A NUMERICAL SIMULATION OF FLUID-STRUCTURE INTERACTION PROBLEMS BY A MESH-FREE METHOD

FLORIN FRUNZULICĂ[*,**], ALEXANDRU DUMITRACHE[**], MARIUS STOIA-DJESKA[*]

Abstract. The solving of fluid-structure interaction problems are traditionally based on arbitrary Lagrangian-Eulerian formulations in which moving grids are used. However, for problems with large deformations, moving material interfaces, deformable boundaries and free surfaces these methods can encounter severe difficulties. The mesh-free method is advantageous in this situation. In this paper, we present a mesh-free Smoothed Particle Hydrodynamics type method for the numerical simulation of the fluid flow, which can be easily integrated into a fluid-structure interaction solver. The focus in this work is on the mathematical aspects concerning the construction of the fluid particles and development of the discretized governing equations and on the boundary condition enforcement. Numerical results are presented for a shock tube closed at the left end by an elastically supported piston.

Key words: fluid-structure interaction, mesh-free, SPH, coupling algorithm.

1. INTRODUCTION

Computer-aided simulation of complex engineering problems has become a current reality. The associated mathematical forms, transposed in a discrete form, adapted at the physical phenomenon, implemented into computational programs, allow the solving and analysis of the complex problems from various domains: flow of fluids, structures, problems of fluid-structure interaction, etc. The methods of numerical simulation are based on one of the well-known discretized techniques, the one with finite volumes and finite elements. The spatial discretization for equations of flow requires and depends on the computational network. For the applications with moving boundaries, the computational network should also be a moving one.

In the latest years, due to the disadvantages of using moving computational grids, there are new computational methods drawing the attention of the researchers, which allow the total or partial elimination of the inconvenient

[*] Faculty of Aerospace Engineering, "Politehnica" University from Bucharest, Gh. Polizu Street, no. 1-7, sect. 1, Bucharest, Romania, email: ffrunzi@yahoo.com

[**] Institute of Statistics and Applied Mathematics, 13 September Street, sect. 5, Bucharest, Romania

Rev. Roum. Sci. Techn. – Méc. Appl., Tome 55, Nº 2, P. 135–144, Bucarest, 2010

elements from the traditional methods. The basic idea of these new methods is to eliminate the computational network, reason for which they are called *mesh-free methods*. A general particularity of these methods is to express the equations in a Lagrangian reference system.

This paper presents the implementation of a SPH-type method to simulate a fluid flow and the fluid-structure interaction problems.

2. BASIC PRINCIPLES OF SPH

Integral Representation of a Function. The following identity stands at the basis of the integral representation of a scalar field $f(\mathbf{x})$:

$$f(\mathbf{x}) = \int_{\Omega} f(\mathbf{x}') \cdot \delta(\mathbf{x} - \mathbf{x}') \mathrm{d}\mathbf{x}', \tag{1}$$

where δ is the Dirac function. If the Dirac function is replaced with a smoothing function or nucleus, $W(\mathbf{x} - \mathbf{x}', h)$, Fig. 1, then the equation (1) becomes:

$$\langle f(\mathbf{x}) \rangle = \int_{\Omega} f(\mathbf{x}') \cdot W(\mathbf{x} - \mathbf{x}', h) \mathrm{d}\mathbf{x}', \tag{2}$$

where h is a smoothing length defining the support domain of the nucleus of function W. The smoothing function is compulsorily even, positive, normed and satisfies the condition of compact support:

$$W(\mathbf{x} - \mathbf{x}', h) = 0, \quad \text{for} \quad |\mathbf{x} - \mathbf{x}'| > \varepsilon h, \tag{3}$$

where ε is a positive constant.

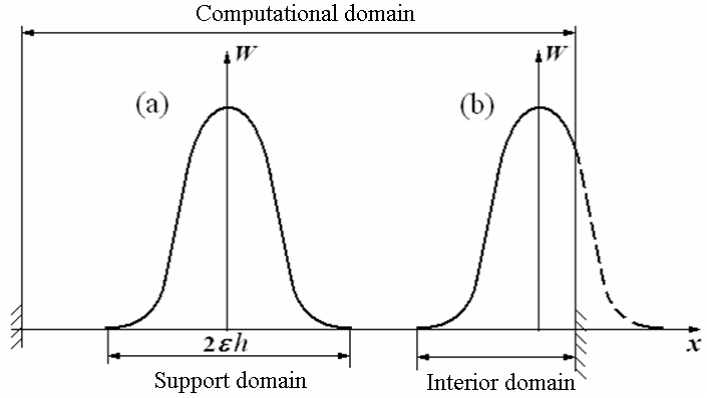

Fig. 1 – Support domain of the function W: a) placed integrally in the computational domain; b) placed at the boundary of the computational domain.

