International Journal of Engineering, Mathematical and Physical Sciences
ISSN: 2517-9934
Vol:7, No:4, 2013

Development of a New CFD Multi-Coupling Tool Based on Immersed Boundary Method: toward SRM Analysis

Ho Phu TRAN and Frédéric PLOURDE

***Abstract*—**The ongoing effort to develop an in-house compressible solver with multi-disciplinary physics is presented in this paper. Basic compressible solver combined with IBM technique provides us an effective numerical tool able to tackle the physics phenomena and especially physic phenomena involved in Solid Rocket Motors (SRMs). Main principles are introduced step by step describing its implementation. This paper sheds light on the whole potentiality of our proposed numerical model and we strongly believe a way to introduce multi-physics mechanisms strongly coupled is opened to ablation in nozzle, fluid/structure interaction and burning propellant surface with time.

***Keywords*—**Compressible Flow, Immersed Boundary Method, Multi-disciplinary physics, Solid Rocket Motors.

I. Introduction

NUMERICAL tool in computational fluid dynamics has made impressive progress in recent decades. With given equations and appropriate numerical algorithms, CFD tool is able to provide accurate solution of external/internal flows. However, multi-physics modeling is still a considerable challenge. For example, considering solid-propellant rocket motors (SRMs), the internal flow field is very complex and multi-physics phenomena interact: turbulence, potential acoustic resonance, interaction between thermal inhibitor and the flow field as well as regression of the propellant during firing are naturally fully coupled. As far as we know, no mathematical modeling is yet able to depict all these phenomena together. Our goal is to develop a numerical tool able to consider multi-physic problem in an easy way. To do so, a basic solver combined with Immersed Boundary Method (IBM) is presented. The IBM technique is a clever way to couple mathematically numerical models that were not initially developed to such couplings. This paper describes how our model renders multi-physic coupling feasible to address propellant firing in time.

Frédéric PLOURDE is with ENSMA-CNRS, UPR3346, Institut Polytechnique de Poitiers (Pprime), France (phone: +33 6 10 15 06 07; e-mail: frederic.plourde@ensma.fr).

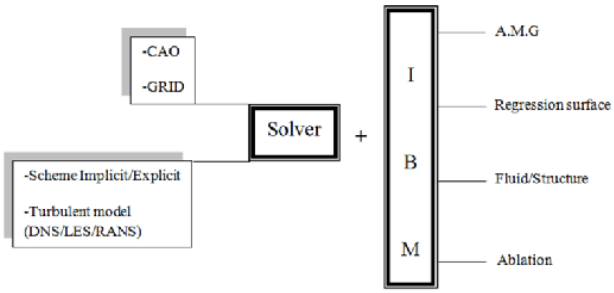

Fig. 1 Topology of multi-physics solver

II. Numerical Approach

A. Governing Equation

The flow field dynamics is governed by the viscous Navier–Stokes equations ensuring the mass, momentum and energy balances:

$$\begin{cases} \dfrac{\partial \rho}{\partial t} + \dfrac{\partial \rho u_j}{\partial x_j} = 0 \\ \dfrac{\partial \rho u_i}{\partial t} + \dfrac{\partial}{\partial x_j}\left(\rho u_i u_j\right) = -\dfrac{\partial p}{\partial x_j} + \rho g_i + \dfrac{\partial \sigma_{ij}}{\partial x_j} \\ \dfrac{\partial \rho E}{\partial t} + \dfrac{\partial}{\partial x_j}\left[\left(\rho E + p\right)u_j\right] = \rho g_i u_j + \dfrac{\partial}{\partial x_i}\left(\sigma_{ij} u_j\right) + \dfrac{\partial}{\partial x_j}\left(\kappa \dfrac{\partial T}{\partial x_j}\right) \end{cases} \quad (1)$$

where :

$$\sigma_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\mu \frac{\partial u_k}{\partial x_k}\delta_{ij} \quad (2)$$

$$\rho E = \rho H - p = \rho h - p + \frac{1}{2}\rho u_k u_k \quad (3)$$

and ρ is the gas density, p the thermodynamic pressure, ρu_i the momentum balance following the x_i direction and ρE the total energy. We assume fluid behaves like a Newtonian fluid following the perfect gas law:

$$p = \rho R T \quad (4)$$

While the dynamic viscosity is assumed to follow the Sutherland's law:

$$\mu(T) = \mu_{ref}\left(\frac{T}{T_{ref}}\right)^{3/2}\frac{T_{ref} + S}{T + S} \quad (5)$$

S being a reference temperature and κ correspond to the thermal conductivity.

$$\kappa = \frac{c_p \mu}{\Pr} \tag{6}$$

The turbulence model available in the code is RANS or LES but all configuration tested hereafter have been assumed to be 2D laminar.

