Phi_{i,1S}, \Phi_{s,1F}, T_{s,1S}, T_{i,1S}).
\end{align*}
\]

Scheme \( \mathcal{H} \) also fulfills condition \( \mathcal{F} \).

The use of adiabatic wall temperature. In schemes \( \mathcal{F}, \mathcal{S} \) and \( \mathcal{H} \), the temperature of the fluid is assessed as a static temperature \( T_{i,F} \). However, \( \Phi = h(T_{i,F} - T_{m}) \) is the convective heat flux (with convection coefficient \( h \)) across a surface of temperature \( T_{i,F} \), wetted by a flow of adiabatic wall temperature \( T_{m} \). It could therefore be attractive to use \( T_{m} = T_{i,F} \left( 1 + C \frac{1}{2} \left( \frac{M_{i,F}}{C_{0,i}} \right)^2 \right) \) instead of the static temperature \( T_{i,F}^0 \) (\( C \) is the recovery factor, \( M_{i,F} \) is the Mach number in cell \( F \)). This would be especially true for Robin conditions.

If a coarse grid is used, the heat flux could be computed more accurately. If the grid is refined, \( T_{i,F}^0 \) is almost equal to \( T_{i,F} \), because \( M_{i,F} \) is small in the vicinity of the wall. The heat flux is then almost unaffected.

Nevertheless, \( C \) depends on the simulated case (geometry, laminar or turbulent flow). Moreover, using \( T_{i,F}^0 \) instead of \( T_{i,F} \) may not improve the computation of \( T_i \) for all interpolation schemes. Eventually, in most cases, it may be better not to rely on this option to compensate for a poor mesh refinement.

Schemes \( \mathcal{F} \) (which is recommend here) and \( \mathcal{H} \) will be compared to scheme \( \mathcal{S} \), which is usually used in the literature, in Section 5.6. Besides, the results produced by the use of static and adiabatic wall temperatures will illustrate the discussion of the current section.

4.2. The conservative correction method

The Finite Volume Method ensures conservativity inside the domains. The corrective method described in Section 3.3 allows conservative coupled computations. This method computes the corrective energies \( \Delta E_i \) for the solid and \( \Delta E_s \) for the fluid. \( \Delta E_i \) and \( \Delta E_s \) can be assigned as corrective increments of temperature in the adjacent cells of (volume \( V \));

\[
\Delta T_{S,n} = -\frac{\Delta E_s}{\rho_s c_s \nu_s}, \quad \Delta T_{R,n} = \frac{\Delta E_i}{\rho_i c_p \nu_i},
\]

Directly adding \( \Delta T \) to the temperature of the adjacent cell could require alteration to the fluid and solid solvers. The corrected version of the original scheme \( \mathcal{F} \)

\[
\begin{align*}
T_i = & \mathcal{F}_T (\Phi_{i,1S}, \Phi_{s,1F}, T_{i,1S}, T_{i,1F} + \Delta T_{R}), \\
\Phi_i = & \mathcal{F}_T (\Phi_{i,1S}, \Phi_{s,1F}, T_{i,1S} + \Delta T_{S,n}, T_{i,1F} + \Delta T_{R}), \\
ht = & \frac{\Phi_i}{T_i}, \\
T_{int} = & T_{i,F} + \Delta T_{R}, \\
\Phi_{int} = & -\Phi_{i,F}.
\end{align*}
\]

is a less intrusive approach which achieves the pre-update correction (Fig. 6).

