 CMES, vol.8, no.3, pp.191-200, 2005

Elastodynamics with the Cell Method

F. Cosmi[1]

Abstract: The Cell Method is a recently developed numerical method that is giving interesting results in several fields of physics and engineering. In this paper, first a brief description of the method for elasticity problems is given and successively the elastodynamics formulation is derived. The method leads to an explicit solution system, combining the advantages of a diagonal mass matrix and the possibility of using unstructured meshes. The convergence rate has been tested in reference to the problem of free harmonic vibrations in a system with one degree of freedom, showing that the Cell Method has the same convergence rate of II order Runge Kutta method, but its accuracy is better. The Cell Method results in 2D and 3D have been compared with those obtained with the commercial codes ANSYS and ABAQUS in the problem of the longitudinal vibration of a bar with free ends, for which the exact analytic solution is found in literature. The Cell Method results are comparable with or better than those obtained with FEM, and they are particularly interesting from the point of view of computation time and memory requirements for very large meshes.

keyword: Numerical method, Cell method, Elastodynamics, Transient analysis

1 Introduction

The modeling and simulation of systems behavior by means of numerical methods is a common procedure in the design and development phases of technological and industrial products. One of the most important aspects in the process of mechanical response evaluation consists in a correct estimate of the stress and strain state in the machine components under transient loading, particularly when discontinuities and stress concentrations are present. The prediction of mechanical components behavior by means of numerical simulations, performed on virtual components, accelerates the mechanical systems optimization process and results in an important reduction of experimental tests and project development costs.

A widely used method is the Finite Element Method (FEM), either under the time domain or the frequency domain approach. In particular, transient analysis in time domain is computed by means of numerical integration. Two kinds of numerical integration methods are available: conditionally stable methods and unconditionally stable methods.

The first approach requires a small integration step and the answer of the system at the end of the integration step is computed based on the conditions at the beginning of the step. These methods, also called explicit methods, are very convenient from the computational point of view but are only applicable when the mass matrix is diagonal. The other requirement is that the Courant condition be satisfied: the integration step must be smaller than the minimum period of time required for a disturbance to travel between two nodes of the mesh, the wave propagation velocity in the material being known. Unfortunately, the Finite Element Method mass matrix is in general not diagonal, so that explicit integration methods are not employable. On the other side, the Finite Differences in Time Domain Method (FDTD) yields to a diagonal mass matrix and an explicit system but requires the use of structured meshes that present important drawbacks, for example difficulties in curve shapes modeling.

Unconditionally stable methods, also called implicit methods, compute the answer of the system at the end of the integration step based on the conditions at the end of the step. The answer is numerically stable for any integration step, but this doesn't mean that any step can be used, because the solution accuracy decreases when the step becomes larger. The process is very heavy from the computational point of view, as it requires the solution of an algebraic system at each time step. The computational burden of a Finite Element dynamic analysis, necessarily higher than that of a static analysis, can be reduced if the mass matrix is rendered diagonal (lumping), at the cost

[1] Department of Mechanical Engineering, University of Trieste, Trieste, Italy, cosmi@units.it

of a loss in the solution accuracy.

Therefore, although the Finite Element Method is a very important and widely used tool, there is a motivation for the development of new numerical methods in order to improve the analysis results, with regard to both computation speed and solution accuracy.

The Cell Method (CM) is numerical method recently developed by Tonti (2001a). In general, the results achievable are similar to those obtainable with the Finite Element Method (FEM), although the two approaches are considerably different, as will be discussed in the following. CM is currently applied in several fields, as in heterogeneous materials modelling, biomechanics, diffusion, structural mechanics problems, etc. A list of works is available at http://www.dic.units.it/perspage/discretephysics/.

Tonti (2001b) presented a direct discrete formulation of acoustics in fluids, introducing the classification of physical variables into configuration, source and energy variables and using two cell complexes for both spatial and temporal elements. A section of the same paper is dedicated to a comparison with the Finite Element and Finite Volume Methods where the dynamics integration schemes arising from the different formulations are not discussed.

Marrone, Frasson and Hernàndez-Figueroa (2002) obtained interesting results by employing the CM for dynamic problems in the electromagnetic field.

The application of CM to the elastic analysis of components under dynamic loading is the object of this work. In this paper, it will be shown that for dynamic problems the Cell Method is able to combine the advantages of the Finite Element and of the Finite Differences in Time Domain Methods, since the CM formulation gives a diagonal mass matrix and an explicit system, though it can employ also unstructured meshes.

The object of the next section is a brief account of the basis of the method with reference to elastostatics, preliminary to the development of the elastodynamics formulation that will be presented in the third section. Results of simulations and comparison with commercial FEM codes will be discussed in the fourth and fifth section.

2 The Cell Method for elastostatics

Numerical methods such as Finite Element Method, Finite Difference Method, Boundary Method and Finite Volume Method are all based on differential formulations, in fact they write balance laws by introducing differential relations among the variables of the approximated field. The numerical solution is then obtained by the discretization of differential equations. Several such discretizations are possible, and different methods lead to different set of algebraic equations for the same mesh.

