Thermal fluid–structure interaction on re-entry vehicle TPS

S. Borreca[a,*], G. Tumino[b], P. Gaudenzi[a]

[a] *Dipartimento di Ingegneria Aerospaziale ed Astronautica, University of Rome 'La Sapienza', Via Eudossiana 18, 00184 Rome, Italy*
[b] *ESA-ESTEC, Keplerlaan 1, 2200 AG Noordwijk, The Netherlands*

Abstract

The present study focuses on the simulation of the aero-thermal interaction between the flow field surrounding a re-entry vehicle and its thermal protection system (TPS). An interface tool has been developed to couple computational fluid-dynamics (CFD) results and thermo-structural simulations in order to evaluate the actual thermal loads acting on the structure along the re-entry trajectory. In order to show the efficiency of the developed tool, this technique has been applied to a meaningful two-dimensional test case, using MSC-Nastran as thermo-structural code. It is remarkable that, even though the presented model is two-dimensional, the above interface procedure has been developed to be applied to both 2D and 3D cases.

Keywords: Thermal fluid–structure interaction; Multidisciplinary analysis; Heat transfer; Design; Coupling procedure; Interpolation

1. Introduction

Because of high-energy phenomena involved during re-entry, all the heat transfer mechanisms [1] play a fundamental role in a space vehicle TPS thermal balance. On one side there is a severe aerodynamic heating due to the very high re-entry velocity [2] and to possible chemical gas–surface interactions (superficial catalysis phenomena). On the other side there are conductive heat transfer into the structure, and radiative thermal exchange among surfaces and towards space. The former diffuses the internal energy into the structure, whereas the latter can have, at very high surface temperatures, a significant cooling effect re-emitting energy according to the Stefan-Boltzmann law. As a consequence, in order to correctly evaluate the TPS superficial and internal temperature distributions, it is necessary to take into account all the above heat transfer phenomena and their mutual interaction. In fact, radiation and conduction mechanisms, modifying the TPS internal energy distribution, influence the convective heat transfer between the structure and the surrounding flow field and can strongly reduce the vehicle surface temperature. Hence a space vehicle TPS design requires an accurate multidisciplinary analysis that takes into account thermal fluid–structure interaction phenomena in the TPS thermal balance.

Different tools are generally used to analyze aerodynamic heating and conduction and radiation heat transfer. Aero-thermal loads are evaluated by means of CFD codes that allow us to simulate the aero-thermodynamic flow field surrounding the vehicle during re-entry. They numerically solve the Navier-Stokes system of equations for chemically reacting gas mixtures, using generally a finite volume technique [3]. On the other hand, heat transfer into the structure and radiation mechanisms are often analyzed using thermo-structural codes that, by means of a Finite Elements (FE) approach, are able to perform thermo-elastic analysis, once the temperature distribution is computed.

This paper presents a procedure that allows us to carry out a TPS thermal analysis coupling CFD results and thermo-mechanical simulations, based on an interpolation procedure that allows the assignment of the surface temperature (or heat flux) distribution, computed by means of a CFD solver, as boundary conditions to an FE thermo-structural model. This approach presents the advantage that both the CFD and the thermo-structural codes can be chosen independently, as the most appropriate for each computational case.

Corresponding author. Tel.: +39 3285927522; Fax: +39 0644585670; E-mail: steborreca@libero.it

© 2005 Elsevier Ltd. All rights reserved.
Computational Fluid and Solid Mechanics 2005
K.J. Bathe (Editor)

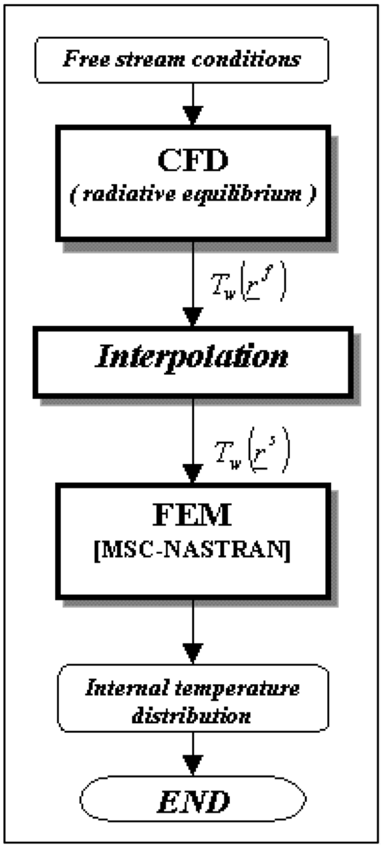

Fig. 1. Coupling procedure.