It is also possible to apply the correction as an additional heat flux at the interface. For a particular face (of area \( A \)) at this interface, at each time step \( n \),

\[
\Phi_{i,S} = \frac{\Delta E_s}{\frac{A}{\Delta T}}, \quad \text{and} \quad \Phi_{i,F} = \frac{\Delta E_i}{\frac{A}{\Delta T}}.
\]

where \( \tau _i \) is a characteristic time. If \( \forall n, \tau _i = \Delta t \) (duration of the cycle), the corrective energy is homogenously distributed all along the cycle. An amount of energy \( \frac{\Delta E}{\Delta t} \) is transferred at each time step of duration \( \Delta t \) (\( \Delta E = \Delta E_s \) in the solid or \( \Delta E = \Delta E_i \) in the fluid). \( \Delta E \) can also be distributed according to a geometric progression if at each step \( n \), a proportion \( k \Delta E_n \) of the remaining energy \( \Delta E_n = (1 - k)^{n-1} \Delta E \) is transferred. In this case, \( \tau _i = \frac{\Delta t}{k(1 - k)^{n-1}} \) \((k < 0,1)\). The corrective fluxes thus read \( \Phi_{i,S} = \kappa(1 - k)^{n-1} \Delta E_s / \Delta t \) and \( \Phi_{i,F} = \kappa(1 - k)^{n-1} \Delta E_i / \Delta t \).

The post-update correction is achieved for \( \kappa = 1 \); the whole corrective energy is assigned at the first iteration of the cycle.

\( \Phi_{i,S} \) and \( \Phi_{i,F} \) are added to the heat flux at the interface (for instance on the solid side and by using scheme \( \mathcal{F} \)) in the following way:

\[
\begin{align*}
\forall n \in \{1, N_s\}, & \quad \Phi_{i,S} = \mathcal{F}_s (\Phi_{i,1S}, \Phi_{i,F} + \Phi_{i,S}) + \Phi_{i,S}, \\
\forall n \in \{1, N_s\}, & \quad \Phi_{i,F} = \Phi_{i,S} - \Phi_{i,F} + \frac{1}{\mathcal{F}_s (\Phi_{i,1S}, \Phi_{i,F} + \Phi_{i,S}) - \Phi_{i,F}} (\Phi_{i,S} - \Phi_{i,F}).
\end{align*}
\]

which is a new corrected version of the Neumann and Robin conditions of Eq. (18). In the case of scheme \( \mathcal{S} \), the same approach can be used to obtain a corrected version of the Robin condition of Eq. (20). This method implies that no Dirichlet condition is selected in the domain that receives the correction.

4.3. Coupling methodology

The coupling method must be stable and accurate. Stability is mainly linked to the selection of the couple of boundary conditions and to the parameters of the conservative correction method (spatial and temporal distribution). Indeed, a stability analysis which was partially presented in reference [17] shows that:

1. the couple of boundary conditions ensuring the most robust computations is: a Robin condition for the solid and a Dirichlet condition for the fluid. This observation is quite usual [7,20], except that, here, a conservative correction method is included;
2. the most robust corrective approach is provided by a distributed correction (Fig. 6) assigned as a heat flux on the solid side of the interface (and only on this side) as follows:

\[
\Phi_{i,S} = -\frac{\Delta E_s}{\Delta t} \quad \text{and} \quad \Phi_{i,F} = 0.
\]

To that end, the weighting coefficient is \( k = 0 \), which means that the energy actually transferred across the interface during the cycle is computed on the fluid side (Fig. 5, Eq. (10)). Besides, the homogeneous temporal distribution of corrective energy is retained because it releases the whole energy the most robust
way. It must be noted that the pre-update and post-update corrections can produce slightly more accurate solutions than the distributed correction but they are less robust. As a consequence, the coupling boundary conditions are deduced from Eqs. (17), (18) and (27):

\[
\begin{align*}
\forall n \in \{1, N_f\}, & \quad T_{1f}^n = T_f(\Phi_{0}^{1}, T_{ff}, \Phi_{0}^{1}, T_{ff}^0), \\
\forall n \in \{1, N_s\}, & \quad \Phi_{0}^{1} = h_r(T_{1f}^n - T_{ff}^n) - \Phi_{0}^{1} + \Delta h_n,
\end{align*}
\]

and the global coupling methodology is depicted in Fig. 7. It must be pointed out that, apart from the calculation of the energy transferred across the interface during cycles ($E_s$ and $E_f$), this methodology is not very intrusive into the solvers.