The approximation for the spatial derivatives of the function $f(\mathbf{x})$ is obtained by replacing the function $f(\mathbf{x})$ in the relation (2), with its gradient $\nabla f(\mathbf{x})$:

$$\langle \nabla f(\mathbf{x}) \rangle = -\int_{\Omega} f(\mathbf{x}') \cdot \nabla W(\mathbf{x} - \mathbf{x}', h) \mathrm{d}\mathbf{x}' . \tag{4}$$

Particle Approximation. In the SPH method, the computational domain is represented by a system of N particles, each of them transporting a series of associated parameters. If the volume and density associated to particle j are ΔV_j and ρ_j, then the particle's mass is:

$$m_j = \Delta V_j \rho_j . \tag{5}$$

The basic idea is to approximate the integral of (2) with a summing, after an arbitrary set of particles, which are distributed in the support domain (Fig. 2). Thus, for a particle i, the value of a function and of its derivatives may be approximated with:

$$\langle f(x_i) \rangle = \sum_{j=1}^{N} \frac{m_j}{\rho_j} f(x_j) W_{ij} , \tag{6}$$

$$\langle \nabla f(x_i) \rangle = \sum_{j=1}^{N} \frac{m_j}{\rho_j} f(x_j) \nabla_i W_{ij} , \tag{7}$$

where

$$W_{ij} = W(\mathbf{x}_i - \mathbf{x}_j, h) = W(|\mathbf{x}_i - \mathbf{x}_j|, h), \tag{8}$$

$$\nabla_i W_{ij} = \frac{\mathbf{x}_i - \mathbf{x}_j}{r_{ij}} \frac{\partial W_{ij}}{\partial r_{ij}} = \frac{\mathbf{x}_{ij}}{r_{ij}} \frac{\partial W_{ij}}{\partial r_{ij}} . \tag{9}$$

In (6), by replacing the function f with the density ρ, we obtain the so-called approximation of the density by summing:

$$\rho_i = \sum_{j=1}^{N} m_j W_{ij} . \tag{10}$$

An improved variant of the derivative's expression given by the equation (7) was proposed by Monaghan [1], by introducing density under the gradient operator:

$$\langle \nabla f(x_i) \rangle = \rho_i \left[\sum_{j=1}^{N} m_j \left[\frac{f(x_j)}{\rho_j^2} + \frac{f(x_i)}{\rho_i^2} \right] \cdot \nabla_i W_{ij} \right]. \tag{11}$$

The smoothing function must allow a numerical approximation, which should represent loyally the desired scalar field. The consistence of the approximation of the function and its derivatives may be studied starting from the development of the variable of field in Taylor series and following the manner in which the approximation represents the equations of the problems studied when the distance between particles goes towards zero.

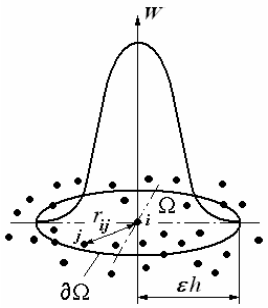

Fig. 2 – The support domain (local) of the particle i
(is represented circularly with the radius εh).

The Gaussian-type nucleus [2] is a smooth function and allows derivation of superior order. However, because it tends to zero when $\overline{r} = r/h \to \infty$, the support domain is relatively big. An alternative is the polynomial function of 4th degree, proposed in [3] (Fig. 3):

$$W(\overline{r},h) = \begin{cases} c_d\left(\frac{2}{3} - \frac{9}{8}\overline{r}^2 + \frac{19}{24}\overline{r}^3 - \frac{5}{32}\overline{r}^4\right), & 0 \le \overline{r} \le 2 \\ 0, & \overline{r} > 2 \end{cases}, \tag{12}$$

where the constant c_d takes the values $1/h$ in one-dimensional, $15/7\pi h^2$ in two-dimensional and $315/208\pi h^3$ in tridimensional. This function is used in this paper.