2. Coupling procedure

The problem to be solved is the steady-state thermal analysis of a continuum representative of the vehicle TPS, coupling aero-thermodynamic and thermo-structural simulations. The external flow field is solved using a CFD code (finite volume approach), and the thermodynamic field in the solid by means of Nastran (FEM). The procedure represented by the flow-chart in Fig. 1 allows the solution of the addressed problem under the assumption that the solid surface reaches the *radiative-equilibrium temperature* [4].

Firstly, the wall-temperature distribution is computed by means of a CFD simulation, assuming a local balance between convective and radiative heat fluxes (*radiative-equilibrium condition*):

$$-\lambda\left(\frac{\partial T}{\partial n}\right)_w = \sigma\varepsilon T_w^4 \qquad (1)$$

where T_w is the local surface temperature (*radiative equilibrium wall temperature*), λ is the gas thermal conductivity, ε is the body surface emissivity, and $\sigma = 5.67 \times 10^{-8}\ \mathrm{W/m^2K^4}$ is the Stefan-Boltzmann constant.

Second, through the developed interpolation procedure, the obtained radiative-equilibrium wall-temperature distribution is assigned as boundary condition to the thermo-mechanical model.

Third, the thermal analysis of the solid can be performed taking into account conductive heat transfer and re-radiation effects, and possible material anisotropies.

The obtained temperature distribution in the solid can be used to perform thermo-elastic analysis, in order to evaluate stresses induced in the structure by material thermal expansions.

2.1. Interpolation procedure

The core of the coupling procedure is the interpolation box (Fig. 1) that provides the interface between the two codes. Since the surface mesh is usually much finer than the one required in FE thermal analysis, a *nearest-neighbor* interpolation algorithm has been considered especially suitable. The selected interpolation criterion assigns to each FE node the value of temperature of the nearest CFD grid point.

The developed interface tool initially reads and records, from the CFD output file (supposed to be in Tecplot ASCII data-file format), the surface grid points and their corresponding temperatures and, from the Nastran input file, the coordinates of surface nodes of the FE model.

Let

$$\underline{r}_i^f \qquad i \in \{1, \ldots, N_f\}$$
$$\underline{r}_j^s \qquad j \in \{1, \ldots, N_s\}$$

be the CFD and FEM grid points positions with respect to a common reference frame. Moreover, let $f\left(\underline{r}_i^f\right)$ be a function of $\underline{r}_i^f$, such as wall temperature in our case.

According to the nearest neighbor interpolation

$$f\left(\underline{r}_j^s\right) = f\left(\underline{r}_{i_{\min}}^f\right), \qquad \forall j \in \{1, \ldots N_s\}$$

where

$$i_{\min}: \quad \left\|\underline{r}_j^s - \underline{r}_{i_{\min}}^f\right\|_2 = \min_{i \in \{1,\ldots N_f\}} \left\|\underline{r}_j^s - \underline{r}_i^f\right\|_2$$

Finally, the developed procedure automatically provides in output the Nastran TEMP entry that defines temperature at grid points.

2.2. Two-dimensional test case

As a test case, the massive ramp shown in Fig. 2 has been considered surrounded by a flow field at Mach = 5,

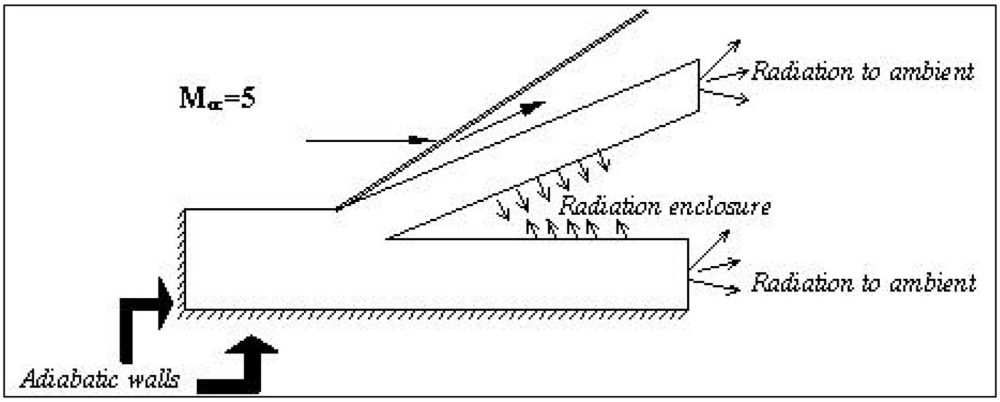

Fig. 2. Two-dimensional test case.