Considerations of accuracy determine the selection of scheme $S$ and the use of a conservative approach in the current methodology, as will be illustrated in the following section.

5. Accuracy

The numerical results of the current method are compared with the analytical results provided by Pozzi et al. [1]. This test-case is intrinsically unsteady and is therefore appropriate to assess the accuracy of the current method.

5.1. Analytical case

In the considered analytical case, an infinite thick plate is wetted by a laminar flow impulsively accelerated to a constant Mach number $M_\infty$. The physical parameters are: $M_\infty = 3$, $Pr = 0.72$, $\gamma = 1.4$, $p = \frac{b \sqrt{Re_\infty \alpha_0(T_\infty)}}{L \lambda_s} = 1$, $t_0 = \frac{x_s}{b \sqrt{\alpha_0}} = 1$ ($x_s$ is the thermal diffusivity of the solid, $\alpha_s = \frac{x_s}{\rho_s C_s}$) and $\theta_s = \frac{T_s - T_\infty}{T_\infty} = 2$.

$Pr$ and $Re_\infty = \frac{\rho \lambda_s L \sqrt{\alpha_0}}{\mu (T_\infty)}$ are the Prandtl and Reynolds numbers, respectively. $b$ is the plate thickness and $L$ is a characteristic length ($L = b$). $T_\infty$ and $T_s$ are the flow velocity and the temperature far from the plate. $\theta_s$ is the relative non-dimensional temperature enforced at the rear face of the thick plate ($T_s$ is the dimensionless temperature at this point). $p$ and $t_0$ are two non-dimensional groups defined by Pozzi et al. as the main parameters of conjugate heat transfer problems.

Pozzi et al. obtained analytical results in compressible conditions thanks to the Stewartson - Dorodnitsyn transformation which requires a linear dependence of fluid thermal conductivity and viscosity with respect to temperature. As a consequence, $\mu_f(T) = \mu_{\infty} \frac{T}{T_\infty}$.

At $t = 0$, the flow Mach number suddenly becomes $M_\infty = 3$ and the temperature is $\theta_s = 0$ everywhere except at the rear face where $\theta_s = 2$. To compare the numerical approach to the analytical results, the main criterion will be the time-evolution of the non-dimensional temperature $\theta = \frac{T_s - T_\infty}{T_\infty}$ at the solid/liquid interface with respect to the non-dimensional time $\tau = \frac{t \sqrt{\alpha_0}}{L}$.

The numerical solution $\theta_n$ will thus be compared to the analytical solution $[1]$

\[
\theta_n(\tau) = \theta_{n0} \left[ 1 - \frac{\sum_{n=0}^{\infty} (-A)^n}{\sqrt{\pi \tau}} \right] + \frac{\sum_{n=0}^{\infty} (-A)^n}{\sqrt{\pi \tau}} \left[ \frac{2 n + 1}{2} \right],
\]

where $\theta_{n0} = \theta_{n0} \frac{\Delta h_n}{\lambda_s}$ is the initial solution, $\theta_{n0} = 2 \frac{\sqrt{\alpha_0}}{\sqrt{\pi \tau}} \arctan \left( \frac{2}{\sqrt{\pi \tau}} \right)$ is the non-dimensional adiabatic wall temperature, $\vartheta = \frac{2}{1 + \Lambda}$, $A = \frac{1 - \Lambda}{1 + \Lambda}$ and $\Lambda = \frac{\sqrt{\alpha_0}}{\sqrt{\pi \tau} \sqrt{Pr}}$.

