Solving a compressible fluid flow by Asymptotic Numerical Method (ANM) with Moving Least Squares (MLS)

M. RAMMANE[a], S. MESMOUDI[a], A. TRI[b,c], B. BRAIKAT[a], N. DAMIL[a]

a. Laboratoire d'Ingénierie et Matériau (LIMAT), Faculté des Sciences Ben M'sik, Hassan II University of Casablanca, B.P 7955 Sidi Othman, Casablanca, Morocco.
b. Laboratoire de Mécanique, Faculté des Sciences Ain Chock, Hassan II University of Casablanca, Morocco.
c. Institut Supérieur des Etudes Maritimes (ISEM), Km 7 Route d'El Jadida, Casablanca, Morocco.

Abstract :

In this work, the Asymptotic Numerical Method (ANM) with a Moving Least Squares method (MLS) for the simulation of a compressible fluid flow is presented. The strong formulation of compressible viscous isothermal Navier-Stokes equations is the starting point. This proposed high order implicit algorithm is based on the implicit Euler scheme, a homotopy technique, a Taylor series, the MLS method and a continuation method. The MLS is a meshless collocation method and has attracted the attention of many researchers in recent years. Thanks to Taylor development, the nonlinear partial differential equations written under the strong formulation of a compressible fluid are transformed into a succession of linear differential equations with the same operator. This algorithm makes it possible to obtain the solution in a very along time interval with a less expensive CPU time. The results obtained by the proposed algorithm will be compared with those obtained using an explicit Runge-Kutta scheme and the Finite Difference Method (FDM) and those calculated by the Newton-Raphson method with MLS method also. The efficiency of this algorithm is tested on a standard benchmark of fluid mechanics such as the lid-driven cavity problem.

Keywords : Moving Least Squares, Asymptotic Numerical Method, nonlinear computation, Compressible fluid flow.

1 Introduction

The compressible fluid flows takes a large part of fluid mechanics and find their applications in many fields such as aeronautics,the hydraulics of free flows and the propulsion of various machines. These fluid flows are described by algebraic equations which has been the subject of several investigations to elaborate an efficient algorithm. In their generality, these equations are relatively non-linear and complex thus give rise to resolutions that make using several numerical methods and important computation times. The use of mathematical models for the simulation of such phenomena has become essential as a means of prediction. The need for a software robust able to treat this type of problem led to the development of numerical methods able to handle the different difficulties applied on different formulations [1, 2]. Hence, the incremental–iterative as Newton–Raphson method was one of the most useful ways for solving the nonlinear problem depending on a single parameter. It can be coupled with spatio-temporal discretization methods such as the implicit schemes with the Finite Difference Method (FDM) or Finite Element Method (FEM) or meshless methods.

The aim of this work is to implement a high order implicit algorithm to simulate the compressible viscous isothermal Navier-Stokes equations. The numerical solution of the Navier-Stokes non linear Partial Differential Equations (PDEs) is a very challenging and demanding problem. In this case, we adopt an efficient algorithm for solving the nonlinear problems numerically. This algorithm is based on the following steps : the Euler implicit scheme for the temporal discretization, the MLS method for the spatial discretization, the Taylor series, the homotopy technique and the continuation procedure. The homotopy transformation consists to introduce an arbitrary parameter "a" and an invertible pre-conditioner $[K^{*}]$ by modifying the problem to be solved [3, 4, 5]. When the homotopy parameter is zero, we get a linear problem easy to solve and when it is equal to one, we find the initial problem. The boundary conditions are treated by the collocation method [4]. This algorithm has been tested successfully on non linear problems evolving in time and space [4, 5]. To demonstrate the effectiveness and the robustness of the implicit high-order solver in comparison with the explicit Runge-Kutta scheme using FDM and the Newton-Raphson with MLS solvers, the example of a compressible fluid flow in a lid-driven cavity is studied.

