IV International Conference on Computational Methods for Coupled Problems in Science and Engineering
COUPLED PROBLEMS 2011
M. Papadrakakis, E. Oñate and B. Schrefler (Eds)

AERO-THERMO-MECHANICAL COUPLING FOR FLAME-WALL INTERACTION

B. BAQUE*, M.-P. ERRERA, A. ROOS*,
F. FEYEL*, E. LAROCHE***, D. DONJAT*****

*ONERA - Department of Metallic Materials and Structures
BP 72 - 29 avenue de la Division Leclerc, 92322 Châtillon Cedex, France
e-mail: benedicte.baque@onera.fr, web page: http://www.onera.fr/

**ONERA - Department of Computational Fluid Dynamics and Aeroacoustics
BP 72 - 29 avenue de la Division Leclerc, 92322 Châtillon Cedex, France

**ONERA - Department of Aerodynamics and Energetics Modeling
BP 74025 - 2 avenue Edouard Belin, 31055 Toulouse Cedex 4, France

Key words: aero-thermo-mechanical algorithm, heat transfer, thermal strain

Abstract. This paper investigates a flame-wall interaction consisting of a premixed flame impinging on a metallic plate. This is a coupled problem as the heat transfer from the flame increases the temperature of the plate and bends it, which in turn modifies the shape of the flame. This study aims at designing an aero-thermo-mechanical coupling between both codes CEDRE (Computational Fluid Dynamics) and Z-SeT (computational solid mechanics and heat conduction) to simulate this complex system. Numerical results for aero-thermal coupling are compared with experimental data.

1 INTRODUCTION

Numerical simulations of real-world engineering systems involve a large number of physical phenomena. In the case of fluid-structure interaction with heat transfer such as the one occurring in a flame-wall interaction, many phenomena are strongly coupled: convection in the fluid, heat conduction through the solid and its deformation. Separate simulations of Computational Fluid Dynamics (CFD), computational solid mechanics and conduction generally do not give accurate solutions due to the assumptions at boundaries (such as heat flux, temperature, pressure or position) made in the separate calculations.

In the fluid-solid coupling, information provided by each model is complementary. The external coupling approach has the advantage to build upon the experience put into the separate solvers and to use the most appropriate methods in each discipline. It is therefore

Figure 1: Flame-wall interaction setup (ONERA Toulouse).

an attractive solution, but a robust and efficient algorithm between the fluid and the solid media is required.

The goal of this paper is to analyze a turbulent premixed flame impinging on a wall. The numerical problem is challenging due to the great time disparities of the physical models in interaction. Furthermore, a procedure has to be defined over a long period of time, such as the one of the experiment (300 s), with respect to the chemical and aerodynamical characteristic times. Thus, the numerical coupled strategy must provide reasonable solutions at acceptable computing efforts.

In this paper, a numerical approach is validated by comparison to experimental data. The external coupling involves three independent solvers (CEDRE for fluid, and Z-SeT for structure and heat conduction) and needs an efficient algorithm to predict the transient aero-thermo-mechanical process. The coupling strategy is responsible for determining which variables are relevant in the multi-physics interaction and which ones must be exchanged at the interfaces. The resulting algorithm carries out the temporal exchanges and is closely related to the physics to be studied.

2 FLAME-WALL INTERACTION

The experiment has been performed at the ONERA Toulouse (see Roinard *et al.* [4]). A Bunsen burner produces a premixed propane-air flame, with an equivalence ratio of $r = 1.2$ (fuel excess) and an ejection velocity of $V_0 = 2,4$ m/s. At ejection, it corresponds to a Reynolds number Re of 9000, which means that the flame is turbulent. The maximum temperature inside the flame is about 1700°C. Two flame to wall distances H (i.e. between the exit of the burner nozzle and the plate) are tested successively: 54 and 108 mm, for a nozzle diameter $D = 41$ mm. The sample wall consists of metallic plates made of two different aeronautical alloys (TA6V and INCO600), with dimensions of 400 × 400 mm and a thickness of 3 mm. The experiment lasts 300 s. The transient evolution of the system is recorded for temperatures on the upper plate surface (infra-red thermography), as well as its position (spectroscopic photogrammetry) and the flame front location (ultra-violet camera). An envelope flame is observed: hot burnt gases are trapped between the