5.2. Main parameters for the simulations

The method described in the current paper has been assessed on this test-case. The conservation laws [1] presented in Section 2.1 are numerically solved in the solid and fluid domains. The coupling method described in Section 4.3 is used to fulfill condition (3) at the interface of the solid and fluid domains. Then, unless explicitly specified, interpolation scheme $S$ is used according to Eq. (28).

The simulations were performed at a pressure $P_\infty = 101325$ Pa and with $b = 0.05$ m, $T_\infty = 300$ K and $\mu_{\infty} = 1.808 \times 10^{-5}$ Pa s. An extrapolation condition was used at the top boundary of the fluid domain (Fig. 8). The semi-infinite geometry was modeled by the use of periodicity conditions at both left and right ends of the domain.

The computations were performed on three grids described in Table 1. The finest one is shown in Fig. 8. Due to the semi-infinite

![Fig. 7. Coupling methodology. $h_s = h_f, T_r = T_{ref} = T_{eff} = -\Phi_{eff}$.

Fig. 8. Mesh for the verification case.](image-url)
configuration, only the transverse direction ($Y$) needs to be refined. There are five points in the streamwise direction for all three grids. The fluid and solid meshes were generated with matching nodes. The solid meshes are uniform ($d_s$ is the cell size in the transverse direction). In the fluid domain, the cell size increases from the wall ($d_h$) to the far-field, according to a geometrical law of common ratio 1.05.

The flow was integrated with an explicit forward Euler method which admits a CFL condition $\sigma < 1$ [16]. Here, $\sigma = 0.6$. A first-order HLLC upwind scheme (Harten Lax Van Leer scheme, version by Toro [23]) was used. Time integration of the solid was also performed with an explicit method. Thus, the integration Fourier number $a = \frac{2\pi\delta t}{\delta x^2}$ must be smaller than 0.5. Here, $a = 0.4$.

No turbulence model was used, since the case is laminar. This prevents the problems discussed in Section 4.1.1, related to wall functions.

5.3. Influence of the grid refinement

The computations were performed on the three grids in case $dt_0$ (defined in Table 2). In this case, there is a single time step per cycle in both fluid and solid domains: $\Delta t = \delta t_F = \delta t_S$.

All the simulations with scheme $\mathcal{I}$ are rather inaccurate (Fig. 9(b)). For scheme $\mathcal{J}$, apart from a discrepancy in the early part of the computation, there is very little difference between the results on the fine and medium grids. As a consequence, most of the following discussion will be based on computations on the medium grid (which are cheaper). The simulations on the coarsest grid are clearly inaccurate, which disqualifies this mesh. It is also interesting to observe that, for the coarse grid, the results of schemes $\mathcal{J}$ and $\mathcal{I}$ are very similar.

Moreover, with scheme $\mathcal{J}$, the simulations on the fine and medium grids exhibit a non-physical peak of temperature in the first moments of the computation (Fig. 9(a)). The reason is linked to issues of grid cell size, as will be explained in the following section.

5.4. Resolution of the unsteady boundary layer

The thickness $\delta$ of the laminar boundary layer over an infinite flat plate theoretically increases over time ($t$) as follows [24]:

$$\delta \simeq 3.64 \sqrt{\nu \tau}.$$  \hspace{1cm} (30)

The flow was integrated with an explicit forward Euler method which admits a CFL condition $\sigma < 1$ [16]. Here, $\sigma = 0.6$. A first-order HLLC upwind scheme (Harten Lax Van Leer scheme, version by Toro [23]) was used. Time integration of the solid was also performed with an explicit method. Thus, the integration Fourier number $a = \frac{2\pi\delta t}{\delta x^2}$ must be smaller than 0.5. Here, $a = 0.4$.

No turbulence model was used, since the case is laminar. This prevents the problems discussed in Section 4.1.1, related to wall functions.

5.3. Influence of the grid refinement

The computations were performed on the three grids in case $dt_0$ (defined in Table 2). In this case, there is a single time step per cycle in both fluid and solid domains: $\Delta t = \delta t_F = \delta t_S$.