2 Compressible viscous fluid model

The fluid medium constitutes a particular case of continuous medium for which a state of rest cannot be maintained in the presence of non-zero shear stresses unlike solids, which remain in a state of rest under shear forces. In this study, it is assumed that the fluid is compressible at constant temperature and that the system is closed by limiting itself to the first three conservation laws which are expressed as follows :

$$\left\{\begin{array}{lcl} \frac{\partial\rho}{\partial t}+\nabla.\beta & = & 0 \\ \frac{\partial\beta}{\partial t}+\nabla(\beta.V) & = & \nabla\tau-\nabla p \\ p & = & C_0\rho \\ \tau & = & \mu({}^t\nabla V+\nabla V)-\frac{2\mu}{3}(\nabla V)I \\ \beta & = & \rho V \end{array}\right. \tag{1}$$

where ρ is the density, $V = {}^t < u, v >$ is the velocity field, p is the hydrostatic pressure, τ is the viscous stress tensor and C_0 is a constant defined by $C_0 = b^2$ with b is the speed of sound. We combine the unknowns ρ, u and v into a single vector $X_m = {}^t\!< \rho, u, v >$ to write the previous equation in the following compact form :

$$M\dot{X}_m + L(X_m) + Q(X_m, X_m) = 0 \tag{2}$$

For the boundary conditions on the interface of the contact fluid/solid, the velocity of the particles fluid must be equal the velocity of the solid wall. For the mass density boundary conditions are obtained by the scalar product of the moment equation with the normal $\{n\} = {}^t < n_x, n_y >$ of the solid wall. This boundary conditions are expressed as :

$$\begin{cases} u & = & u_{wall} \\ \\ v & = & v_{wall} \\ \\ C_0 \nabla\rho.n & = & (\nabla\tau - \frac{\partial\beta}{\partial t} - \nabla(\beta.V)).n \end{cases} \tag{3}$$

2.1 Time discretization and Moving Least Squares

Firstly, to introduce a tomporelle discretization using the implicit Euler scheme, one considers a change of variables as follows :

$$X_m^{n+1} = X_m^n + \delta X_m \tag{4}$$

where X_m^n represents the value of the unknown X_m at time $t^n = n\delta t$ with δt is the time step. The nonlinear problem verified by the new unknown is written in the following form :

$$M\delta X_m + \delta t\mathcal{L}^k(\delta X_m) + \delta t Q(\delta X_m, \delta X_m) + \delta t\mathcal{F}^k(X_m^n) = 0 \tag{5}$$

where $\mathcal{L}^k(\delta X_m)$ and $\mathcal{F}^k(X_m^n)$ are written such as :

$$\begin{cases} \mathcal{L}^k(\bullet) & = & L(\bullet) + Q(X_m^n, \bullet) + Q(\bullet, X_m^n) \\ \\ \mathcal{F}^k(\bullet) & = & L(\bullet) + Q(\bullet, \bullet) \end{cases} \tag{6}$$

The second step of the proposed algorithm consists in introducing a MLS type approximation for spatial discretization at any point M of the studied fluid domain. The MLS approximation was devised by mathematicians in data fitting and surface construction [9]. It can be categorized as a method of series representation of functions. This approximation makes permits to write the unknown at point $M(x, y)$ by means of a set of situated points in a sub-domain called the support of the considered point. This approximation in considered point $M(x, y)$ is defined by :

$$\delta X_m = [\Phi]\{\delta X\} \tag{7}$$

where $[\Phi]$ is a matrix that contains the shape functions computed in each neighbourhood of point $M(x, y)$ and $\{\delta X\}$ is the nodal field variable at all neighbourhood nodes of point $M(x, y)$. Taking into account of this approximation and after substitution and assembly techniques, the problem (5) is written in the following condensed form :

$$[K_T(\{X^n\})]\{\delta X\} + \{F_q(\{\delta X\})\} = \{F(\{X^n\})\} \tag{8}$$

where $[K_T(\{X^n\})] = [K_T^n]$, $\{F(\{X^n\})\} = \{F^n\}$ are the tangent matrix and a right hand side depending on the solution $\{X^n\}$ at the previous instant $t^n = n\delta t$ and $\{F_q(\{\delta X\})\}$ is a quadratic form.