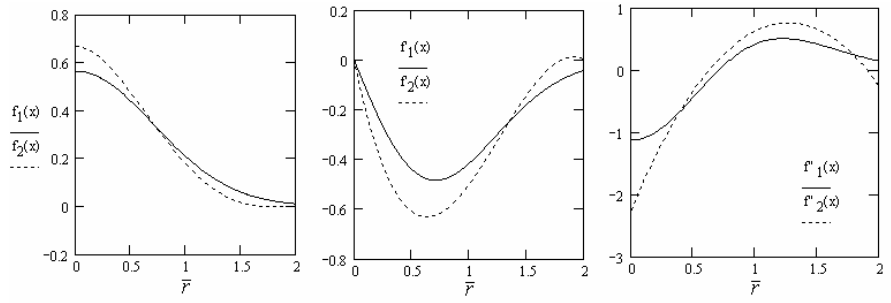

Fig. 3 – Representation of the Gaussiene function (f_1) and of the polynomial function of 4th degree (f_2), as well as of the derivatives of first and second order.

3. SPH METHOD APPLIED TO A FLUID FLOW

SPH Formulation for the Navier-Stokes Equations. In the Lagrange reference system, the fluid particle is "pursued" from an initial point $P_0(x_0, t_0)$ where it was at the initial moment, up to a point $P(x, t)$ where it has arrived in the moment t. One of the most often met models for the continuity equation is built by introducing density under the divergence operator, in discrete form:

$$\frac{D\rho_i}{Dt} = \sum_{j=1}^{N} m_j v_{ij}^{\beta} \frac{\partial W_{ij}}{\partial x_i^{\beta}}, \; v_{ij}^{\beta} = v_i^{\beta} - v_j^{\beta}. \tag{13}$$

In order to increase accuracy of this model [4] the normed form of the equation is used (10):

$$\rho_i = \left(\sum_{j=1}^{N} m_j W_{ij} \right) / \left(\sum_{j=1}^{N} \frac{m_j}{\rho_j} W_{ij} \right). \tag{14}$$

In the absence of the outside forces, one of the discrete forms of the equations of motion used currently in the implementation of SPH method is the following [5]:

$$\begin{aligned} \frac{Dv_i^{\alpha}}{Dt} &= \sum_{j=1}^{N} m_j \left(\frac{\sigma_i^{\alpha\beta}}{\rho_i^2} + \frac{\sigma_j^{\alpha\beta}}{\rho_j^2} \right) \frac{\partial W_{ij}}{\partial x_i^{\beta}} = -\sum_{j=1}^{N} m_j \left(\frac{p_i}{\rho_i^2} + \frac{p_j}{\rho_j^2} \right) \frac{\partial W_{ij}}{\partial x_i^{\alpha}} + \\ &+ \sum_{j=1}^{N} m_j \left(\frac{\mu_i \varepsilon_i^{\alpha\beta}}{\rho_i^2} + \frac{\mu_j \varepsilon_j^{\alpha\beta}}{\rho_j^2} \right) \frac{\partial W_{ij}}{\partial x_i^{\alpha}} \end{aligned} \tag{15}$$

where the deformation speed is given by the relation:

$$\varepsilon_i^{\alpha\beta} = \sum_{j=1}^{N} \frac{m_j}{\rho_j} v_{ji}^{\beta} \frac{\partial W_{ij}}{\partial x_i^{\alpha}} + \sum_{j=1}^{N} \frac{m_j}{\rho_j} v_{ji}^{\alpha} \frac{\partial W_{ij}}{\partial x_i^{\beta}} - \left(\frac{2}{3} \sum_{j=1}^{N} \frac{m_j}{\rho_j} \mathbf{v}_{ji} \nabla_i W_{ij} \right) \delta^{\alpha\beta}. \tag{16}$$

Euler System. If the dissipative terms from the equations (14), (15) and (16) are neglected, it results the discrete form for the Euler equation system, which was used in this paper.