All the simulations with scheme $\mathcal{I}$ are rather inaccurate (Fig. 9(b)). For scheme $\mathcal{J}$, apart from a discrepancy in the early part of the computation, there is very little difference between the results on the fine and medium grids. As a consequence, most of the following discussion will be based on computations on the medium grid (which are cheaper). The simulations on the coarsest grid are clearly inaccurate, which disqualifies this mesh. It is also interesting to observe that, for the coarse grid, the results of schemes $\mathcal{J}$ and $\mathcal{I}$ are very similar.

Moreover, with scheme $\mathcal{J}$, the simulations on the fine and medium grids exhibit a non-physical peak of temperature in the first moments of the computation (Fig. 9(a)). The reason is linked to issues of grid cell size, as will be explained in the following section.

5.4. Resolution of the unsteady boundary layer

The thickness $\delta$ of the laminar boundary layer over an infinite flat plate theoretically increases over time ($t$) as follows [24]:

$$\delta \simeq 3.64 \sqrt{\nu \tau}.$$  \hspace{1cm} (30)

$\nu = \frac{\rho}{\rho}$ is the kinematic viscosity of the fluid. Fig. 10 shows that the boundary layer is under-resolved in the beginning of all the computations: $\delta/d_h$ is smaller than 1, which means that the size of the first cell is larger than the boundary layer. For the coarse grid, this problem is encountered during almost the whole computation, which explains that the simulation cannot be accurate.

For the other grids, the bad resolution of the boundary layer is temporary and triggers peaks of temperature. For the medium grid, $\theta$ increases to 1.27 at $\tau = 3.7 \cdot 10^{-4}$, whereas the analytical solution is 0.73. At this time, $\delta \simeq 4d_h$ and $\nu^+ \simeq 3$ (also shown in Fig. 10). For

<table>
<thead>
<tr>
<th>Grid</th>
<th>Number of cells in the boundary layer at the end of the first cycle</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fine</td>
<td>0</td>
</tr>
<tr>
<td>Medium</td>
<td>0</td>
</tr>
<tr>
<td>Coarse</td>
<td>0</td>
</tr>
<tr>
<td>Medium</td>
<td>3</td>
</tr>
<tr>
<td>Medium</td>
<td>7</td>
</tr>
<tr>
<td>Medium</td>
<td>17</td>
</tr>
</tbody>
</table>

$\nu = \frac{\rho}{\rho}$ is the kinematic viscosity of the fluid. Fig. 10 shows that the boundary layer is under-resolved in the beginning of all the computations: $\delta/d_h$ is smaller than 1, which means that the size of the first cell is larger than the boundary layer. For the coarse grid, this problem is encountered during almost the whole computation, which explains that the simulation cannot be accurate.

For the other grids, the bad resolution of the boundary layer is temporary and triggers peaks of temperature. For the medium grid, $\theta$ increases to 1.27 at $\tau = 3.7 \cdot 10^{-4}$, whereas the analytical solution is 0.73. At this time, $\delta \simeq 4d_h$ and $\nu^+ \simeq 3$ (also shown in Fig. 10). For

<table>
<thead>
<tr>
<th>Grid</th>
<th>Number of cells in the boundary layer at the end of the first cycle</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fine</td>
<td>0</td>
</tr>
<tr>
<td>Medium</td>
<td>0</td>
</tr>
<tr>
<td>Coarse</td>
<td>0</td>
</tr>
<tr>
<td>Medium</td>
<td>3</td>
</tr>
<tr>
<td>Medium</td>
<td>7</td>
</tr>
<tr>
<td>Medium</td>
<td>17</td>
</tr>
</tbody>
</table>

$\nu = \frac{\rho}{\rho}$ is the kinematic viscosity of the fluid. Fig. 10 shows that the boundary layer is under-resolved in the beginning of all the computations: $\delta/d_h$ is smaller than 1, which means that the size of the first cell is larger than the boundary layer. For the coarse grid, this problem is encountered during almost the whole computation, which explains that the simulation cannot be accurate.