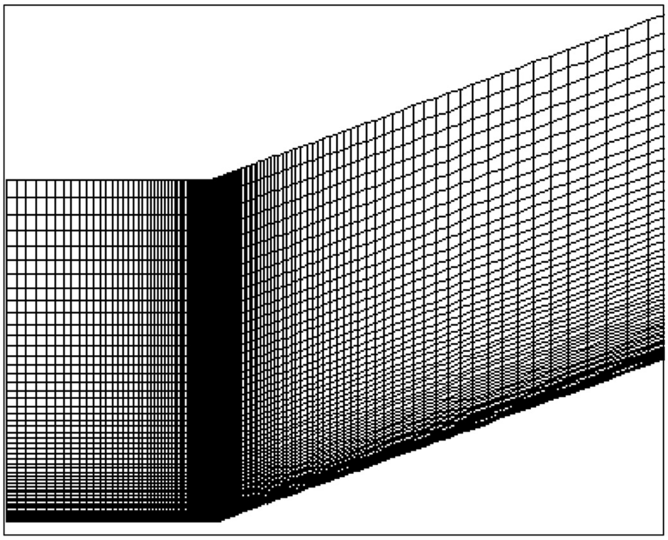

Fig. 3. CFD computational grid.

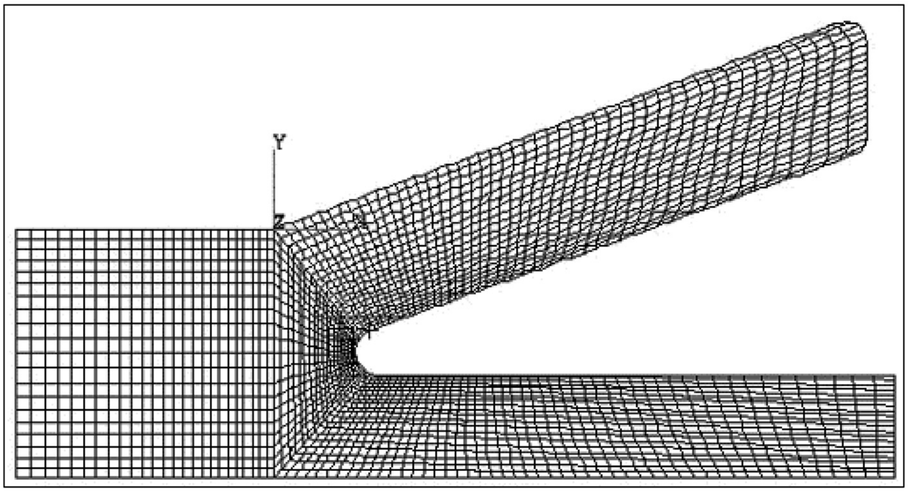

Fig. 4. Finite element model.

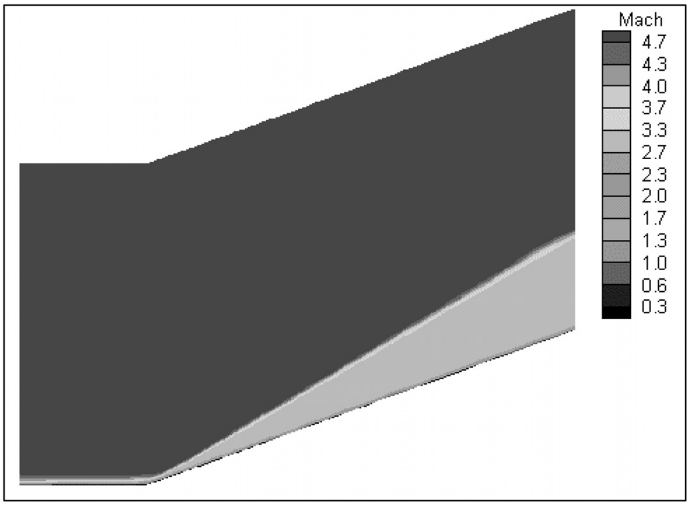

Fig. 5. Mach number flow field.

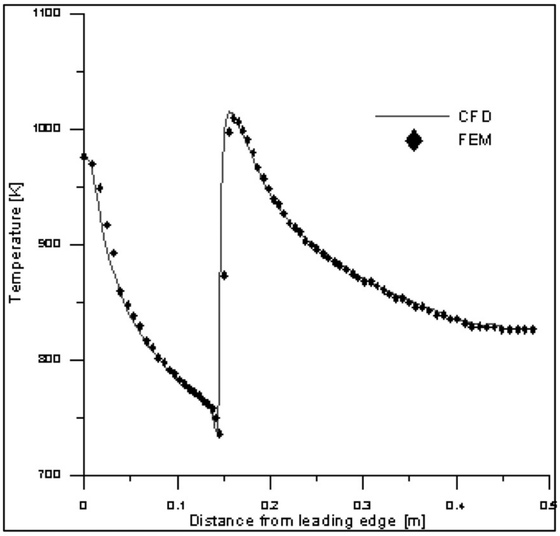

Fig. 6. Comparison between CFD and FEM temperature profiles.