Artificial Viscosity. For the numeric algorithm to be able to capture shock waves without developing oscillations of the solution, a supplementary term is introduced: artificial viscosity. The mathematical formulation of this term, as proposed by Monaghan [6], is:

$$\Theta_{ij} = \begin{cases} \dfrac{-a\,\bar{c}_{ij}\phi_{ij} + b\phi_{ij}^2}{\bar{\rho}_{ij}} , & \mathbf{v}_{ij}\cdot\mathbf{x}_{ij} < 0 \\ 0 , & \mathbf{v}_{ij}\cdot\mathbf{x}_{ij} \geq 0 \end{cases}, \tag{17}$$

where

$$\phi_{ij} = \frac{h_{ij}\mathbf{v}_{ij}\mathbf{x}_{ij}}{\left|\mathbf{x}_{ij}\right|^2 + \varphi^2} , \quad \mathbf{x}_{ij} = \mathbf{x}_i - \mathbf{x}_j , \quad \mathbf{v}_{ij} = \mathbf{v}_i - \mathbf{v}_j , \tag{18}$$

$$\bar{c}_{ij} = \frac{1}{2}\cdot\left(c_i + c_j\right), \quad \bar{\rho}_{ij} = \frac{1}{2}\cdot\left(\rho_i + \rho_j\right), \quad h_{ij} = \frac{1}{2}\left(h_i + h_j\right), \tag{19}$$

and c represents the local speed of sound, the constants a and b have values around 1. The variable $\varphi = 0.1h_{ij}$ is introduced to prevent the numerical divergence when the two particles come close one from another. The artificial viscosity is added at the term of pressure, diffusing the extreme values in the domain of flow and dissipating the energy of terms of high frequency.

The Navier-Stokes equation system becomes:

$$\frac{D\rho_i}{Dt} = \sum_{j=1}^{N} m_j v_{ij}^{\beta}\frac{\partial W_{ij}}{\partial x_i^{\beta}} , \tag{20}$$

$$\frac{Dv_i^{\alpha}}{Dt} = \sum_{j=1}^{N} m_j \left(\frac{\sigma_i^{\alpha\beta}}{\rho_i^2} + \frac{\sigma_j^{\alpha\beta}}{\rho_j^2} + \Theta_{ij}\right)\frac{\partial W_{ij}}{\partial x_i^{\beta}} , \tag{21}$$

$$\frac{De_i}{Dt} = \frac{1}{2}\sum_{j=1}^{N} m_j \left(\frac{p_i}{\rho_i^2} + \frac{p_j}{\rho_j^2} + \Theta_{ij}\right) v_{ij}^{\beta}\frac{\partial W_{ij}}{\partial x_i^{\beta}} + \frac{\mu_i}{2\rho_i}\varepsilon_i^{\alpha\beta}\varepsilon_i^{\alpha\beta} , \tag{22}$$

$$\frac{Dx_i^{\alpha}}{Dt} = v_i^{\alpha} . \tag{23}$$

Smoothing Length *h*. This length has direct implications in SPH because it influences the accuracy of the solution and the efficiency of calculation. There are several methods to evaluate h so that the number of neighbor particles should remain relatively constant. A method is to calculate h as function of average density, with the formula:

$$h = h_0\left(\rho_0/\rho\right)^{1/d} , \tag{24}$$

where index 0 refers to the initial state, and d is the dimension of the space.

Symmetrization of the interaction between particles. If the smoothing length varies in time for each particle separately, then, there may appear frequently the situation that the domain of influence of the particle *i* should include the particle *j*, but the reverse situation might not be possible. In order to remove this drawback, a symmetrical smoothing length should be used for pairs of interacting particles, according to the criterion:

$$h_{ij} = \frac{h_i h_j}{h_i + h_j}. \tag{25}$$

Advance of the Solution in Time and Pressure Calculation. The discrete form of the flow equations in the SPH formulation is integrated in time with an explicit Runge-Kutta-type method. The utilization of an explicit numerical scheme requires the time step to comply with the condition Courant-Friedrichs-Levy (CFL) for numerical stability:

$$\Delta t = \min\left(h_i / c\right). \tag{26}$$

At each time step, new positions of the particles and velocity, density and total energy of each particle are determined, also imposing the boundary conditions. Afterwards, the pressure is calculated, using the state equations.