For the other grids, the bad resolution of the boundary layer is temporary and triggers peaks of temperature. For the medium grid, $\theta$ increases to 1.27 at $\tau = 3.7 \cdot 10^{-4}$, whereas the analytical solution is 0.73. At this time, $\delta \simeq 4d_h$ and $\nu^+ \simeq 3$ (also shown in Fig. 10). For

<table>
<thead>
<tr>
<th>Grid</th>
<th>Number of cells in the boundary layer at the end of the first cycle</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fine</td>
<td>0</td>
</tr>
<tr>
<td>Medium</td>
<td>0</td>
</tr>
<tr>
<td>Coarse</td>
<td>0</td>
</tr>
<tr>
<td>Medium</td>
<td>3</td>
</tr>
<tr>
<td>Medium</td>
<td>7</td>
</tr>
<tr>
<td>Medium</td>
<td>17</td>
</tr>
</tbody>
</table>

$\nu = \frac{\rho}{\rho}$ is the kinematic viscosity of the fluid. Fig. 10 shows that the boundary layer is under-resolved in the beginning of all the computations: $\delta/d_h$ is smaller than 1, which means that the size of the first cell is larger than the boundary layer. For the coarse grid, this problem is encountered during almost the whole computation, which explains that the simulation cannot be accurate.

For the other grids, the bad resolution of the boundary layer is temporary and triggers peaks of temperature. For the medium grid, $\theta$ increases to 1.27 at $\tau = 3.7 \cdot 10^{-4}$, whereas the analytical solution is 0.73. At this time, $\delta \simeq 4d_h$ and $\nu^+ \simeq 3$ (also shown in Fig. 10). For

<table>
<thead>
<tr>
<th>Grid</th>
<th>Number of cells in the boundary layer at the end of the first cycle</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fine</td>
<td>0</td>
</tr>
<tr>
<td>Medium</td>
<td>0</td>
</tr>
<tr>
<td>Coarse</td>
<td>0</td>
</tr>
<tr>
<td>Medium</td>
<td>3</td>
</tr>
<tr>
<td>Medium</td>
<td>7</td>
</tr>
<tr>
<td>Medium</td>
<td>17</td>
</tr>
</tbody>
</table>
the fine grid, the phenomenon is restricted to very early times but the maximum value of $\delta$ is even larger, 1.77, at $\tau = 6.15 \times 10^{-2}$ ($\delta \simeq 13d_t$ and $y^+ \simeq 0.16$). The numerical error decreases as the resolution of the boundary layer gets better (as $\delta/d_t$ increases or $y^+$ decreases). This study does not quantify a threshold value for $\delta/d_t$ and $y^+$, but at least a few points in the boundary layer seem to be required during the whole computation.

A major goal of the current paper is to assess the accuracy of our coupling method and to demonstrate that it is efficient. The following section will thus deal with the sensitivity of the results of this method with respect to several parameters. Moreover, the computations presented in Section 5.3 have already shown that our method is more accurate than the usual method (scheme $\mathcal{S}$). This point will be discussed more thoroughly in Section 5.6.

5.5. Influence of the duration of cycles and of the conservative correction

Figs. 11 and 12 show the influence of the coupling frequency on the accuracy of simulations. Dimensional and non-dimensional cycle durations ($\Delta t$ and $\Delta t_f$) are provided in Table 2 for cases $dt_0$ to $dt_3$. Eq. (30) has been used to assess the number of cells included in the boundary layer at the end of the first cycle (which is also mentioned in Table 2). The problem of the temporarily under-resolved boundary layer, discussed in the previous section, was only encountered in case $dt_0$. In case $dt_1$, there are already three cells in the boundary layer at the end of the first cycle. Therefore, the peak of temperature does not occur (Fig. 11).