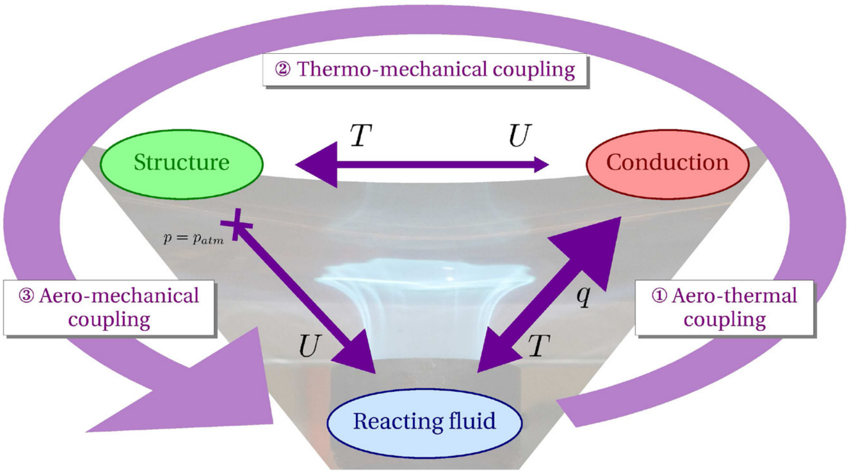

Figure 2: Mutual influences of the different physics in flame-wall interaction.

chemical reaction zone and the wall, which implies a maximal heat transfer at the center of the plate.

Mutual influences and relative emphasis of every physics in the flame-wall interaction are presented in Figure 2. The flame stimulates the coupled system through the transfer of a convective heat flux q (see Milson & Chigier [3]). Thus, the aero-thermal coupling (Conjugate Heat Transfer) is predominant. In reaction, the temperature T increases through the plate, which bends under dilatation and creep (thermo-mechanical coupling). The evolution of the position of the plate U causes a change of the shape of the flame (aero-mechanical coupling), which modifies the flux profile. All along the experiment, the flame pressure p is equivalent to the atmospheric pressure p_{atm}. As the variation of this pressure is negligeable, the system can be simplified.

3 AERO-THERMO-MECHANICAL COUPLING

The resolution of the combustion equations is very time-consuming. As a result, the whole computation time is mainly determinated by CFD. Consequently, the coupling strategy needs to minimize the fluid computation time. Thus, the algorithm in Figure 3 shows a minimum number of fluid calls. Even if the aero-thermal coupling is predominant in the flame-wall interaction, updating the flow with the best prediction of the plate position seems to be cheaper in terms of iterations to update the deforming fluid mesh. For simplicity, the algorithm has been split into two parts.

The first part consists in performing the thermo-mechanical coupling. After the transient conduction computation (step ①), the temperature T at the interface is sent to the structure (step ②), which determines the new plate position U at this coupling instant (steps ③ and ④). An iterative processus is generally needed to reach temperature and

