ECCOMAS Congress 2016
VII European Congress on Computational Methods in Applied Sciences and Engineering
M. Papadrakakis, V. Papadopoulos, G. Stefanou, V. Plevris (eds.)
Crete Island, Greece, 5–10 June 2016

A LAGRANGIAN FINITE ELEMENT METHOD FOR THE SIMULATION OF 3D COMPRESSIBLE FLOWS

M. Cremonesi[1], **and A. Frangi**[1]

[1]Department of Civil and Environmental Engineering
Politecnico di Milano
Piazza Leonardo da Vinci, 32, 20133, Milano, Italy
e-mail: {massimiliaano.cremonesi,attilio.frangi}@polimi.it

Keywords: compressible flow, Lagrangian approach, PFEM.

Abstract. *The numerical solution of compressible fluid flows is of paramount importance in many industrial and engineering applications. Compared to the classical fluid dynamics, the introduction of the fluid compressibility changes the formulation of the problem and consequently its computational treatment.*

Among the possible numerical solutions of compressible flow problems, the finite element method has always been privileged. However, the standard Eulerian approaches with fixed domain are not particularly suited to represent the strong shock waves and the significant movement of the external boundaries. On the contrary, in problems characterized by evolving surfaces, Lagrangian approaches can be very effective.

The governing equations of compressible flow problems are mass, momentum and energy conservation. These equations are discretized in the spirit of the Lagrangian Particle Finite Element Method (PFEM). The strong distortions of the mesh, typical of the Lagrangian approaches, are managed with a continuous remeshing of the computational domain. The nodal unknowns are velocities, density and internal energy. To fully exploit the potential of continuous remeshing, only nodal variables are stored and consequently only linear interpolation are used. In addition, an artificial viscosity has been introduced to stabilize the formation and propagation of shock waves. Finally, explicit time integration of the governing equations enables a highly efficient solution of the discretized problem.

The proposed approach has been validated against typical benchmarks of gas dynamics in the presence of strong shock waves. A very good agreement has been shown in all the tests proving the excellent accuracy and versatility of the proposed method.

1 INTRODUCTION

The solution of the equations of high-speed compressible fluid has a wide range of engineering and physics applications. Due to the presence of shocks, viscous boundary layers and mixing of different constituents the numerical solution of these equations can be very complex.

Different approaches can be used for the solution of hydrodynamics equations: Eulerian [3, 2], Lagrangian [14, 7] or ALE [1, 16]. In this work, a Lagrangian finite element approach is exploited to describe strong evolving material interfaces and shocks propagations.

The differential problem has been solve with a Particle Finite Element Method [11, 5]. As expected with a full Lagrangian approach the mesh can deteriorate in time, consequently, to guarantee a good quality of the mesh, a remeshing technique is proposed. Morevover, an explicit time integration of the governing equations is proposed to solve efficiently the discretized problem. This paper is structured as follows. In section 2 the equations of motion for a compressible fluid are presented. Space and time discretizations are introduced in sections 3.1 and 3.2 respectively. Section 4 is devoted to the mesh update techniques while 5 presents the numerical results.

2 GOVERNING EQUATIONS

A compressible fluid in an evolving domain $\Omega(t)$ is considered. In the Lagrangian description, the particle position $\mathbf{x}$ at time t can be expressed as a function of velocity field $\mathbf{u} = \mathbf{u}(\mathbf{x}, t)$ through:

$$\mathbf{u} = \frac{\mathrm{d}\mathbf{x}}{\mathrm{d}t} \tag{1}$$

In this framework, balance equations can be written as:

$$\text{mass conservation:} \qquad \frac{1}{\rho}\frac{\mathrm{d}\rho}{\mathrm{d}t} = -\nabla\cdot\mathbf{u} \quad \text{in } \Omega(t)\times[0,T] \tag{2}$$