But as a counterpart, if the conservative correction method is not used, it is clear that increasing $\Delta t$ from case $dt_0$ to case $dt_3$ leads to a significant reduction in the accuracy of the computation during the major part of the simulation (Fig. 12); the relative error $e = \frac{\theta_n - \theta_0}{\theta_0}$ increases when $\Delta t$ becomes larger.

The influence of the conservative corrections on accuracy is significant. Fig. 12 shows that, for the non-conservative approach, the relative error decreases by about one order from case $dt_2$ to case $dt_1$. But the accuracy of the conservative computation in case $dt_2$ and of the non-conservative simulation in case $dt_1$ is of the same order (although the coupling steps are around ten times less frequent in the second case). The same conclusion is given by the comparison between simulations $dt_1$ and $dt_0$. This test-case thus shows that the conservative method improves accuracy while reducing the frequency of the coupling steps.

5.6. Dependence on the scheme

The influence of the spatial discretization scheme has been assessed on the medium grid in case $dt_1$. For instance, using an AUSM scheme instead of an HLLC scheme does not significantly affect the results.

A major issue is the influence of the interpolation scheme. The computations with schemes $\mathcal{S}$, $\mathcal{G}$ and $\mathcal{H}$ (schemes presented in Sections 4.1.2 and 4.1.3) are compared in Fig. 13. The usual scheme ($\mathcal{S}$) has been tested with and without conservative corrections, even if most authors use a non-conservative approach. Fig. 13 shows that scheme $\mathcal{S}$ is the most accurate one. Besides, conservative corrections improve the results produced by scheme $\mathcal{S}$. It is also true for scheme $\mathcal{H}$, which is only slightly more accurate than scheme $\mathcal{S}$.

An additional remark is that the results produced by scheme $\mathcal{S}$ in case $dt_0$ are better on the medium grid than on the fine one (Fig. 9). Likewise, on the medium grid, scheme $\mathcal{G}$ is slightly more accurate in case $dt_1$ than in case $dt_0$. These counter-intuitive results are probably due to the low resolution of the boundary layer in the very beginning of the calculations. But a better understanding of this numerical behavior would require a thorough investigation which is beyond the scope of the current paper.

It should be pointed out that, although built after physical considerations, scheme $\mathcal{H}$ is less accurate than scheme $\mathcal{S}$. The problem is linked to an issue already raised in Section 4.1.3. Indeed, the analytical solution (Eq. (29)) shows that the initial temperature is

$$T_i(t = 0 s) = \frac{\beta_f T_{aw}(t = 0 s) + \beta_b T_b(t = 0 s)}{\beta_f + \beta_b} \rightarrow \frac{\beta_f T_{aw}(t = 0 s) + \beta_b T_b(t = 0 s)}{\beta_f + \beta_b}$$

which differs from Eq. (21) used for scheme $\mathcal{S}$ (Eq. (22)) because the adiabatic wall temperature $T_{aw}$ must be used instead of the static temperature $T_i$. The poor accuracy of scheme $\mathcal{H}$ is thus mainly a consequence of the large flow velocity, which implies a noticeable difference between $T_{aw}$ and $T_i$. Schemes $\mathcal{G}$ and $\mathcal{S}$ usually do not account for this effect. Of course, none of them accurately assesses the initial interface temperature.