2.2 Homotopy transformation

In order to avoid the inversion of the tangent matrix at each time step, a homotopy transformation is adopted [3, 4, 5]. Its consists to introduce an arbitrary invertible matrix $[K^*]$ and an arbitrary parameter "a" in the following form :

$$[K^*]\{\delta\chi\} + a([K_T(\{X^n\})] - [K^*])\{\delta\chi\} + a\{F_q(\{\delta\chi\},\{\delta\chi\})\} \quad = \quad a\{F(\{X^n\})\} \tag{9}$$

It allows recovering continuously the original system ($\{\delta X\} = \{\delta\chi\}$) when $a = 1$ and the easier system to solve when $a = 0$ ($\{\delta\chi\} = \{0\}$).

2.3 Taylor series expansion

To solve the nonlinear dynamic problem (9), we seek the solution in the form of a truncated series expansion at order k [6] with respect to homotopy parameter "a" as follow :

$$\begin{cases} \{\delta\chi\} &= a\{\delta\chi_1\} + a^2\{\delta\chi_1\} + ... + a^k\{\delta\chi_k\} \\ \{\delta X\} &= \lim\limits_{a\to 1} \{\delta\chi\} \end{cases} \tag{10}$$

Taking into account of the Taylor series expansion (10), we obtain the following recurrent sequence linear problems easy to solve at each order :

$$\begin{array}{lrcl} \textit{Order } i = 1 : & [K^*]\{\delta\chi_1\} &=& \{F^n\} \\ \textit{Order } 2 \leq i \leq k : & [K^*]\{\delta\chi_i\} &=& ([K^*] - [K_T^n])\{\delta\chi_{i-1}\} + \{F_i^{nl}\} \end{array} \tag{11}$$

2.4 Continuation technique

The basic principle of the continuation is to determine the path by a succession of high order power series expansions with respect to a well chosen path parameter [6, 7]. The maximal value for this path parameter "a" is defined by :

$$a_{max} = \left(\epsilon \frac{||\{\delta\chi_1\}||}{||\{\delta\chi_k\}||}\right)^{\frac{1}{k-1}} \tag{12}$$

where ϵ is a tolerance parameter and $||.||$ represents a given norm. When $a \to 1$ we recover the solution of the problem (8). The solution of the initial problem (2) at time $t^{n+1} = (n+1)\delta t$ is obtained by $\{X^{n+1}\} = \{X^n\} + \{\delta\chi(a_{max} \geq 1)\}$

3 Numerical application and discussion

The lid-driven cavity problem is a standard benchmark in computational fluid mechanics. The simulation domain consists of a square region confined by solid walls of length $L = 1$. The top wall is moving at a constant horizontal speed $v_0 = 10$, while the others are fixed (see figure 1). In this study, the fluid have a dynamic viscosity $\mu = 0.1$, an initial mass density $\rho_0 = 1$, this leads to a Reynolds number $Re = 100$. The flow is considered subsonic with a Mach number $Ma = 0.1$ which means that the speed of sound $b = 100$ ($b = \frac{v_0}{Ma}$). The numerical parameters used in this work are : the number of nodes in the mesh is 3249 nodes i.e 9747 degrees of freedom, the length of support domain is $h = 0.0452$ and the tolerance parameter is taken $\epsilon = 10^{-6}$ for the implicit algorithms (ANM with MLS and Newton-Raphson with MLS) with a time step $\delta t = 10^{-2}s$. While, for the explicit algorithm (FDM explicit R-K), the time step is defined by the CFL-condition [8, 10] $\left(\Delta t < min\left(\frac{h}{a+\sqrt{u^2+v^2}}, \frac{\rho h^2}{4\mu}\right)\right)$, in this case $\delta t = 10^{-4}s$.