Although widely used, this two-step process of differentiation followed by discretization brings some drawback with it. Let's use the term *local variable* to indicate those variables that can be regarded as densities, as opposed to *global variable*. Pressure and strains are examples of local variables, while forces, relative displacements and displacements are examples of global variables. It can be seen that while local variables are properties of points, global variables are always associated with space elements: displacements are properties of points, relative displacements can be associated to lines, and forces are applied to surfaces or volumes. When a differential formulation is used, everything is reduced to the point, the geometrical information is lost and part of the physics of the problem somehow vanishes in the formulation of the numerical problem. For example, the requirements for derivability impose restrictions on the field equations, although such restrictions are not related to the physics of the problem being examined. A proper discretization of the problem should instead retain the geometrical properties of the variables. Tonti (2001) has recently proposed the Cell Method with the aim of introducing a direct discrete formulation of field laws that satisfies this requirement. CM is therefore applicable whenever variables cannot be differentiated, for example when the displacement field undergoes large variations, i.e. when the size of the heterogeneities coincides with the scale of the discretization (Cosmi and Di Marino (2001), Cosmi (2004)).

A classification of physical variables is at the basis of the Cell Method. The variables involved in a field problem can be classified in one of three classes: configuration, source and energy variables. This classification is general and can be adopted in any field problem, although in this paper reference will be limited to solid mechanics.

A collection of diagrams showing the configuration variables, the source variables and the relations between them for several physical theories can be downloaded from the web site http://www.dic.units.it/perspage/discretephysics/.

Configuration variables describe the geometry and the kinematics of the system. *Source variables* describe the source of the field. Nodal displacements and strain tensor are examples of configuration variables, while forces, momenta and stress tensors are source variables used in mechanics. *Energy variables* derive from the product of a configuration variable by a source variable, and will not be used in the following.

In the Finite Element Method formulation, a unique mesh is defined and a differential formulation is used to establish the balance relations for a point – the node.

With the Cell Method two staggered meshes are used: besides a primal mesh defining the nodes as in FEM, a second complex of cells is created in a dual correspondence with the first one, so that each node of the primal complex falls inside a cell of the dual complex.

The dual cell thus constitutes an influence region for the node inside. The configuration variables are linked to the nodes of the primal complex, while the source variables are associated to the dual complex. Several choices are possible for the dual complex.

In this work only simplicial (triangles and tetrahedra) primal cells have been adopted, while the dual complexes have been always obtained by connecting the barycentres of the primal cells and of their sides and edges. Examples of such primal and dual complexes in 1D, 2D and 3D are shown in Fig.1.

Further considerations and a more detailed description of the geometrical and topological aspects of the Cell Method can be found in Tonti (2001).

In both CM and FEM, the displacement field inside a primal cell is approximated by means of an appropriate function of the nodal displacements that constitute the fundamental unknowns of the problem. In this paper an affine (linear function) interpolation is assumed, but higher order interpolations can be used as well, as shown by Zovatto (2000) and Cosmi (2001). With this assumption, strain components $\{\varepsilon\}_c$ are uniform inside the primal cell and are given by the symmetric part of the displacement gradient:

$$\{\varepsilon\}_c = [B]_c \{u\}_c \tag{1}$$

where $\{u\}$ collects the nodal displacements. It can be easily verified that with this choice of interpolation function the matrix $[B]_c$ is the same as in FEM.

The stress tensor components can then be expressed by introducing the constitutive matrix. Assuming a linear-elastic isotropic behavior of the material, the constitutive equation can be written

$$\{\sigma\}_c = [D]_c \{\varepsilon\}_c = [D]_c [B]_c \{u\}_c \tag{2}$$

where $\{\sigma\}_c$ collects the stress components and $[D]_c$ represents the Hooke's law for the primal cell.

Here the similarities with FEM end. In fact, as already mentioned, CM directly writes the equilibrium equations for the discrete volume constituted by the dual cell, using global (integral) variables, thus obtaining a direct discrete formulation of physical laws.

The forces to be considered for balance are:

- the surface forces $\boldsymbol{T}_h$ acting on the sides of the influence region of node h;
- the resultant of the volume force acting on the influence region and the external forces applied on the influence region through its boundary, $\boldsymbol{F}_h$.

The surface forces $\boldsymbol{T}_h$ acting on the sides of the influence region of node h can be easily obtained by integration of the stress components:

- 1D: $T_h = A_c \sigma_c$
- 2D: $\{T_h\}_c = \frac{1}{2} \begin{bmatrix} A_{hx} & 0 & A_{hy} \\ 0 & A_{hy} & A_{hx} \end{bmatrix} \{\sigma\}_c$,

  and, for the three nodes of the primal cell,

  $\{T\}_c = -tA_c [B]_c^T [D]_c [B]_c \{u\}_c$

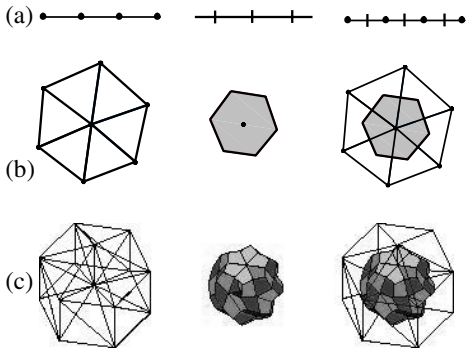

Figure 1 : (a), (b), (c): Respectively: primal, dual complexes of cells and their ensemble in 1D, 2D and 3D.

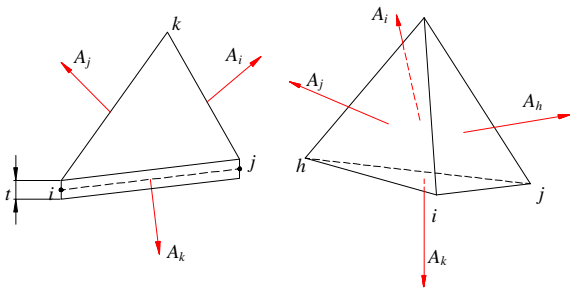

Figure 2 : Geometrical quantities.