However, Fig. 14 shows that, as expected in Section 4.1.3, using $T_{aw}$ does not systematically improve the accuracy of the simulations. The initial temperature produced by scheme $\mathcal{G}$ is too large (whereas it is too small when using $T_{aw}$). In general for scheme $\mathcal{S}$, using the static temperature produces slightly better results than using the adiabatic wall temperature.
On the contrary, $\mathcal{J}(\phi_{T_0}^0, \phi_{T_F}^0, T_{00}, T_{01})$ is slightly more accurate than $\mathcal{J}(\phi_{T_0}^0, \phi_{T_F}^0, T_{00}, T_{01})$. The initial temperature is especially computed very accurately (as expected from Eq. (31)). However, regardless of the selected temperature, scheme $\mathcal{F}$ becomes more accurate than scheme $\mathcal{H}$ after a short time. Besides, the recovery factor for the laminar boundary layer over a flat plate has been inferred from Pozzi's paper [1]: $C_r = \frac{4 \arctan \left( \frac{\sqrt{2}}{\sqrt{3}} - 1 \right)}{\pi \sqrt{2} - 1}$ (which is close to the usual value $\sqrt{Pr}$). But it must be recalled that $C_r$ depends on the case.

It must also be noted that using $T_{in}$ instead of $T_{in}$ does not noticeably change the results for scheme $\mathcal{S}$. This is not surprising: the use of $T_{in}$ was expected to especially affect Robin boundary conditions (Section 4.1.3). But whereas the Dirichlet boundary condition is affected by the use of $T_{in}$ in $\mathcal{S}$, the Robin boundary condition, which depends neither on $S_T$ nor on $S_p$, remains unaffected (Eq. (20)).
As a conclusion, all these comparisons show that scheme $\mathcal{F}$ (used with static temperature) is the most efficient interpolation scheme.

5.7. Main keys to successful coupled simulations

A poor accuracy of the fluid computation (for instance because of a temporary bad resolution of the boundary layer or because of turbulence [25]) leads to rather large errors in heat flux and temperature at the solid/fluid interface. For this reason, the duration of cycles $\Delta t$ must not be too large compared to the physical duration of the phenomenon. For instance in case $\Delta t_3$, although stable, the computation is not accurate at all (Fig. 11).

$\Delta t$ must not be too small either. Indeed, if $\Delta t$ is large enough, the temporal discretization of the two domains is limited by their own stability condition, which ensures optimal calculation for all solvers. Moreover, as already discussed by Duchaine et al. [15], decreasing the frequency of coupling steps allows to reduce the time consumed by coupling routines, and consequently to reduce the global computational cost. In the case discussed in this paper, the most appropriate option consists in using the cycle duration of case $\Delta t_1$ along with conservative corrections. The results are as accurate as in case $\Delta t_0$, although the number of coupling steps is 160 times smaller. Of course, the choice of $\Delta t$ depends on the physics of the computed case. But this test-case shows that a conservative approach allows to increase the duration of cycles.

6. Conclusion

A methodology of numerical coupling has been developed for applications of unsteady conjugate heat transfer. It is mainly based on a parallel coupling strategy. The fluid and solid solvers are simultaneously run and the boundary conditions are periodically updated at their interface. This method is particularly efficient for unsteady computations, since it does not require the use of an iterative method at each time step. At the interface, the boundary conditions are usual (Dirichlet, Neumann or Robin). They are followed by an interpolation scheme which must fulfill a consistency condition. Several interpolation schemes have been defined and assessed in this article. The best scheme is consistent with the spatial discretization of the coupled solvers.

Moreover, heat flux is not strictly conserved at the solid/fluid interface. This is due to the independent behaviors of the solvers between the updates of their interface condition. A corrective method has been developed in order to maintain the intrinsic conservativity of the Finite Volume Method at the interface boundaries. The corrective process is based, first, on the evaluation of the difference between the heat fluxes on the two sides of the interface and, second, on the use of corrective fluxes.

To obtain a robust method, the fluid domains will usually receive a Dirichlet boundary condition at the interface and the solid domains will receive a Robin boundary condition. For the same reason, the whole corrective fluxes will also be assigned to the solid domains.

A verification of the coupling method has been carried out in this paper. In the case of an infinite thick plate wetted by a suddenly accelerated flow, evidence has been given that the proposed interpolation scheme is particularly efficient and that the conservative method leads to a significant gain in accuracy.

References