$$\text{momentum conservation:} \qquad \rho\frac{\mathrm{d}\mathbf{u}}{\mathrm{d}t} = \nabla\cdot\boldsymbol{\sigma} \quad \text{in } \Omega(t)\times[0,T] \tag{3}$$

$$\text{energy conservation:} \qquad \rho\frac{\mathrm{d}e}{\mathrm{d}t} = \boldsymbol{\sigma} : \nabla\mathbf{u} \quad \text{in } \Omega(t)\times[0,T] \tag{4}$$

where $\rho = \rho(\mathbf{x}, t)$ is the density, $\boldsymbol{\sigma} = \boldsymbol{\sigma}(\mathbf{x}, t)$ the Cauchy stress tensor and $e = e(\mathbf{x}, t)$ the internal energy per unit of mass. In many compressible gas-dynamics applications, the viscosity in very low and consequently shear terms in the Cauchy stress tensor can be neglected, leading to:

$$\boldsymbol{\sigma} = -p\mathbf{I} \tag{5}$$

where $p = p(\mathbf{x}, t)$ is the pressure field. Following this assumption, (2)-(4) reduce to the standard compressible Euler equations.

An equation of state for polytropic ideal gas is considered, linking pressure to density and internal energy through:

$$p = (\gamma - 1)\rho e \tag{6}$$

where $\gamma > 1$ is a constant adiabatic index.

3 DISCRETIZATIONS

3.1 Space discretization

To discretize in space the equations (2)-(4) a standard Galerkin approach has been followed. Linear continuous finite element are used to discretize both kinematic variables (velocities) and thermodynamic variables (pressure,density and internal energy). Introducing $\mathbf{U}$ the vector of the nodal values of the velocity and $\mathbf{E}$ the nodal values of the internal energy, the semidiscretized form of momentum and energy conservation reads:

$$\mathbf{M}_u \frac{d\mathbf{U}}{dt} = -\mathbf{F}_u \tag{7}$$

$$\mathbf{M}_e \frac{d\mathbf{E}}{dt} = \mathbf{F}_e \tag{8}$$

where $\mathbf{M}_u$ and $\mathbf{M}_e$ are mass matrices while $\mathbf{F}_u$ and $\mathbf{F}_e$ are the vectors of internal forces.

Mass conservation (2) is discretized starting from the strong form [14, 8]:

$$\rho(\mathbf{x}, t) J(\mathbf{x}, t) = \rho(\mathbf{x}, 0) \tag{9}$$

where $J(\mathbf{x}, t) = \det \mathbf{J}(\mathbf{x}, t)$ is determinant of the deformation Jacobian. Introducing the finite element approximation, the discretized problem reads:

$$\mathbf{M}_\rho(t)\mathbf{R} = \mathbf{R}_0 \tag{10}$$

where vector $\mathbf{R}$ contains the unknown nodal density. The equation of state (6) can be then used pointwise in every node of the mesh to directly to find the pressure field:

$$\mathbf{P} = (\gamma - 1)\,\mathbf{R}\mathbf{E} \tag{11}$$

3.2 Time discretization

Introducing a forward Euler integration scheme to discretize the time derivatives, the full discretized system writes:

$$\mathbf{M}_u \mathbf{U}^{n+1} = \mathbf{M}_u \mathbf{U}^n + \Delta t \mathbf{F}_u(\boldsymbol{\sigma}^n, \mathbf{U}^n) \tag{12}$$

$$\mathbf{M}_e \mathbf{E}^{n+1} = \mathbf{M}_e \mathbf{E}^n - \Delta t \mathbf{F}_e(\boldsymbol{\sigma}^n, \mathbf{U}^{n+1/2}) \tag{13}$$

$$\mathbf{M}_\rho^{n+1} \mathbf{R}^{n+1} = \mathbf{R}^0 \tag{14}$$

where $\mathbf{U}^{n+1/2} = \frac{1}{2}(\mathbf{U}^{n+1} + \mathbf{U}^n)$ is used to preserve total energy from time t^n to time t^{n+1} [7, 8]. The pressure unknowns is computed directly by equation of state:

$$\mathbf{P}^{n+1} = (\gamma - 1)\,\mathbf{R}^{n+1}\mathbf{E}^{n+1} \tag{15}$$

The system of equations (12)-(14) is particolarly suited for the fully explicit solution. If mass lumping is introduced velocity and energy can be evaluated node by node without solving linear systems. To respect the CFL condition, an adaptive time step is defined as:

$$\Delta T = \alpha \min_e \left(\frac{h_e}{\sqrt{\gamma_e\, p_e / \rho_e}} \right) \tag{16}$$

where h_e is a characteristic length of an element and α is a safety parameter [15, 8].

3.3 Artificial viscosity

To follow the shocks propagation, a tensor artificial viscosity has been introduced in the equations. A viscous term, activated only in the shock regions, has been added to the Cauchy stress tensor (5):

$$\boldsymbol{\sigma} = -p\mathbf{I} + \boldsymbol{\sigma}_a \tag{17}$$

where $\boldsymbol{\sigma}_a$ is defined as:

$$\boldsymbol{\sigma}_a = \mu_e \frac{1}{2}(\nabla \mathbf{u} + \nabla \mathbf{u}^T) \tag{18}$$

and μ_e is a viscous coefficient defined on every element e of the mesh as:

$$\mu_e = \psi_0 \psi_1 \rho h_e (\frac{2}{3} h_e |C_e| + \frac{1}{4} \psi_2 c_e) \tag{19}$$

where c_e is the speed of sound a C_e a measure of the compressibility. The stabilization is activated only in compression region through the compressibility switch ψ_1. The switch ψ_2 is a measures the vorticity-compressibility ratio while ψ_0 represents a smoothness detector. Details on the artificial viscosity can be found in [12].

4 NUMERICAL TECNIQUE

The Lagrangian Particle Finite Element Method is here used to solved the system of equations (12)-(14). The PFEM was orginally developed to solve problems involving free surfaces fluid flows [9, 13, 4] and fluid-structure interaction [10, 6]. The method is here revisited and applied to the simulation of compressible problems.

Due to the Lagrangian nature of the equations, mesh nodes are moved following the fluid velocity leading to possible mesh distortion. An index of the element distortion is used to check whether the mesh should be regenerated or not. When a new mesh should be created aN efficient Delaunay tesselletion technique is used to redefine the nodal connectivity starting from the current node position. The solution scheme is sketched in 1.

Algorithm 1 Solution scheme with remeshing

for $n = 1, ..., N_{steps}$ **do**
  check mesh distortion
  if mesh too distorted **then**
    Generate new mesh with Delaunay tessellation
    Compute $\mathbf{M}_u$, $\mathbf{M}_e$ and $\mathbf{R}^0$
  end if
  compute velocity $\mathbf{U}^{n+1}$ from eq.(12)
  update position $\mathbf{x}^{n+1} = \mathbf{x}^n + \Delta t\, \mathbf{U}^{n+1}$
  compute density $\mathbf{R}^{n+1}$ through eq.(14)
  compute internal energy $\mathbf{E}^{n+1}$ from eq.(13)
  compute pressure $\mathbf{P}^{n+1}$ from eq.(15)
  estimate the new stable Δt from eq. (16)
end for

5 NUMERICAL EXAMPLE

5.1 Sod shock tube

In the simple domain defined in figure 1, two gases are separated by a discontinuity in the middle. Density, pressure and velocity are initialized as reported in table 1. Internal energy is

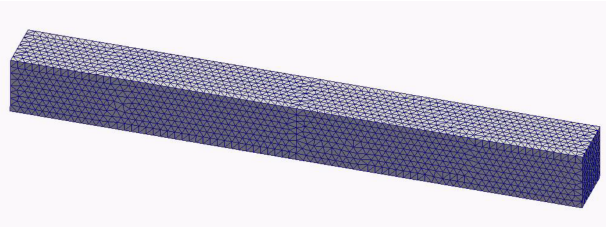

Figure 1: Sod shock tube: geometry and mesh.

directly derived from equation of state (6) and γ is fixed to $5/3$.