Imposing the Boundary Conditions. In order to treat the boundary conditions, the method of virtual particles is used, which are of two types. Thus, the first type of particles is placed on the solid boundary and moves with the boundary at the normal velocity $v_n = \mathbf{u}_w \mathbf{n}$. The state parameters of these particles are extrapolated from the inside of the flow domain. Their role is to assure impermeability of the solid boundaries by creating a rejection force on the particles that tend to penetrate the solid surface, [7]:

$$F_{ij} = \begin{cases} \gamma\left[\left(\dfrac{r_0}{r_{ij}}\right)^{n_1} - \left(\dfrac{r_0}{r_{ij}}\right)^{n_2}\right]\dfrac{\mathbf{x}_{ij}}{r_{ij}^2}, & \dfrac{r_0}{r_{ij}} \le 1 \\ 0, & \dfrac{r_0}{r_{ij}} > 1, \end{cases} \tag{27}$$

where the parameters n_1 and n_2 have the values 12 and 4. Choosing the variable γ depends on the type of problem. The distance r_0 activating the rejection force is given by the initial value of the distance between the particles at the level of the boundary. The virtual particles from the second type are placed outside the computational domain and are built by symmetry, so that to achieve the condition at wall.

Searching for the Neighbour Particles. The evolution of the particle system leads to the modification of their position and so, implicitly, of those that are, at a certain time, in the support domain of a particle *i*. Subsequently, a procedure of efficient research is necessary, at each step of time, of the particles neighbor to particle *i*. The application presented here involves a small number of particles; therefore, we have used an algorithm based on an auxiliary Cartesian network overlapped on the computational domain allowing the identification of the neighbors, based on the belonging to a cell of the Cartesian network.

Action on the Solid Boundaries. The pressure is calculated by using the real particles from the neighbourhood of the solid wall, by extrapolation in its previously specified points. Extrapolation may be done using the method of diffuse approximation.

4. FLUID-STRUCTURE INTERACTION MODEL

Fluid-Structure Coupling. The dynamic structural model is represented by the Lagrange equations:

$$m\frac{\mathrm{d}^2 q}{\mathrm{d}t^2} + f\left(q,\dot{q}\right) = Q\,, \tag{28}$$

where q represents the generalized coordinates and Q are generalized forces of the structural model. These forces represent the effect of fluid action on the walls and therefore they depend on the generalized coordinates and speeds. The problem of interaction is solved by a time-staggered-type algorithm [8] illustrated in the scheme from Fig. 4.

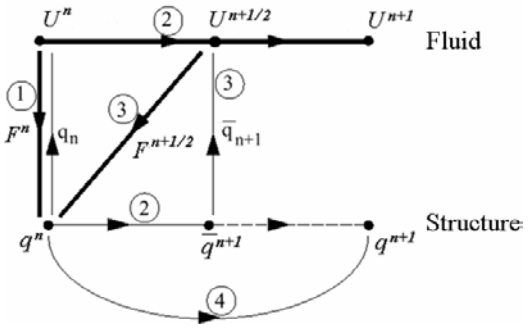

Fig. 4 – The computational algorithm of the fluid-structure interaction.

The important advantage of this type of algorithm is the numerical decoupling of the interaction problem and the separate solution, at each step of time, of the problems of fluid dynamics and structure dynamics.

Advance in Time of the Structural Model. Integration in time of the equation (28) is made using the HHT method (Hilbert, Hughes & Taylor, [9]). The algorithm is unconditionally stable and of second order of precision in time.

5. RESULTS AND CONCLUSIONS

Results. The algorithm was tested on a typical problem of fluid-structure interaction: shock tube closed at one end, the other end being closed with an elastically fixed piston (Fig. 5). The number of particles used to shape the motion of the fluid in the tube is of 500, they having the same mass at the initial moment.

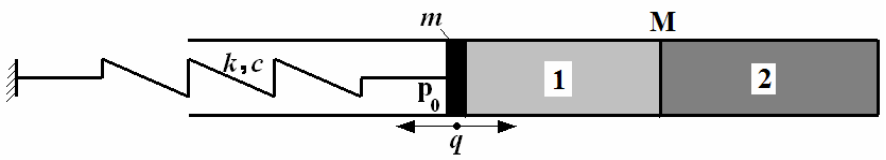

Fig. 5 – The problem of shock tube with a fix wall and a mobile wall.