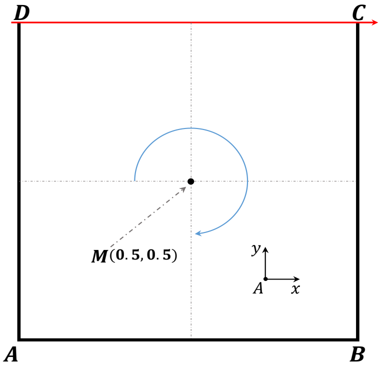


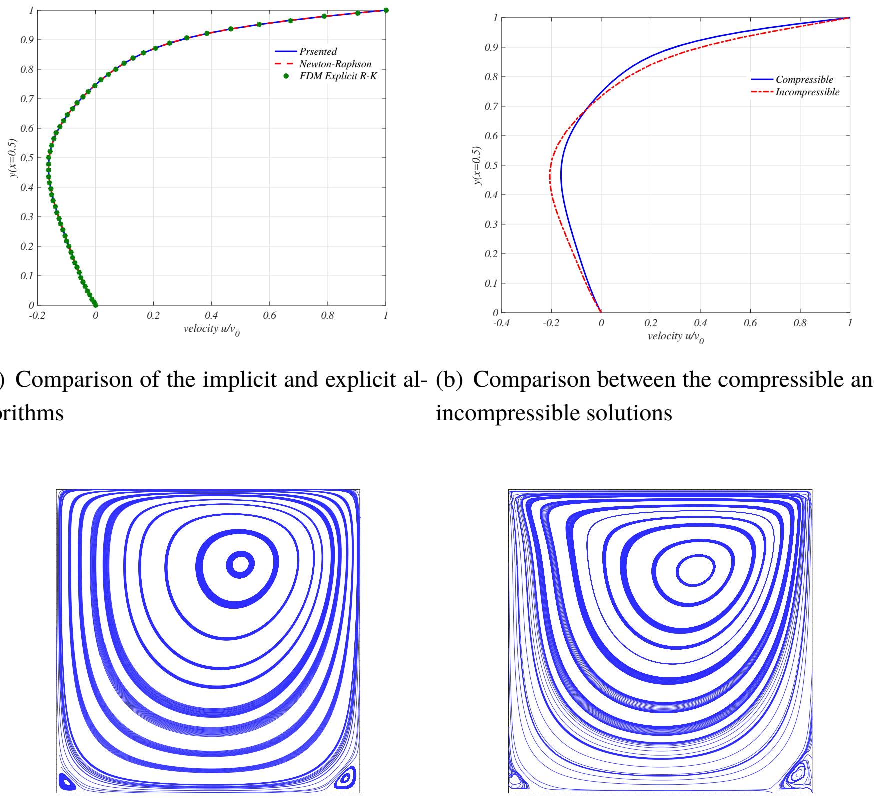
FIGURE 1 – Domain and boundary conditions

The table 1 presents the number of matrix decompositions and the time CPU (minute) of the used algorithm, the Newton-Raphson solver and the FDM using explicit Rang-Kutta. According to this table, the ANM with MLS algorithm requires a very small number of matrix inversions and a time CPU compared to the explicit and Newton-Raphson solvers. The solution is obtained by the presented algorithm from a truncation order $k \geq 8$, indeed when this last increases the Nb. of decompositions decreases, while the time CPU increases with 5 to 10 minutes from the first order $k = 8$. Note that, all numerical computations are achieved until the time $t = 2s$.

Algorithms	order	Nb. of decompositions	CPU time (min)
ANM with MLS	$k = 8$	4	21
	$k = 12$	2	27
	$k = 16$	1	31
Newton-Raphson		979	398
FDM Explicit R-K		20000	1290

TABLE 1 – Number of matrix inversions and CPU time in minute using $\epsilon = 10^{-6}$ for the implicit algorithms

In the figure 2, we represent a comparison of the x-component of velocity u/v_0 versus the space variable $y(x = 0.5)$ and the profile solution between the compressible and incompressible cases. Figure 2a shows that the obtained results are in good agreement with those computed by the Newton-Raphson and the explicit algorithms. While from the figures 2 b,c and d, we observe the difference of velocity and streamlines profiles between a viscous gas (viscous compressible fluid) and viscous liquid (viscous incompressible fluid).