B. Immersed Boundary Technique

Immersed boundary technique is based on "discrete forcing" method [1]. To take into the moving surface, we construct firstly a Cartesian mesh (from octree or 2^n tree mesh) and mesh points split into four different categories: (a) forcing points, which correspond to fluid points that have at least one neighboring point in the solid phase, (b) ghost cell points, that are similar the forcing point but in the solid side, (c) solid points and (d) the remaining points that correspond to fluid points (Fig. 2). For the forcing point and ghost-cell point, a linear interpolation is constructed with the form $\Phi = c_1 x + c_2 y + c_3 z + c_4 z$. Value of Φ is calculated depending the type of condition either Dirichlet condition or Neumann condition.

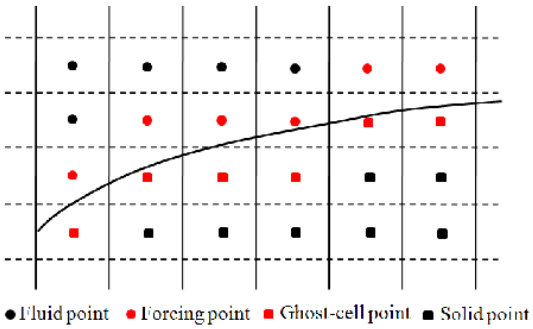

Fig. 2 Schematic of IBM method

The c_i coefficients are calculated according to values of neighbor points as an interpolation procedure. In our case, three neighbor points and one point on the "immersed boundary" are required to reach the first order interpolation. The c_i coefficients are estimated following:

$$[C] = [V]^{-1}[\Phi] \tag{7}$$

where

$$[\Phi] = \begin{bmatrix} \Phi_1 \\ \Phi_2 \\ \Phi_3 \\ \Phi_4 \end{bmatrix} \text{ and } [C] = \begin{bmatrix} c_1 \\ c_2 \\ c_3 \\ c_4 \end{bmatrix} \tag{8}$$

Matrix $[V]$ depend on the type of condition employed [2], we have:

Dirichlet condition:

$$[V] = \begin{bmatrix} x_1 & y_1 & z_1 & 1 \\ x_2 & y_2 & z_2 & 1 \\ x_3 & y_3 & z_3 & 1 \\ x_4 & y_4 & z_4 & 1 \end{bmatrix} \tag{9}$$

Neumann condition:

$$[V] = \begin{bmatrix} x_1 & y_1 & z_1 & 1 \\ x_2 & y_2 & z_2 & 1 \\ x_3 & y_3 & z_3 & 1 \\ n_x & n_y & n_z & 0 \end{bmatrix} \tag{10}$$

where $(n_x, n_y, n_z) = \vec{n}$ is normal vector at "Immersed boundary"

C. Solver and 2^n tree Automatic Grid Structure

2^n tree like organization is a very efficient way to automatically generate grid for complex geometries and reduce the number of mesh cells [3]. 2^n tree mesh is a part of the family of octree but the 2^n tree method is a more efficient one and more flexible too with a specific decrease of the number of unnecessary grid elements. For spatial scheme, a Riemann solver with ROE's scheme extended to second order using the MUSCL approach is applied for the convection flux discretization. To limit slope of gradients in case of shocks, a min-mod TVD limiter function is taken into account as well as the Harten's correction to avoid incorrect behavior of entropy parameter. For the viscous flux, a second accurate central difference scheme is used. For temporal scheme, explicit scheme with 2nd order Runge Kutta is applied despite implicit scheme with Van Leer construction is available.