	Left	Right
density (ρ)	1.0	0.125
pressure (p)	1.0	0.1
velocity ($\mathbf{u}$)	0.0	0.0

Table 1: Sod shock tube: geometry and mesh.

A time interval $t = [0, 0.2]$ is considered. Results at final time $t = 0.2$ are compared with the analytical solution in Figure 2 showing a good agreement. Contour plot of the final solution are depicted in Figure 3.

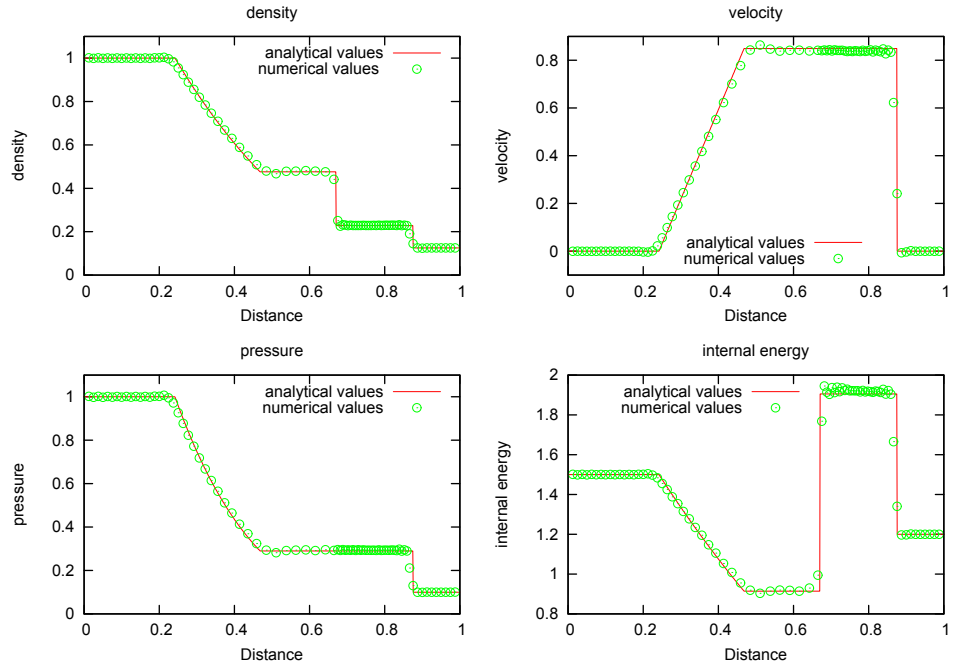

Figure 2: Sod shock tube: results at $t = 0.2$ (horizontal line in the middle of the 3D domain).

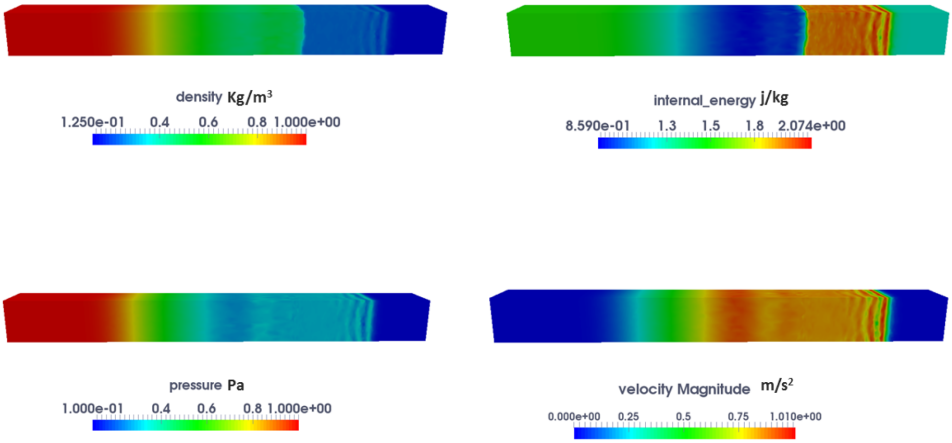

Figure 3: Sod shock tube: contour plot at $t = 0.2$.

