Computers and Fluids 134–135 (2016) 146–156

Contents lists available at ScienceDirect

Computers and Fluids

journal homepage: www.elsevier.com/locate/compfluid

Unified formulation for modeling heat and fluid flow in complex real industrial equipment

Q. Schmid[a], E. Marchal[b], J.M. Franchet[b], J. Schwartz[b], Y. Mesri[a], E. Hachem[a,*]

[a] Ecole des Mines de Paris, Centre de Mise en Forme des Matériaux (CEMEF), UMR CNRS 7635 Sophia-Antipolis, France
[b] Snecma, Site de Gennevilliers, 171 bd de Valmy - BP 31 Colombes Cedex 92702, France

ARTICLE INFO

Article history:
Received 31 August 2015
Revised 6 May 2016
Accepted 8 May 2016
Available online 10 May 2016

Keywords:
Natural convection
Forced convection
Turbulent gas–solid interaction
Immersed method
Complex geometries

ABSTRACT

The direct simulation of conjugate heat transfer has recently attracted considerable attention in many industrial applications. However, the underlying physical phenomena are less clear due to the intrinsically interactions occurring at the interface of different materials at different temperatures with the surrounding turbulent gas. To simulate these direct exchanges, here we study a unified multi-material formulation in two and three-dimension. The coupled system is solved using stabilized finite element methods combined with a posteriori error estimator for anisotropic mesh adaptation enhancing the interface representation. Computational results are compared with experimental data in complex 3D industrial setups with up to four different materials. The results have demonstrated that convection, conduction and radiative heat transfer can be simultaneously used and easily applied yielding accurate predictions.

1. Introduction

With the increasing demands for reducing energy consumption and the pollutant emissions, the simulation of turbulent gas–solid has considerably attracted several researchers in the past few years [1–3]. Indeed, applications range from thermal energy conversion systems, industrial furnaces, quenching and cooling control systems, to environmental processes such as pollutant control and de-icing issues. The objective is generally to understand the physical problem, to optimize the design and to control the process. Recall also that many factors play an important role in such processes: minimization of local temperature gradients, ensuring a uniform temperature within the load, avoiding surface defects such as skid marks, minimizing energy usage, maximizing the quality of the steel product in terms of hardness, toughness and resistance [3,4]. Therefore, it is essential in most of these situations to treat precisely the heat exchange at the fluid–solid or the solid–solid interfaces and to evaluate with high accuracy the transfer and the gradients.

In many works on conjugate heat transfer, we use classically appropriate heat transfer coefficients to account for a surrounding fluid or for another solid in contact, and thus we sidestep the interfacial heat exchanges problems. Indeed, the standard strategy is to divide the global domain into several local subdomains where a local model (equation to be solved) can be carried out independently. The global solution is then reconstructed by suitably piecing together local solutions from individually modeled subdomains. However, these transfer coefficients, deduced in general by expensive experimental studies, may not reflect necessary a complex situation, large diversity of shapes and geometries, the contact between multi-material, or most importantly the turbulent gas-solid interactions in confined domains.

In this context, there is a renewed interest in modeling the thermal coupling using full Eulerian frameworks in which the different domains are taken into account by the use of a characteristic function (level set). Consequently, one set of equations with different thermal properties is solved. It avoids the use of empirical data to determine the heat transfer coefficient at the fluid–solid interface. The heat exchange is obtained directly by simulating the turbulent surrounding gas in the whole domain. The boundary conditions for radiative heat transfer are replaced by solving an additional radiative transport equation (RTE) in both domains. It will generate a volume source term that is in turn introduced into the energy equation and rendered by the sharp discontinuity of the temperature and the properties of the materials.

In spite of large number of research works on Eulerian formulation with or without level sets [5–7], less attention has been paid to turbulent gas–solids flows and no experimental validation on industrial setup were obtained [8]. This is explained by the difficulty to obtain sharply defined interfaces which makes the

* Corresponding author. Fax: +33 492389752.
E-mail addresses: quentin.schmid@mines-paristech.fr (Q. Schmid), edouard.marchal@snecma.fr (E. Marchal), jean-michel.franchet@safran.fr (J.M. Franchet), julien.schwartz@snecma.fr (J. Schwartz), youssef.mesri@mines-paristech.fr (Y. Mesri), Elie.Hachem@mines-paristech.fr (E. Hachem).

http://dx.doi.org/10.1016/j.compfluid.2016.05.007
0045-7930/

approach inherently challenging to use and time consuming in particular for high Reynolds number flows with boundary layers.

In this work, a fully Eulerian unified framework is proposed to simulate solid–gas high Reynolds number flows with heat transfer. We particularly focus on accurately representing the interfaces of different physical domains and on dealing with the conjugate heat transfer that occurs among them in complex and challenging configurations. Therefore, we refer to the use of an immersed stress method coupled with an a posteriori error estimator for anisotropic mesh adaptation. The distinguish features of this work, compared to [8] resides on the capability to generate extremely stretched anisotropic elements at the interfaces, on the ability to control the number of grid points in the mesh, and finally on solving the coupled system using a three-field stabilized finite element to obtain oscillation-free solutions.

The proposed approach is first applied to simulate an illustrative example and then it is used to simulate three real complex industrial problems with natural convection, forced convection and radiative heat transfer. The results show that even when several layers of different materials are used, the numerical simulations are in very good agreement with experimental ones.