- 3D: $\{T_h\}_c = \frac{1}{3}\begin{bmatrix} A_{hx} & 0 & 0 & A_{hy} & 0 & A_{hz} \\ 0 & A_{hy} & 0 & A_{hx} & A_{hz} & 0 \\ 0 & 0 & A_{hz} & 0 & A_{hy} & A_{hx} \end{bmatrix} \{\sigma\}_c$

and, for the four nodes of the primal cell,

$$\{T\}_c = -V_c [B]_c^T [D]_c [B]_c \{u\}_c$$

where: A_c is the area and t the thickness of the primal cell, V_c its volume and the meaning of A_{ij} is shown in Fig.2.

The equilibrium condition for the dual cell, influence region of the node inside, can be written as

$$\boldsymbol{T}_h + \boldsymbol{F}_h = 0 \tag{3}$$

for each of the N influence regions of the N nodes.

The system (1) constitutes a set of linear equations that can also be expressed in the form

$$\{F\} = -\{T\} = [K]\{u\} \tag{4}$$

and solved with the usual methods.

The stiffness matrix $[K]$, which has the same meaning and properties of the FEM one, depends on the choice of the dual complex. With the assumptions stated in this paper (linear interpolation function of nodal displacement and barycentric dual complex), the FEM and CM stiffness matrixes are coincident, while in general the force vectors $\{F\}$ will be different for the two methods.

Opposite to the Finite Element Method, that has a differential formulation at its basis, the Cell Method can be regarded as a direct discrete method that only uses global (integral) variables to write equilibrium. This deep difference between the two methods brings some advantages when the CM is used. For example, the *constitutive matrix can be different* from one cell to the neighbor, so that the heterogeneities characteristic length can be comparable with the mesh size. Moreover, the CM *solution is directly obtained in the nodes*, while with FEM codes the solution is computed in the superconvergent points and then moved to nodal locations by extrapolation (Cosmi (2001)). A further characteristic of the Cell Method is that *no locking* occurs: the solution error will not increase even for very small mesh sizes, being constant at the worst (Zovatto (2000)). This feature makes CM particularly suitable for a local evaluation of stress and strain, as in stress concentration or hot spots.

For static problems, both in the elastic and plastic field, it has been shown in previous works that the accuracy and convergence rate obtainable with the CM are comparable with, and in some cases better than, those achieved with the FEM (Cosmi (2001), Nappi A., Rajgelj S. and Zaccaria D. (2001)).

3 The Cell Method for elastodynamics

When CM is applied to dynamic problems, the two cell complexes used in elastostatics are used again. Moreover, the use of a dual complex is not limited to the discretization of space, but can be extended to the time variable. As shown in Fig.3, the time step is constant and indicated by τ. It is the same for both time complexes, which are staggered of $\tau/2$.

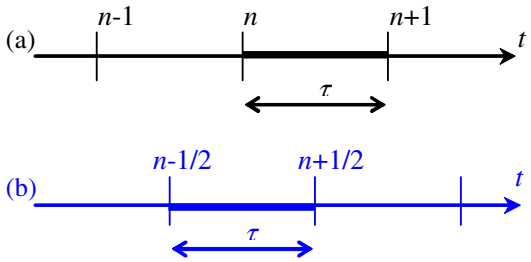

Figure 3 : Primal and dual complexes for the time variable.

Displacements of a node are computed at instants belonging to the primal time complex. Let $\boldsymbol{u}_k^{n-1}$ and $\boldsymbol{u}_k^n$ be the node h displacements respectively at instant n-1 and n. The velocity of node h is defined by the cinematic equation

$$\boldsymbol{v}_h^{n-1/2} = \frac{1}{\tau}\left(\boldsymbol{u}_h^n - \boldsymbol{u}_h^{n-1}\right). \tag{5}$$

Since velocity is computed from the difference between $\boldsymbol{u}_h^n$ and $\boldsymbol{u}_h^{n-1}$, it is naturally associated to the central in-

stant of the time interval that is an instant of the dual time complex.

It has already been noted in statics that the dual cell can be considered as an influence region for the node that rests inside. The linear momentum $\boldsymbol{p}_h^{n-1/2}$ of the influence region of node h will be associated to the instant n-1/2 of the dual time complex and can be computed using the mass m_h of the dual cell and the velocity of node h:

$$\boldsymbol{p}_h^{n-1/2} = \frac{m_h}{\tau}\left(\boldsymbol{u}_h^n - \boldsymbol{u}_h^{n-1}\right). \tag{6}$$

This is equivalent to assume that the barycentre of the dual cell coincides with the node of which the dual cell is the influence region. This condition is approximately verified for the nodes that don't rest on the boundary (see Fig.1). It may introduce a larger error for the boundary nodes, but they are usually a very small percentage of the total number of nodes.