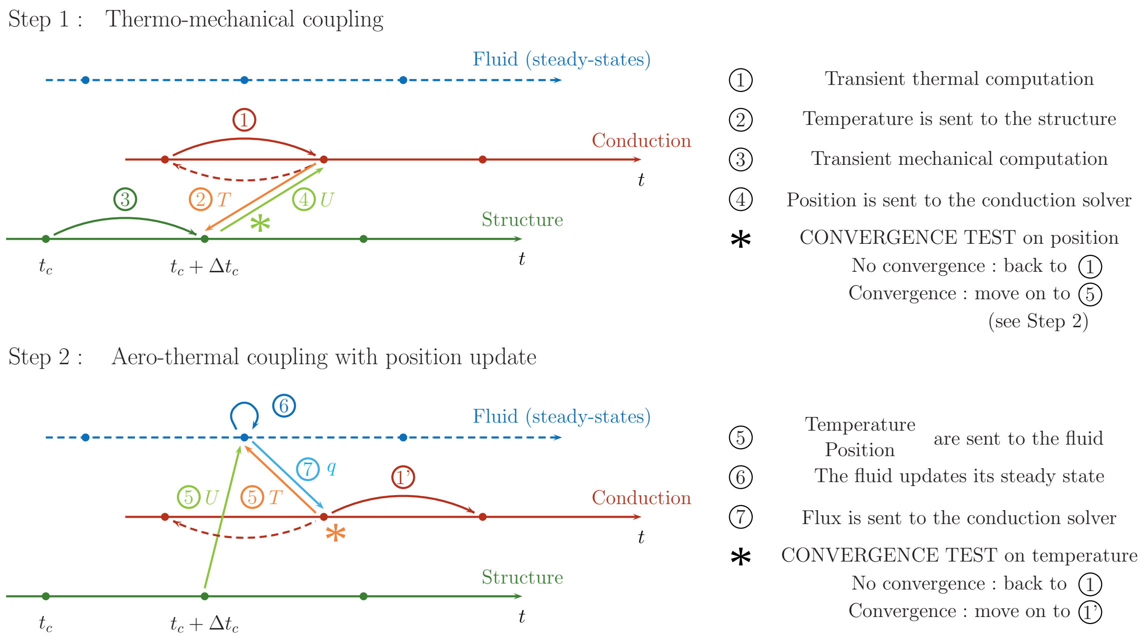

Figure 3: Aero-thermo-mechanical coupling algorithm.

position equilibria.

As for the second part, once the two fields (T, U) are converged at $t_c + \Delta t_c$, they are simultaneously sent to the fluid (step ⑤). Thus the fluid mesh can be updated according to the new plate position before computing a new steady-state flow (step ⑥). The aero-mechanical coupling is limited to the update of the plate position (one-way coupling). Then, the fluid sends the heat flux q to the conduction solver (step ⑦).

Because of the significant discrepancy of characteristic times between fluid and solid, a simplified aero-thermal coupling based on a Conjugate Heat Transfer (see Baqué *et al.* [1]) is used: the flow solution may be considered as a sequence of steady states, whereas in the solid the fields evolve in a fully transient manner. In order to reach temperature and flux equilibria, a fixed point procedure - which includes the thermo-mechanical coupling loop - is used at every coupling time step.

4 RESULTS FOR AERO-THERMAL COUPLING

The aero-thermal coupling is carried out in an axisymmetric configuration.

The turbulence of the flame is processed by a RANS model (k-ω). The reacting fluid computation produces a cool central core flame (Figure 4), which differs from the experiment: cool unburnt gases are in contact with the material at the center without heat transfer to the plate. A heat flux peak is reached at the impingement point of the flame front (around $x = 5$ cm).

Foat *et al.* [2] experimentally showed that the transition between an envelope flame and a cool central core flame is a quasi-instantaneous process. These two flames can occur