The system has the following characteristics:
– mechanical assembly:

$$m = 0.5\,\text{kg}, \quad c = 0.01\ \text{kg/s}, \quad k = 10^5\ \text{kg/s}^2;$$

– fluid state at the left and right of membrane M:

$$\rho_1 = 1\ \text{kg/m}^3, \quad p_1 = p_0 = 10^5\ \text{N/m}^2,$$
$$\rho_2 = 10\ \text{kg/m}^3, \quad p_2 = 10^6\ \text{N/m}^2.$$

After the "tearing" of the membrane M ($t = 0$), the Riemann problem leads to the appearance of a shock wave and of the expansion waves, the fluid moving in the tube. The position and speed of the piston in time are presented in Fig. 6. The state of the fluid at the moment $t = 0.743$ ms is presented in Fig. 7. We notice the displacement of the shock wave to the left (toward the piston) whereas the expansion waves move to the right.

Conclusion. For the unsteady, one-dimensional flow problem, in a long tube with moving boundaries, a SPH-type numerical algorithm was implemented and tested. The mathematical model and the numerical scheme will be developed also for two-dimensional and three-dimensional fluid-structure interaction problems.

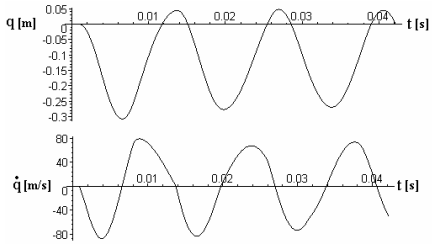

Fig. 6 – Displacement and piston velocity after the initiation of fluid motion in the tube.

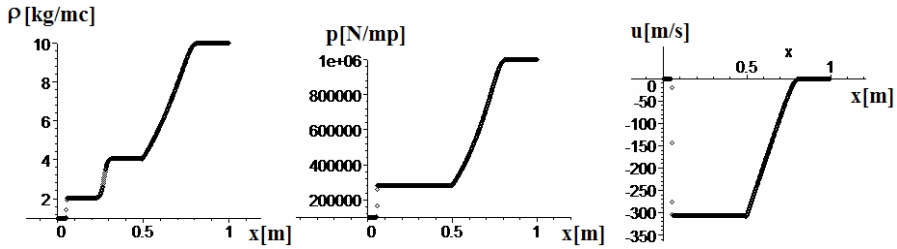

Fig. 7 – Fluid state at the moment $t = 0.743$ ms.

ACKNOWLEDGEMENTS. Supports for this work by project PN2 contract no. 82079/2008 is gratefully acknowledged.

Received on June 6, 2010

REFERENCES

1. MONAGHAN, J.J., *Smoothed particle hydrodynamics*, Annual Review of Astronomical and Astrophysics, **30**, pp. 543-574, 1992.
2. MONAGHAN, J.J., *Particle methods for hydrodynamics*, Computer Physics Report, **3**, pp. 71-124, 1985.
3. LIU, G.R., LIU, M.B., LAM, K.Y., *A general approach for constructing smoothing function for meshfree methods*, Proceedings of the 9th International Conference on Computational in Civil and Building Engineering, pp. 431-436, Taipei, Taiwan, 2002.
4. CHEN, J.K., BERAUN, J.E., CARNEY, T.C., *A corrective smoothed particle method for boundary value problems in heat conduction*, International Journal for Numerical Methods in Engineering, **46**, *2*, pp. 231-252, 1999.
5. LIU, G.R., LIU, M.B., *Smoothed Particle Hydrodynamics – a mesh free particle method*, World Scientific Publishing Co, 2003.
6. MONAGHAN, J.J., POINRACIC, J., *Artificial viscosity for particle methods*, Applied Numerical Mathematics, **1**, pp. 187-194, 1985.
7. LIU, M.B., LIU, G.R., ZONG, Z., LAM K.Y., *Numerical simulation of incompressible flows by SPH*, International Conference on Scientific & Engineering Computational, Beijing, 2001.
8. STOIA-DJESKA, M., FRUNZULICĂ, F., *The numerical simulation of a typical fluid-structure interaction problem*, GAMM 2009 Conference, Gdansk, Poland, February 9-13, 2009.
9. GÉRARDIN, R., RIXEN, D., *Théorie des vibrations*, Edition Masson – Collection: Recherche en Mécanique, 1997.