By definition, the resultant force on the dual cell is the time rate of change of momentum. Since momenta are computed at instants in the dual time complex, a difference between momenta is naturally associated with the central instant of the time interval that in this case is an instant of the primal time complex:

$$\boldsymbol{T}_h^n + \boldsymbol{F}_h^n = \frac{1}{\tau}\left(\boldsymbol{p}_h^{n+1/2} - \boldsymbol{p}_h^{n-1/2}\right). \tag{7}$$

Equations (6) and (7) can be written in form of an explicit system that can be solved for each time-step:

$$\begin{cases} \{u\}^n = \{u\}^{n-1} + \tau\left[1/M\right]\{p\}^{n-1/2} \\ \{p\}^{n+1/2} = \{p\}^{n-1/2} + \tau\left(-[K]\{u\}^n + \{F\}^n\right) \end{cases} \tag{8}$$

where $\left[1/M\right] = \mathrm{diag}\left[1/m_1, ..., 1/m_h, ..., 1/m_N\right]$

It should be noted that the procedure described is *not* equivalent to a discretization of the operators, in fact the system (8) has been *directly* written in discrete form.

4 Convergence and accuracy

The problem of free harmonic vibrations in a system with one degree of freedom has been examined (Timoshenko, S.; Young, D.H. (1955)).

Let m be the mass of the body, $x(t)$ the coordinate determining the configuration of the system, k the spring constant, then the well-known balance equation

$$m\ddot{x}(t) + kx(t) = 0 \tag{9}$$

leads to the exact solution

$$x = A\sin\omega t, \quad , \quad A = \omega/\dot{x}(0). \tag{10}$$

Equation (9) can be rewritten in the form

$$\begin{cases} \dot{x} = p(t)/m \\ \dot{p}(t) = -kx(t) \end{cases} \tag{11}$$

and, in order to test the convergence rate and accuracy of the Cell Method integration scheme, as

$$\begin{cases} x^n = x^{n-1} + \tau p^{n-1/2}/m \\ p^{n+1/2} = p^{n-1/2} - \tau k x^n \end{cases} \tag{12}$$

Figure 4 shows the result of the comparison of the maximum errors obtained with the Cell Method and with II and IV order Runge Kutta methods, assumed m=1 kg, k= 4000N/m, A=10 s/m.

It can be seen that the Cell Method has the same convergence rate of II order Runge Kutta method, but its accuracy is better.

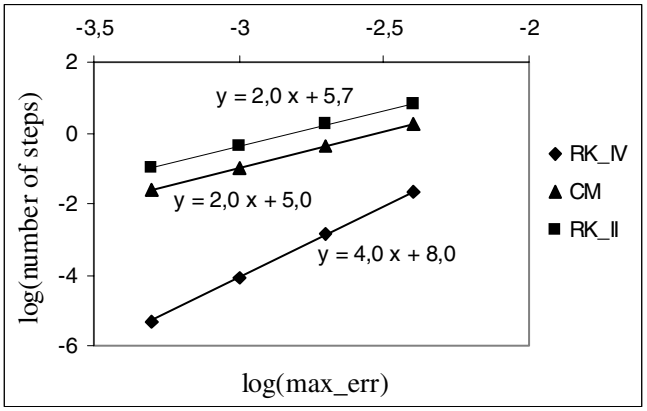

Figure 4 : Results: CM = Cell Method, RK_II and RK_IV = Runge Kutta method, II and IV order respectively.

5 Comparison with FEM codes

A case for which the exact analytic solution is found in literature (Timoshenko, S.; Young, D.H. (1955)) is given by the longitudinal vibration of a bar with free ends (Figure 5) produced by a longitudinal force P suddenly applied at the end $x = l$.

After a period of time t, the displacement of the end of the bar to which the force P is applied is given by

$$(u)_{x=l} = \frac{t^2 P}{2\rho lA} + \frac{2lP}{\pi^2 a^2 \rho A} \sum_{i=1,3,5,\ldots} \frac{1}{i^2} \left(1 - \cos \frac{i\pi at}{2l}\right), \tag{13}$$

where ρ is the mass density of the bar material, l and A are respectively the length and the cross-sectional area of the bar and $a = \sqrt{E/\rho}$ is the velocity of sound in the bar.

Figure 5 : Bar with free ends.

Equation (13) shows that all modes of vibration of the bar are produced. The displacement of the end of the bar when $t = l/a$ (half the fundamental period of vibration) is

$$(u)_{x=l} = \frac{Pl}{2AE} + \frac{4Pl}{\pi^2 AE}\left(1 + \frac{1}{9} + \frac{1}{25} + \ldots\right) = \frac{Pl}{AE} \tag{14}$$

and equals the extension produced in the bar by the action of the uniform tensile load P.

This case has been used in order to compare the Cell Method results with those obtainable with the commercial FEM codes ANSYS and ABAQUS.

Models in 1D, 2D and 3D have been considered. The parameters of the simulations are shown in Table 1.

Table 1 : Parameters of the simulations.

Material	Geometry	Load
elastic modula $E = 210$ GPa $\nu = 0.3$	length of the bar $l = 21$ m	$P = 840$ kN
density $\rho = 7600$ kg/m^3	cross-sectional area $A = 0.7$ m^2	

5.1 1D model

For the 1D simulation, the bar has been divided in 16 primal cells/elements (17 nodes). Table 2 shows the displacement of end of the bar where the load is applied ($x = l$, node 17) when $t = l/a$= 4 ms. The integration time step was Δt = 0.2, 20 steps.

Table 2 : Displacement (mm) of node 17 at $t = l/a = 4$ ms.

Target result	ANSYS	ABAQUS	CM
0.12000	0.11717	0.11982	0.12006
error	-2.35 %	- 0.15 %	0.05 %

In Figure 6, the time history of the displacements computed with the different methods in three nodes are shown: node 1 is in the free end of the bar (x = 0), node 17 is in the other end ($x = l$) and node 9 is the central node ($x = l/2$).