D. Multi-Coupling Technique

IBM technique allows one to access to multi-coupling and is able to depict ablation problem (regression of surface) and interaction fluid/structure. To do so, the well-known regression propellant law i.e. the classical aP^n law [4] is applied. This law describes the regression speed of the propellant surface and displacement of the grid point is estimated locally from that regression speed in a given interval of time. From the IBM technique, pressure of the forcing point is calculated by an interpolation with Neumann condition dp/dn=0. As soon as the displacement is known, the new mesh is constructed and so on. For the fluid/structure interaction, coupling with Abaquus and Aster CAE are available (i.e. interaction between thermal protection and the internal flow field).

III. Numerical Results

A. Shock tube Problem

Our intention is to shed light on several multi-coupling simulation test cases in order to underline the potentially of the proposed tool. To present the main features of our solver, a

first shock tube test case give feedbacks on how accurate our solver may capture discontinuities. The domain length is equal to 10m and observation time is 3.9 ms. Number of grid elements is equal to 1000 grids. The left and right state (Fig. 3) are respectively u_L = 0 m/s, P_L=100000 Pa and T_L=248 K (i.e. high pressure region); and u_R = 0 m/s, P_R=1000 Pa et T_R=248 K (i.e. low pressure region).

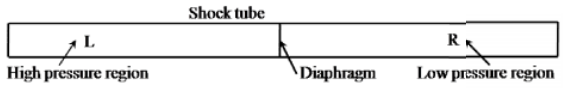

Fig. 3 Scheme of shock tube case with diaphragm in the center.

Results of our solver are similar to theoretical results (Fig. 4). It proves that our solver can capture the discontinuities (shock wave, of expansion, and discontinuity of contact).

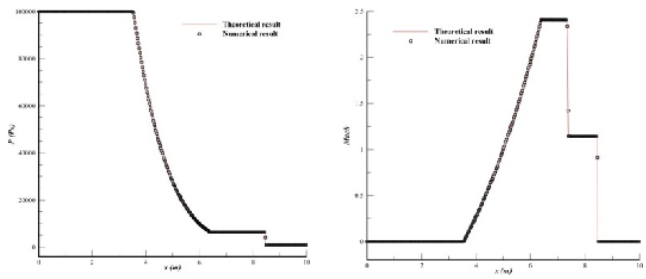

Fig. 4 Comparison of Pressure and Mach number between theoretical and numerical result

B. Divergence-Convergence Nozzle

The second configuration studied is also a very well-known case, i.e. the 2D transonic flow in a converging-diverging nozzle. The expected solution is a smooth distribution from subsonic to supersonic, i.e. with no shock. In case of bad numerical scheme especially linked to the use of the IBM technique (i.e. to present the boundary of the nozzle), the entropy condition is not satisfied and an expansion shock may be produced. To underline how IBM technique is suitable to such a complex flow field, the geometrical nozzle is fairly similar to [5]; the latter is geometrically is derived from the formula $y = \sqrt{(0.1 + x^2)/\pi}$ with x from - 0.5m to 0.5m (Fig. 5). In Fig. 6, the distribution of the Mach number is plotted and the fact that no expansion shock occurs underlines the correct aptitude of the proposed solver to depict such nozzle flow field; the Mach profile along the nozzle axis (Fig. 7) is smooth and continuously increases. No spurious expansion shock develops. The numerical result is a little different with theoretical result because of the effect 2D in our case but results are very encouraging.