5.2 Bubble test

In this test a bubble of helium is immersed in a gas at rest. A shock wave moves from the right size of the box and collides with the bubble. Figure 4 depicts the geometry and the initial conditions of the problem. The mesh is composed by 106.196 nodes and 582.626 elements.

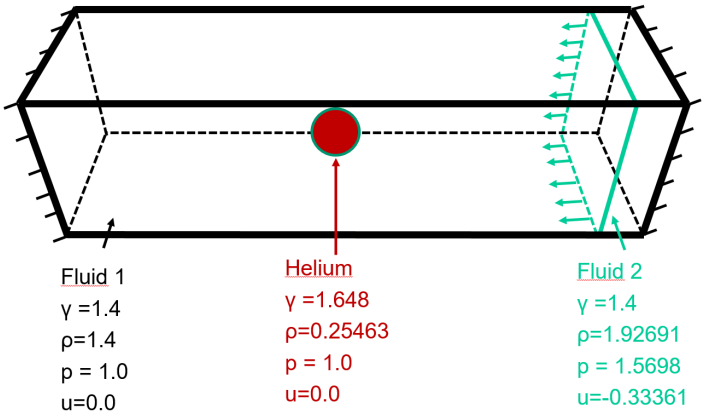

Figure 4: Bubble test: geometry and initial conditions

Figure 5 shows the time evolution of the density and internal energy of a central section of the 3D domain.

6 CONCLUSIONS

In the present work, a Lagrangian finite element method has been introduced for the numerical simulation of compressible fluid flows. Particle Finite Element Method has been used to the numerical solution of the balance equations. An efficient Delaunay tessellation has been used to avoid excessive mesh distortion typical of the Lagrangian flows. An explicit time integration has been proposed to solve rapidly the discretized balance equations. To validate the proposed approach, two benchmarks have been presented.

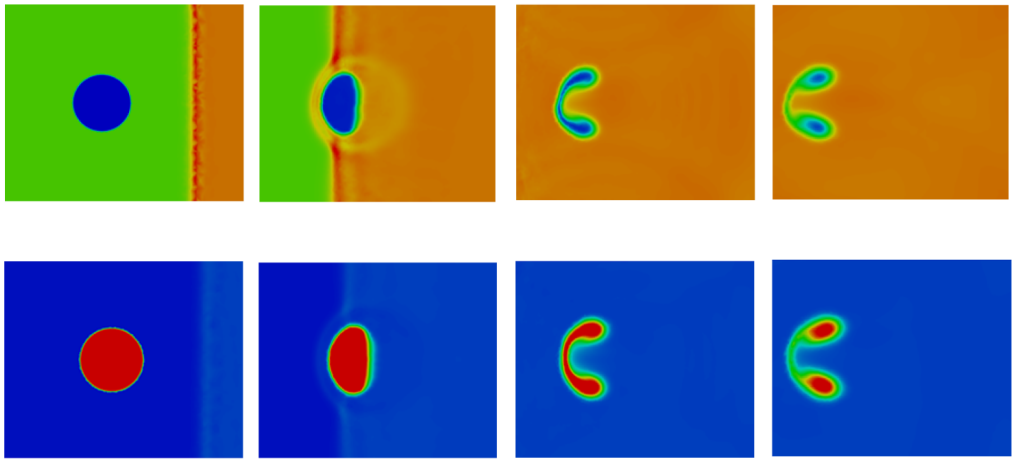

Figure 5: Bubble test. Top: time evolution of the density; bottom: time evolution of the internal energy.