In order to explain the differences it should be noted that a lumped mass matrix was used in the ANSYS analysis, while in the ABAQUS analysis the implicit system was solved at each time step, and the solution has been obtained interactively with Newton's method.

On the contrary, as already mentioned, the solving system obtained with the Cell Method is an explicit system.

Moreover, while in ANSYS and ABAQUS the nodal velocity is obtained as numerical derivative of the nodal displacements, with the Cell Method it is explicitly obtained

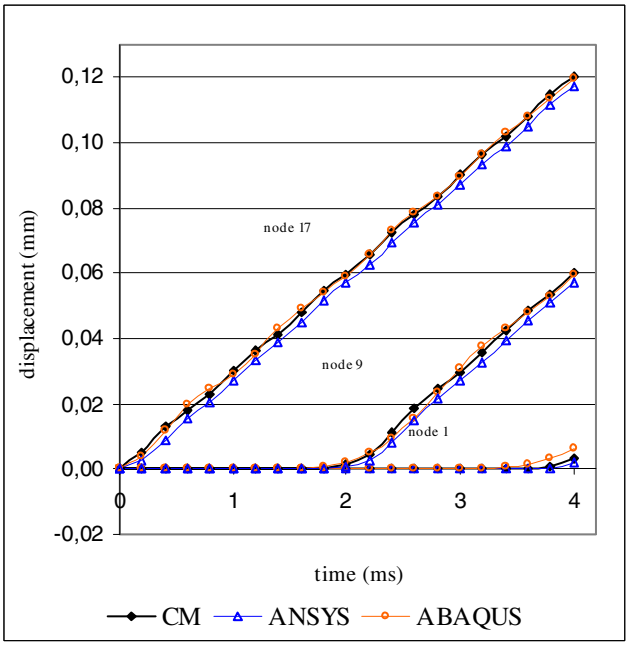

Figure 6 : 1D model, displacements of node 1($x = 0$), node 17 ($x = l$) and node 9 ($x = l/2$).

from the linear momenta of the nodes.

Figures 7, 8 and 9 show the consequent differences arising in the velocity evaluation in node 1(x = 0), node 9 ($x = l/2$) and node 17 ($x = l$) respectively.

5.2 2D and 3D models

For the plane analysis, three nodes triangular cells/elements have been used, while four nodes tetrahedra have been adopted for the 3D analysis.

In Table 3, the main characteristics of the models are summarized.

Table 3 : Models.

mesh	nodes	cells/elements
2D_1 (mapped)	605	960
2D_2 (mapped)	183	240
2D_3 (Delaunay)	186	246
3D_1 (Delaunay)	456	1,051
3D_2 (mapped)	732	2,160
3D_3 (mapped)	190,944	1,031,136

For the 2D and 3D models, only the comparison between Cell Method and ABAQUS was considered.

The load was applied as a uniform tensile stress applied at the end of the bar, and the displacement was computed as the average value of the displacements in the loaded extremity, where $x = l$. The time step used in these simulations was Δt = 0.02 ms (200 steps), except for model 3D – 3, where Δt = 0.0025 ms (1600 steps) was used in order to satisfy Courant condition for the extremely small elements.

The analyses results are summarized in Table 4. It can be seen that the results are practically independent from the number of nodes used. Moreover, since the 2D and 3D models have comparable element sizes, the solution obtained with the Cell Method appears to be more stable when changing from 2D to 3D analysis.

Figures 10, 11 and 12 show the different types of 2D mesh used in the simulations. In the same figures the results computed at t= 4ms are shown.

The CM results in the longitudinal x direction are numerically different from those obtained with ABAQUS, as shown in Table 4, but the pattern is very similar, as seen in (a) vs. (c).

The results in the y direction show instead a different pat-

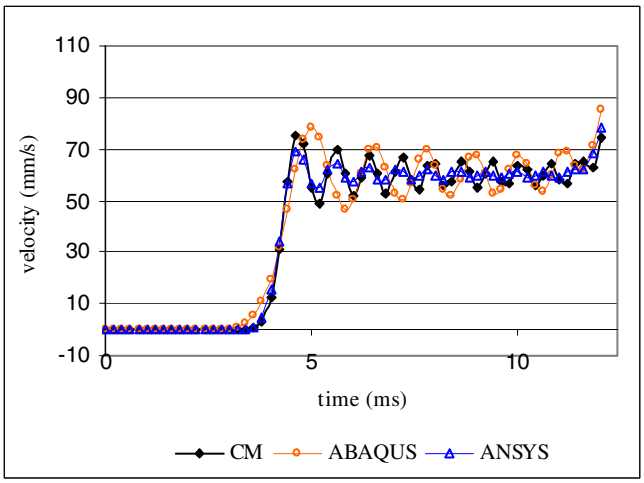

Figure 7 : 1D model, velocity of node 1, $x = 0$.