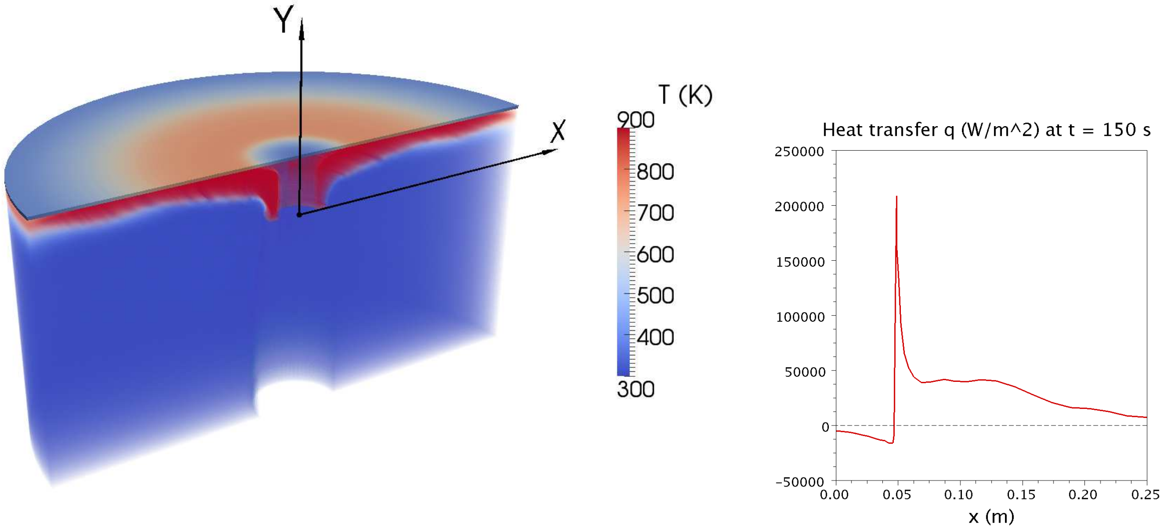

Figure 4: Temperature fields in fluid and solid and flux profile at the wall at $t = 150$ s.

at the same equivalence ratio (when the reactant mixture is fuel-rich, see Zhang & Bray [5]) depending on whether the fuel flow is increasing or decreasing. Indeed, once a cool central core has been formed it is difficult for atmospheric air to diffuse through the hot burnt gases zone, as the predominant direction of the flow is away from the stagnation zone.

As for the solid part, a radiative flux (plate emission) and a natural convective flux are set as thermal boundary conditions at the back of the plate. The external lateral side of the plate is an adiabatic wall (no heat transfer). The results in Figure 4 refer to the computation with a plate made of INCO600. In this aero-thermal computation, the deformation of the plate is not considered.

The coupling time step is $\Delta t_c = 30$ s in order to perform only ten couplings during the entire computation. The computation encountered a flame instability after $t = 150$ s.

All along the computation, the conductive heat transfer through the plate increases the temperature at the center of the plate. This implies that heat enters the boundary layer flow inside the cool central core of the flame from the plate. Moreover, the impingement point gets slowly closer to the center of the flame. Combined, these two phenomena may cause a change of the flame shape into an envelope as the one observed in the experiment, which might explain the instability of the aero-thermal calculation.

5 CONCLUSIONS

The flame-wall interaction problem has been studied on axisymmetric domains for aero-thermal coupling. As the procedure follows a logical step-by-step approach, the next computation will be the axisymmetric thermo-mechanical coupling. The influence of the flame will be reprensented by a heat transfer coefficient h associated to the adiabatic temperature at the front af the plate. Furthermore, creep generates plastification inside the plate. Indeed, a permanent deformation of the plate is experimentally observed after the flame extinction. This will also be taken into account in the mechanical model.

Extension to a full aero-thermo-mechanical coupling is currently underway in 3D sectorial domains and will be considered in future work.

Analogies between non-reacting jets and reacting jets (flames) with impingement were noted by Milson & Chigier [3]. In order to save computation time, the aero-thermo-mechanical algorithm will be first validated on an equivalent case of a hot jet, which produces a bell-shaped flux.

REFERENCES

[1] Baqué, B., Errera, M.-P. and Roos, A., Coupling procedure for modelling transient Conjugate Heat Transfer. *Proceedings of Coupled Problems* (2009).

[2] Foat, T., Yap, K.P. and Zhang, Y., The Visualization and Mapping of Turbulent Premixed Impinging Flames. *Combustion and Flame* (2001) **125**:839-851.

[3] Milson, A. and Chigier, N.A., Studies of Methane and Methane-Air Flames Impinging on a Cold Plate. *Combustion and Flame* (1973) **21**:295-305.

[4] Roinard, H., Micheli, F., Reulet, P. and Millan, P., Transferts thermiques lors de l'interaction flamme paroi. *Proceedings of Société Française de Thermique* (2007).

[5] Zhang, Y. and Bray, K.N.C., Characterization of Impinging Jet Flames. *Combustion and Flame* (1999) **116**:671-674.