(a) Comparison of the implicit and explicit algorithms (b) Comparison between the compressible and incompressible solutions

(c) Streamlines of the compressible flow (d) Streamlines of the incompressible flow

FIGURE 2 – x-component of velocity versus space variable y at $x = 0.5$

In figure 3, the mass density versus space variable $y(x = 0.5)$ for a Mach number $Ma = 0.1$, $Ma = 0.2$,$Ma = 0.3$ and $Ma = 0.4$ are plotted. It shows that the change in the mass density increases when the Mach number increases.

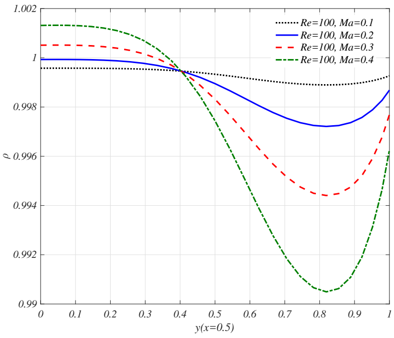


FIGURE 3 – Evolution of mass density at the line $y(x = 0.5)$ for several Mach numbers

4 Conclusion

A high-order implicit algorithm has been presented in order to propose an efficient tool for the simulation of compressible fluid flows. The introducing of a homotopy technique made it possible to reduce cost time in terms of the inversions number of the tangent matrix in order to obtain the solution in a time interval. The effectiveness in flows simulation is tested on a viscous compressible isothermal flow in a lid-driven cavity with a comparison of the obtained result with those computed by the Newton-Raphson algorithm and a explicit Range-Kutta scheme using a Finite Difference Method.

Références

[1] B. Thornber, M. Groom, D. Youngs, A five-equation model for the simulation of miscible and viscous compressible fluids, Journal of Computational Physics, 372, 2018, pp :256–280.

[2] H. Riahi, M. Meldi, J. Favier, E. Serre, E. Goncalves, A pressure-corrected Immersed Boundary Method for the numerical simulation of compressible flows, Journal of Computational Physics, 374, 2018, pp :361–383.

[3] M. Jamal, B. Braikat, S. Boutmir, N. Damil and M. Potier-Ferry. A high order implicit algorithm for solving instationary non-linear problems. Computational Mechanics, 28(5), 2002, pp :375-380.

[4] A. Timesli, B. Braikat, H. Lahmam and H. Zahrouni. A new algorithm based on moving least square method to simulate material mixing in friction stir welding. Engineering Analysis with Boundary Elements, 50, 2015, pp :372-380.

[5] S. Mesmoudi, A. Timesli, B. Braikat, H. Lahmam and H. Zahrouni. A 2D mechanical–thermal coupled model to simulate material mixing observed in friction stir welding process. Engineering with Computers 33(4), 2017, pp :885-895.

[6] B. Cochelin, N. Damil & M. Potier-Ferry. Méthode asymptotique numérique . Hermes Lavoissier, 2007, p. 297.

[7] B. Cochelin. A path-following technique via an asymptotic-numerical method. Computers & structures, 53(5), 1994, pp :1181-1192.

[8] CL. Felter, JH Walther and C. Henriksen. Moving least squares simulation of free surface flows. Computers & Fluids, 91, 2014, pp :47-56.

[9] P. Lancaster and K. Salkauskas. Surfaces generated by moving least squares methods. Mathematics of Computation, 37(155), 1981, pp :141-158.

[10] BAO, Weizhu et JIN, Shi. High-order I-stable centered difference schemes for viscous compressible flows. Journal of Computational Mathematics, 2003, p. 101-112.