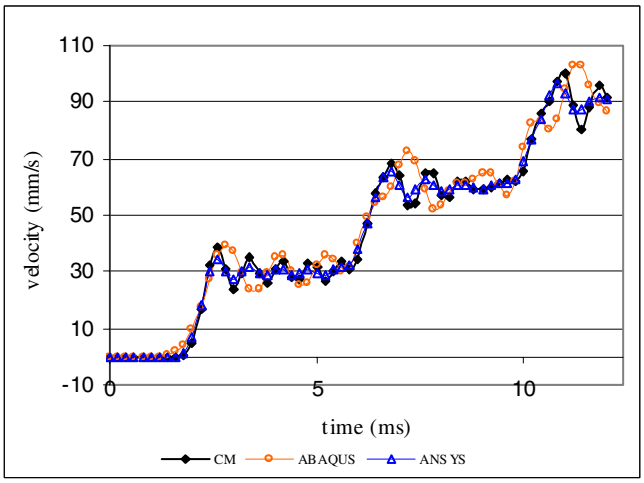

Figure 8 : 1D model, velocity of node 9, $x = l/2$.

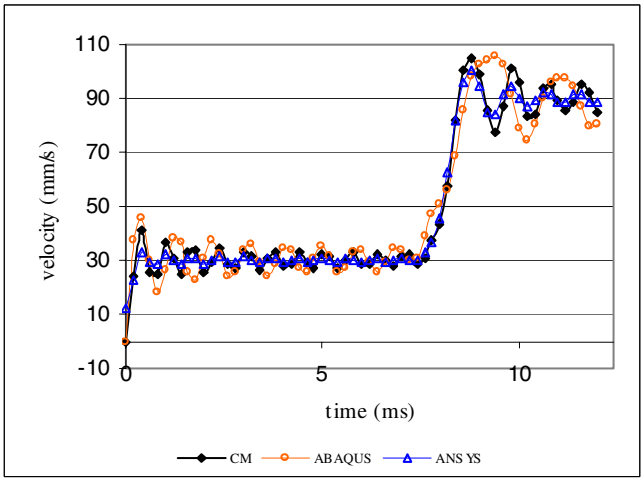

Figure 9 : 1D model, velocity of node 17, $x = l$.

tern, the ABAQUS results displaying some irregularities, as in Figure 11 (b) vs. (d). In general, the differences between CM and ABAQUS results appear larger when a mesh with a lower number of elements is used.

Table 4 : Results.

mesh	target result	CM	ABAQUS
2D_1	0.12000	0.12011	0.11768
2D_2	0.12000	0.12019	0.11787
2D_3	0.12000	0.12004	0.11793
3D_1	0.12000	0.12018	0.12012
3D_2	0.12000	0.12022	0.12022
3D_3	0.12000	0.12009	–

In Figure 13, the models used for the 3D simulations are illustrated. The longitudinal displacements computed with CM are also shown in the picture, while the ABAQUS results are omitted because coincident.

In Figure 14, CM and ABAQUS results in the z transversal direction are compared for mesh 3D_1.The irregularities of the ABAQUS solution do not appear with CM. Results in the y transversal direction are similar and have not been shown for brevity.

In order to test the performance of the method, a model with more than 10^6 elements (mesh 3D – 3) was used. In this mesh the nodes are arranged as in the 3D-2 mesh shown in Figure 13 (b), but the mesh cannot be displayed due to the extremely small size length of the elements. The computation on a commercial PC (INTEL PENTIUM IV 3,2 GHZ processor with a 4Gb RAM) required approximately 30 minutes for the creation of the global stiffness matrix, plus 25 minutes for the complete solution of the dynamic problem. It was not possible to perform a comparison with ABAQUS on the same PC, because the simulation aborted due to higher memory requirements.

6 Conclusions

In this paper, following a brief description of the Cell Method for elastostatics, the method has been extended for the solution of elastodynamics problems. The proposed formulation combines the advantages of FEM and FDTD, leading to a diagonal mass matrix and an explicit solving system, even if unstructured meshes are used. The results obtained have the same convergence rate of II order Runge Kutta method, but the accuracy of the Cell method is better. In the comparison for both plane an three-dimensional problems, the accuracy of the Cell Method was comparable or better than that of two widely diffused FEM codes, ABAQUS and ANSYS.

In the paper, it has been shown that Cell Method results are also very interesting from the point of view of computation time and memory requirements for very large meshes.

Acknowledgement: The Author is deeply grateful to prof. Enzo Tonti for his precious advices and careful reading of this work. The assistance of Dr. Diego Dreossi in the implementation of the 3D model is acknowledged.

References

Cosmi F. (2001): Numerical solution of Plane Elasticity Problems with the Cell Method, *CMES: Computer Modeling in Engineering & Sciences*, Vol.2, n.3, pp. 365-372

Cosmi F.; Di Marino F. (2001): Modelling of the Mechanical Behaviour of Porous Materials: A New Approach, *Acta of Bioengineering and Biomechanics*, vol.3, n.2, pp. 55-66

Cosmi F. (2004): Two-dimension estimate of effective properties of solid with random voids, *Theoretical and Applied Fracture Mechanics*, Elsevier Science, vol.42, n.2, pp. 183-186.

Ferretti E. (2004): A Cell Method (CM) Code for Modeling the Pullout Test Step-wise, *CMES: Computer Modeling in Engineering & Sciences*, Vol. 6, No. 5, pp. 453-476

Ferretti E. (2004): Crack-Path Analysis for Brittle and Non-Brittle Cracks: A Cell Method Approach, *CMES:Computer Modeling in Engineering & Sciences*, Vol. 6, No. 3, pp. 227-244

Marrone M.; Frasson A.M.F.; Hernàndez-Figueroa H.E. (2002): A Novel Numerical Approach for Electromagnetic Scattering: The Cell Method, Proc. of the 2002 IEEE AP-S and URSI, San Antonio, Texas, USA, vol. 1, pp. 160-163.