REFERENCES

[1] A. J. Barlow. A compatible finite element multi-material ale hydrodynamics algorithm. *International Journal for Numerical Methods in Fluids*, 56(8):953–964, 2008.

[2] A. Bernard-Champmartin, J.P. Braeunig, and J.M. Ghidaglia. An eulerian finite volume solver for multi-material fluid flows with cylindrical symmetry. *Computers & Fluids*, 83:170 – 176, 2013. Numerical methods for highly compressible multi-material flow problems.

[3] J.-P. Braeunig, B. Desjardins, and J.-M. Ghidaglia. A totally eulerian finite volume solver for multi-material fluid flows. *European Journal of Mechanics - B/Fluids*, 28(4):475 – 485, 2009.

[4] M. Cremonesi, L. Ferrara, A. Frangi, and U. Perego. Simulation of the flow of fresh cement suspensions by a lagrangian finite element approach. *Journal of Non-Newtonian Fluid Mechanics*, 165:1555–1563, 2010.

[5] M. Cremonesi and A. Frangi. A lagrangian finite element method for compressible flow applications. *submitted for the pubblication*, 2016.

[6] M. Cremonesi, A. Frangi, and U. Perego. A lagrangian finite element approach for the analysis of fluidstructure interaction problems. *International Journal of Numerical Methods in Engineering*, 84:610–630, 2010.

[7] V.A. Dobrev, T.E. Ellis, T.V. Kolev, and R.N. Rieben. Curvilinear finite elements for lagrangian hydrodynamics. *International Journal for Numerical Methods in Fluids*, 65(11-12):1295–1310, 2011.

[8] V.A. Dobrev, T.V. Kolev, and R.N. Rieben. High-order curvilinear finite element methods for lagrangian hydrodynamics. *SIAM Journal on Scientific Computing*, 34(5):B606–B641, 2012.

[9] S. R. Idelsohn, E. Oñate, and F. Del Pin. The particle finite element method: a powerful tool to solve incompressible flows with free-surfaces and breaking waves. *International Journal for Numerical Methods in Engineering*, 61:964–989, 2004.

[10] S. R. Idelsohn, E. Oñate, F. Del Pin, and N. Calvo. Fluid-structure interaction using the particle finite element method. *Computer methods in applied mechanics and engineering*, 195:2100–2113, 2006.

[11] K. Kamran, R. Rossi, E. Oñate, and S.R. Idelsohn. A compressible lagrangian framework for the simulation of the underwater implosion of large air bubbles. *Computer Methods in Applied Mechanics and Engineering*, 255(0):210 – 225, 2013.

[12] Tz.V. Kolev and R.N. Rieben. A tensor artificial viscosity using a finite element approach. *Journal of Computational Physics*, 228(22):8336 – 8366, 2009.

[13] E. Oñate, S. R. Idelsohn, F. Del Pin, and R. Aubry. The particle finite element method. an overview. *International Journal for Computational Methods*, 1:267–307, 2004.

[14] G. Scovazzi, M.A. Christon, T.J.R. Hughes, and J.N. Shadid. Stabilized shock hydrodynamics: I. a lagrangian method. *Computer Methods in Applied Mechanics and Engineering*, 196(4-6):923–966, 2007.

[15] G. Scovazzi, E. Love, and M.J. Shashkov. Multi-scale lagrangian shock hydrodynamics on q1/p0 finite elements: Theoretical framework and two-dimensional computations. *Computer Methods in Applied Mechanics and Engineering*, 197(912):1056 – 1079, 2008.

[16] J. Waltz, N.R. Morgan, T.R. Canfield, M.R.J. Charest, L.D. Risinger, and J.G. Wohlbier. A three-dimensional finite element arbitrary Lagrangian-Eulerian method for shock hydrodynamics on unstructured grids. *Computers Fluids*, 92(0):172 – 187, 2014.