This paper is organized as follows: first, we present a detailed description of the immersed method using both the level set function and the anisotropic mesh adaptation. The time-dependent, three dimensional, coupled problem is given in Section 3 and completed by the discretization as well as the stabilized finite element methods for solving these equations. In Section 4, the numerical performance of the proposed method is demonstrated by means of 2D test cases and 3D real industrial problems. Comparisons with the literature and with the available experimental data are provided. Finally, conclusions and perspectives are outlined.

2. Eulerian framework

Immersed Methods are very practical to setup complex numerical simulations in computational engineering. They are used intensively for fluid-structure interaction and in this work are dedicated to conjugate heat transfer analysis. We recall that they are based on solving a single set of equations for the whole computational domain and treating different subdomains as a single fluid with variable material properties. This section presents the complete description, which in turn is structured into three subsections: immerse and define the immersed solid using the level-set function, apply the anisotropic mesh adaptation in the vicinity of the gas–solid interface and mix the thermo-physical properties appropriately for both domains.

2.1. Level set approach

At any point $\mathbf{x}$ of the computational domain Ω, the level-set function α corresponds to the signed distance from Γ_{im}. In turn, the interface Γ_{im} is given by the zero isovalue of the function α:

$$\begin{aligned} &\alpha(\mathbf{x}) = \pm d(\mathbf{x}, \Gamma_{\text{im}}), \mathbf{x} \in \Omega, \\ &\Gamma_{\text{im}} = \{\mathbf{x}, \alpha(\mathbf{x}) = 0\}. \end{aligned} \tag{1}$$

In this paper, the following sign convention is used: $\alpha \geq 0$ inside the solid domain defined by the interface Γ_{im} and $\alpha \leq 0$ outside this domain. As explained, the signed distance function is used to localize the interface of the immersed solid but it is also used to initialize the desirable properties on both sides of the latter.

Indeed, the physical and thermodynamic properties in the domain are then calculated as a function of α; for instance, the mixed density is calculated using a linear interpolation between the values of the density in the fluid and the solid:

$$\rho = \rho_f H(\alpha) + \rho_s (1 - H(\alpha)) \tag{2}$$

where H is a smoothed Heaviside function given by:

$$H(\alpha) = \begin{cases} 1 & \text{if } \alpha > \varepsilon \\ \frac{1}{2}\left(1 + \frac{\alpha}{\varepsilon} + \frac{1}{\pi}\sin\left(\frac{\pi\alpha}{\varepsilon}\right)\right) & \text{if } |\alpha| \leq \varepsilon \\ 0 & \text{if } \alpha < -\varepsilon \end{cases} \tag{3}$$

ε being a small parameter such that $\varepsilon = O(h)$ and h the mesh size in the normal direction to the interface. It is computed using the following expression:

$$h_{im} = \max_{j,l \in K} \nabla\alpha \cdot \mathbf{x}^{jl} \tag{4}$$

where $\mathbf{x}^{jl} = \mathbf{x}^l - \mathbf{x}^j$ and K is the mesh element.

2.2. Anisotropic mesh adaptation

The difficulty to obtain sharply defined interfaces makes the Eulerian approach inherently challenging to use in particular for high Reynolds number flows with boundary layers. Moreover, the discontinuity of the properties of different materials across the interface add an extra numerical difficulty to handle on the finite element solvers. Therefore, we briefly describe in this section the implementation details of anisotropic mesh adaptation designed to ensure accurate calculation of the temperature distribution along the gas–solid interface. This is critical for a correct modeling of industrial experiments. The algorithm proposed here gradually refines the mesh when approaching the interface. In this way, the mesh becomes locally refined which enables to sharply define the interface and to save a great number of elements compared to a classical isotropic refinement. This anisotropic mesh adaptation algorithm is built in order to compute a mesh and a numerical solution. Both are implicitly coupled since the solution is computed on the mesh and the mesh is specifically adapted to solution. This coupling is attained by iterating a fixed-point algorithm. At each stage, a numerical solution is computed on the current mesh and has to be analyzed by means of an error estimate. The considered error estimate aims at minimizing the interpolation error in norm L_p, independently of the problem at hand. From the metric analysis in [9], an analytic expression of the optimal metric is derived which minimizes the interpolation error in norm L_p (p is here chosen equal to 2). Let u be an analytic solution defined on computational domain and let N denotes the desired number of elements for the mesh, the aim is to create the "best" mesh $\mathbf{M}$, i.e., the optimal continuous metric $\mathcal{M}$, to minimize the interpolation error $||u - \Pi_h u||_2$ in L_2 norm. $\Pi_h u$ denotes the linear interpolate of u on $\mathbf{M}$. The local interpolation error in the neighborhood of a point p is given by:

$$e_{\mathcal{M}}(p) = \sum_{i=1,3} h_i^2 \left| \frac{\partial^2 u}{\partial^2 \alpha_i^2} \right| \tag{5}$$

where h_i stand for mesh size and $\frac{\partial^2 u}{\partial^2 \alpha_i^2}$ represents the eigenvalues of the Hessian of variable u. The objective is to find the metric $\mathcal{M}$ that minimizes the error under the constraint of a fixed number of elements N. To this end, we have to solve the following optimization problem:

$$min_{\mathcal{M}} \mathcal{E}(\mathcal{M}) = \quad min_{\mathcal{M}} \int_\Omega (e_{\mathcal{M}})^2 d\Omega \tag{6}$$

under the constraint:

$$\mathcal{C}(\mathcal{M}) = C_0^{-1} \int_\Omega \Pi_{i=1,3} \frac{1}{h_i} d\Omega = N$$

where C_0^{-1} is the volume of a regular tetrahedron. Following the lines in [9,10], the optimal metric solution of (6) in the L_2 norm