Nappi A.; Rajgelj S.; Zaccaria D. (2001): Application of the Cell Method to Elastic-Plastic Analysis. In: Akhtar S. Khan (ed.) *Physics and Mechanics of Finite Plastic & Viscoplastic Deformation*, Neat Press, Fulton, Mariland, pp. 433-434.

Timoshenko, S.; Young, D.H. (1955) Vibration Prob-

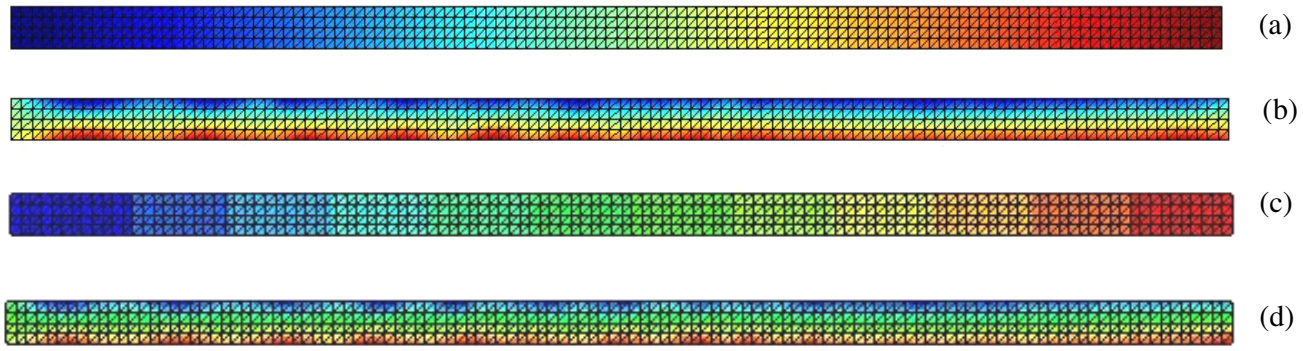

Figure 10 : Mesh 2D_1, displacements at t= 4ms – (a) CM, u_x; (b) CM, u_y; (c) ABAQUS, u_x, (d) ABAQUS ,u_y.

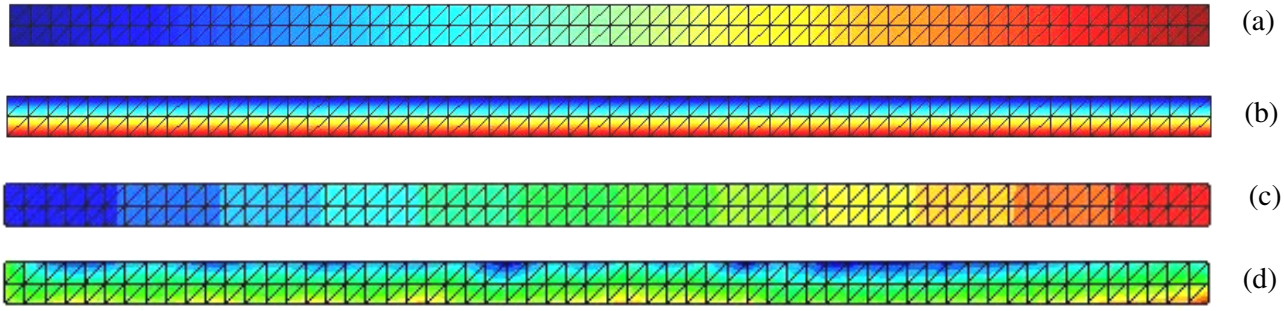

Figure 11 : Mesh 2D_2, displacements at t= 4ms – (a) CM, u_x; (b) CM, u_y; (c) ABAQUS, u_x, (d) ABAQUS ,u_y.

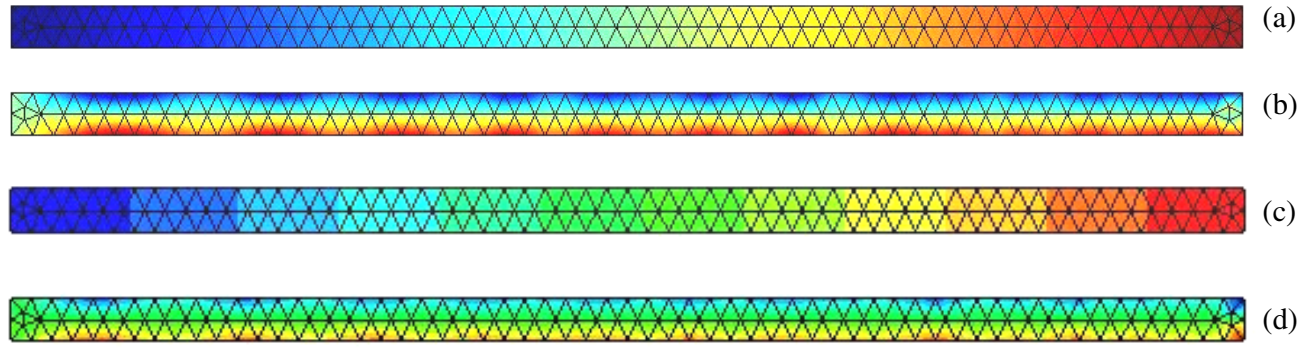

Figure 12 : Mesh 2D_3, displacements at t= 4ms – (a) CM, u_x; (b) CM, u_y; (c) ABAQUS, u_x, (d) ABAQUS ,u_y.