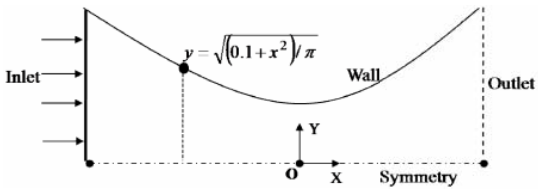

Fig. 5 Configuration and boundary condition of Nozzle test case

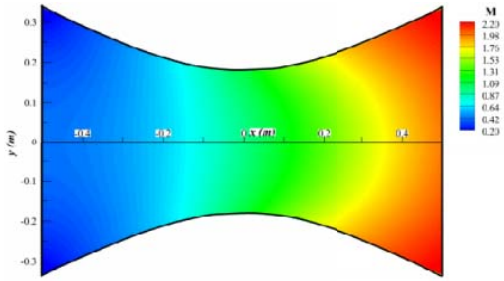

Fig. 6 Average Mach number in the Nozzle

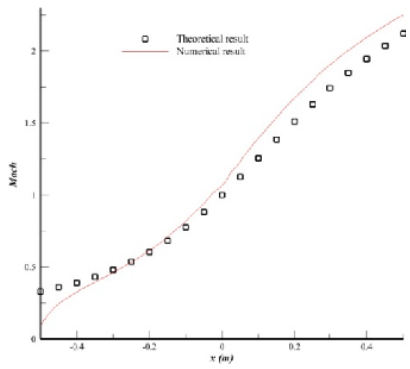

Fig. 7 Average Mach number on the wall

C. Supersonic Flow Past Circular Bump

A more complex configuration has been considered i.e. supersonic flow along a circular bump, the width of the channel being equal to the length of the bump, and the channel length is three times the length of the bump. For this test case, the thickness to chord ratio is 4% and the chosen configuration assimilates itself to a standard test case [6]. For the given uniform Mach number at the inlet, $M_{inlet}=1.4$, and for the used geometry with the choice of $L=100mm$ (Fig. 8), the flow is also supersonic at the outlet. For that reason, all variables are prescribed at inlet, all variables are extrapolated and at outlet, the network of shocks is captured (see Fig. 9) along the bump and downstream from its location and results obtained fairly fit with these obtained by Manna [6]. All these cases have been applied to fixed geometries and the 2^n tree mesh coupled with an IBM treatment is found to offer interesting outputs (see Fig. 10).

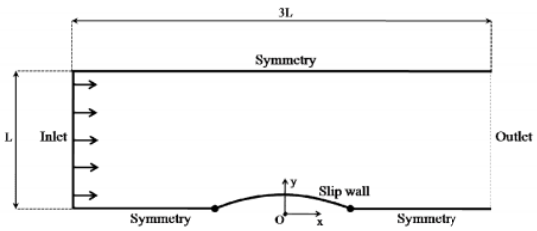

Fig. 8 Configuration and boundary condition of BUMP test case

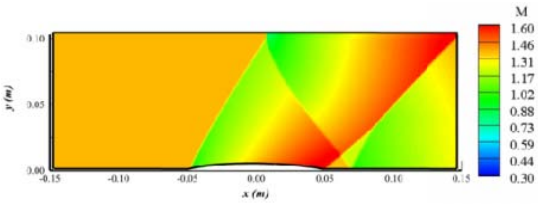

Fig. 9 Mach number distribution in the BUMP test case

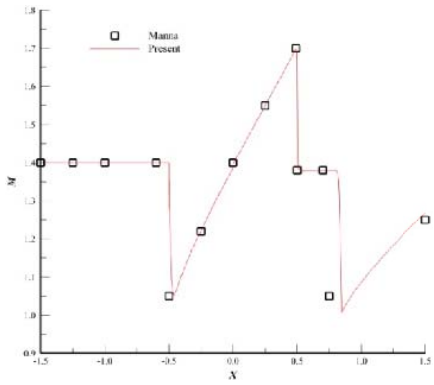

Fig. 10 Mach number distribution on the lower face of BUMP

D.Simple Regression Surface Case with IBM Technique

Finally a time moving geometry is carried out; the internal geometry of a SRM is considered and burning surface of the propellant is modeled via the aP^n law; the surface propellant regression is followed in time and this configuration allow us to exhibit the feasibility of our proposed CFD model. In this test case, one small domain with L x h= 100 mm x 20 mm with 1.5 mm of height of propellant (see Fig. 11) is simulated with the law aP^n. The input variable is chosen respectively a=0.69, n=0.66. The fixed pressure is chosen for this simulation with P=153500 Pa. The parameters of the propellant are propellant temperature equal 303K and propellant density equal 1855 kg/m^3. This test case is similar the Taylor test case. Firstly, in order to obtain the steady state, we simulate the flow with application IBM technique for aP^n law, it means that the regression is not considered. When the flow is established, the regression surface of propellant is activated. We call t_0 correspond the necessary time for established flow state (i.e. the initial time for the procedure of tracking the regression of propellant surface). We will track the change of burning surface step by step from t_0 to t_5 (i.e. the change of forcing and ghost-cell point). Fig. 12 shows the height of propellant from t_0 to t_5. Fig. 13 shows the change of role of the grid point in the domain of simulation when regression surface is counted.