$T = 300\,\text{K}$ and $p = 1\,\text{kPa}$. The ramp windside is supposed to reach the *radiative-equilibrium wall-temperature*, whereas the bottom and frontal surfaces have been considered adiabatic. Radiation heat transfer in the cavity between the ramp leeside and the base surface has been taken into account by means of the Nastran *radiation enclosure* boundary condition [5]. The remaining sides radiate to the external environment. The ramp material thermal conductivity is $k = 18\,\text{W/mK}$. CFD simulations have been performed by means of the commercial code FLUENT, using a 7200 cells computational grid (Fig. 3), with 120 body cells and 60 normal to the boundary, distributed with a 1.11 stretching factor.

In order to perform the thermal analysis of the solid, the FE model shown in Fig. 4 has been adopted, with 2900 CQUAD4 Nastran elements [6] and 75 nodes along the ramp windside.

Figure 5 shows the Mach number flow field: the shock wave corresponding to the body surface deflection is clearly visible.

The obtained radiative-equilibrium wall-temperature distribution has been assigned to the FE thermal model by means of the adopted interpolation procedure. Figure 6 compares the CFD temperature profile and the one obtained on the FE model boundary throughout interpolation: perfect matching of the two curves clearly demonstrates that the chosen nearest-neighbor interpolation algorithm is effective.

Finally, Fig. 7 shows the temperature distribution into the structure, obtained taking into account the radiation heat transfer into the enclosure and to the ambient ($T_{amb} = 300\,\text{K}$), as schematically shown in Fig. 2.

If needed, this internal temperature distribution can be used to perform thermo-elastic analysis once the material thermal expansion coefficient is provided.

3. Conclusions

The present study has highlighted the importance of multidisciplinary analysis in a re-entry vehicle TPS design, in order to take into account local heat transfer phenomena. In the absence of a software tool able to perform a fully coupled fluid–structure analysis, it is

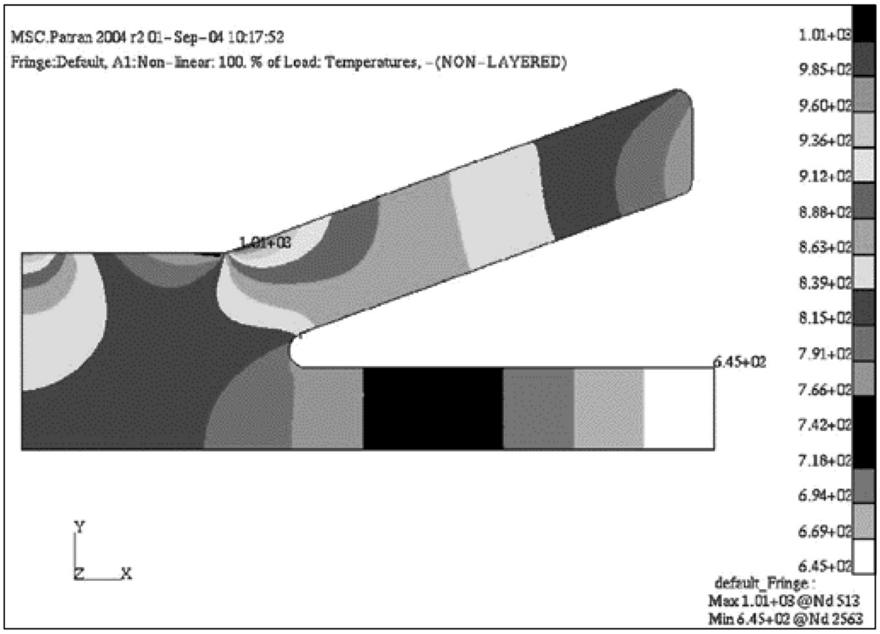

Fig. 7. Ramp internal temperature distribution.

necessary to develop appropriate interfaces between the two tools commonly used in the design of a re-entry vehicle, i.e. CFD and thermo-elastic codes. The coupling procedure presented in this paper, based on a *nearest neighbor* interpolation algorithm, can be considered a quick and effective tool to interface a generic CFD solver and MSC-Nastran (a widely used thermo-structural code). It can be used in combination with any kind of flow solver as long as Tecplot output (widely used in the CFD community) is provided and applies both structured and unstructured meshes.

References

[1] Holman JP. Heat Transfer. New York: McGraw-Hill International Series, 1996.
[2] Anderson JD. Hypersonic and High Temperature Gas Dynamics. Reston, VA: AIAA, 1989.
[3] Anderson JD. Introduction to Computational Fluid Dynamics. New York: McGraw-Hill, 1995.
[4] Kleb WL, Wood WA, Gnoffo PA, Alter SJ. Computational aeroheating prediction for X-34. J Spacecraft Rockets 1999;36(2).
[5] MSC-Nastran. Thermal Analysis User's Guide. St Ana, CA: MSC Corporation, 2004.
[6] MSC-Nastran. Theoretical Manual. Los Angeles, CA: MSC Corporation, 1976.