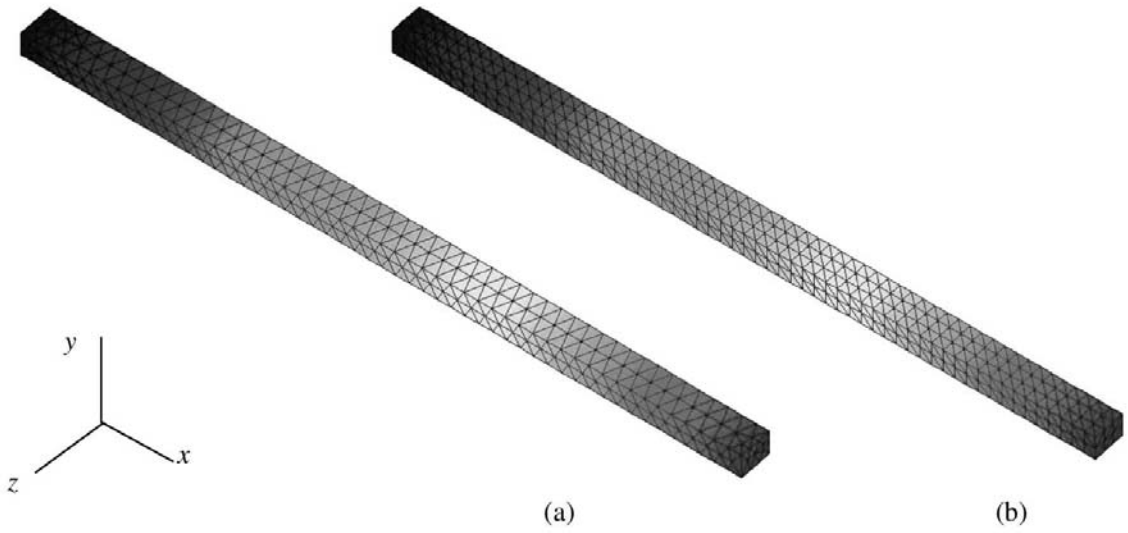

Figure 13 : CM results, longitudinal displacements u_x at $t = 4$ms – (a) mesh 3D_1; (b) mesh 3D_2.

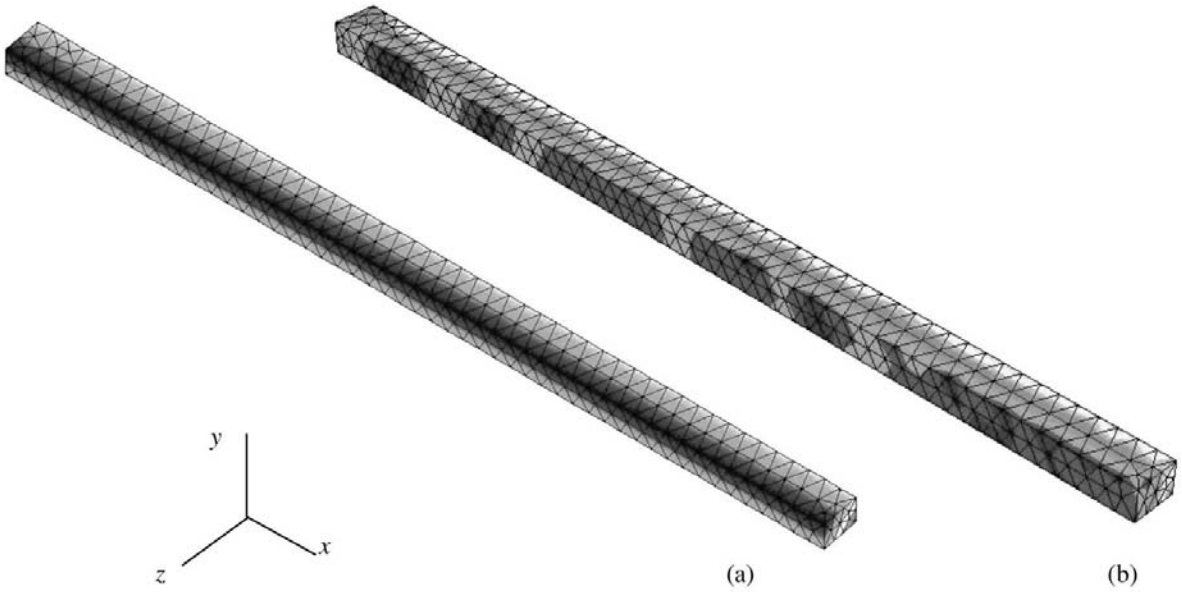

Figure 14 : Mesh 3D_1, results at $t = 4$ms, transversal displacements u_z – (a) CM; (b) ABAQUS.

lems in Engineering, 3rd ed., D. Van Nostrand Co., Inc. New York, pp. 1-5 and 309-311.

Tonti, E. (2001a): A Direct Discrete Formulation of Field Laws: The Cell Method, *CMES: Computer Modeling in Engineering & Sciences*, vol. 2, n.2, pp. 237-258.

Tonti, E. (2001b): A Direct Discrete Formulation for the Wave Equation, *Journal of Computational Acoustics*, vol. 9, n.4, pp. 1355-1382.

Zovatto, L. (2000): Ordine di convergenza superiore in un approccio discreto", Proc. of GIMC 2000, Brescia, Italy, pp. 622-643, in Italian.

ANSYS®Analysis Guide

ABAQUS®Standard User's Manual.