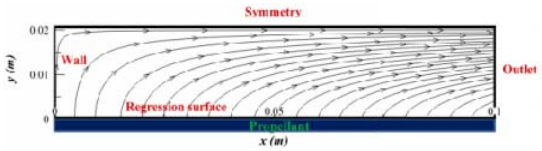

Fig. 11 Configuration and boundary condition

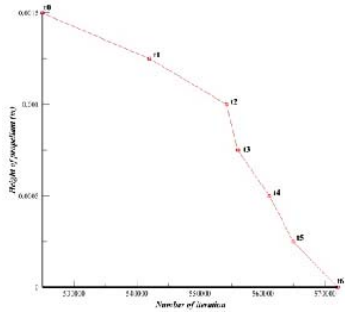

Fig. 12 Height of propellant following the time

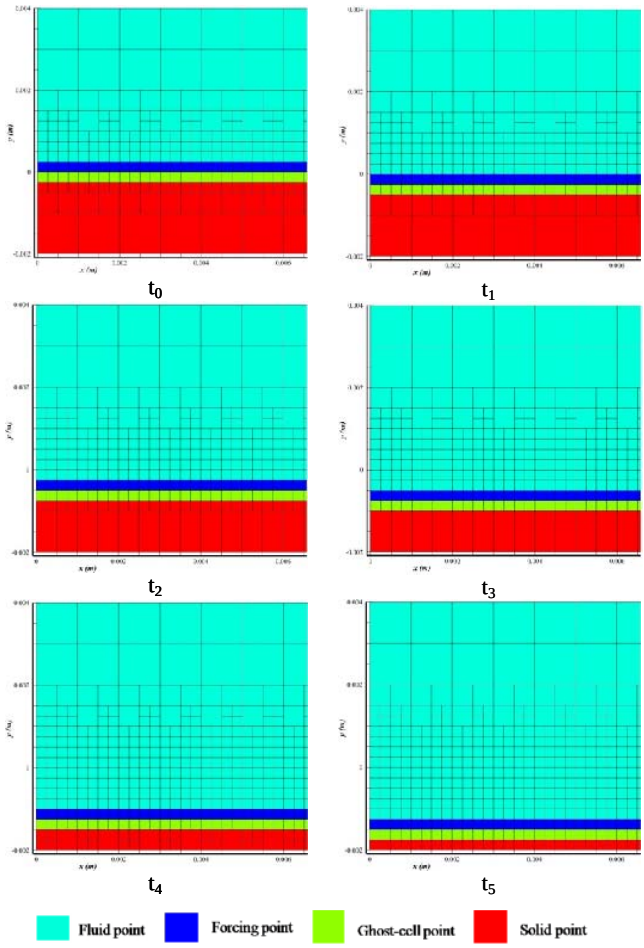

Fig. 13 Tracking of surface of regression

When the domain and regression velocity are small, the influence of regression is not significant for entire flow. However, these results, specially the tracking of forcing and ghost-cell point, prove us the capability of capture of the regression of surface with IBM technique.

IV. Conclusion

The proposed multi-coupling solver is found to successfully address complex aerodynamic phenomena and is suitable to tackle SRM test cases. The IBM technique facilitates the implementation of multi-physics problem. In a near future, simulation will be extended to external flow in fluid/structure interaction to tackle flutter mechanisms.

References

[1] H. Bandringa, "Immersed boundary methods," *MasterΓs thesis, University of Groningen*, vol. 9700, 2010.
[2] J. Yang and E. Balaras, "An embedded-boundary formulation for large-eddy simulation of turbulent flows interacting with moving boundaries," *Journal of Computational Physics*, vol. 215, no. 1, pp. 12–40, 2006.
[3] C. P. T. Groth, D. L. De Zeeuw, K. G. Powell, T. I. Gombosi, and Q. F. Stout, "A parallel solution-adaptive scheme for ideal magnetohydrodynamics," 1999.
[4] D. E. Kooker and C. W. Nelson, "Numerical Solution of Three Solid Propellant Combustion Models During a Gun Pressure Transient," Jan. 1977.
[5] I. Demirdžić, Ž. Lilek, and M. Perić, "A collocated finite volume method for predicting flows at all speeds," *International Journal for Numerical Methods in Fluids*, vol. 16, no. 12, pp. 1029–1050, 1993.
[6] M. Manna, *A Three Dimensional High Resolution Upwind Finite Volume Euler Solver*. Von Karman Institute for Fluid Dynamics, 1992.