



**HAL**  
open science

## Influence of mesh density on a finite element model's response under dynamic loading

Sebastien Roth, Jennifer Oudry, Marwan El-Rich, Hossein Shakourzadeh

► **To cite this version:**

Sebastien Roth, Jennifer Oudry, Marwan El-Rich, Hossein Shakourzadeh. Influence of mesh density on a finite element model's response under dynamic loading. *Journal of Biological Physics and Chemistry*, 2009, 9 (4), pp.210-219. 10.4024/39RO09C.jbpc.09.04 . hal-04327380

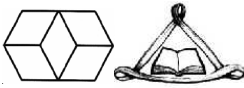
**HAL Id: hal-04327380**

**<https://hal.science/hal-04327380>**

Submitted on 6 Dec 2023

**HAL** is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.



# Influence of mesh density on a finite element model's response under dynamic loading

Sébastien Roth,<sup>1,\*</sup> Jennifer Oudry,<sup>2</sup> Marwan El Rich<sup>3</sup> and Hossein Shakourzadeh<sup>3</sup>

<sup>1</sup> *Université de Technologie de Belfort-Montbéliard UTBM, Laboratoire M3M, 90010 BELFORT Cedex, France*

<sup>2</sup> *Echosens, Research and Development Department, 153 avenue d'Italie, 75013 PARIS, France*

<sup>3</sup> *Altair Engineering France, 2 rue de la Renaissance, 92184 ANTONY Cedex, France*

This paper investigates the influence of mesh density of two-dimensional and three-dimensional finite element models (FE) on elastic wave propagation. These models with their elastic material properties were generated using different mesh sizes and were subjected to a dynamic sinusoidal vibration, in order to investigate the mesh density dependence of a FE model under dynamic loading, such as in crash biomechanics where propagating waves are generated in biological soft tissues. Responses in terms of wave propagation, shear stress and longitudinal displacement and velocity induced by propagation of the elastic waves were studied. Results showed that mesh size has a great influence on the propagation of shear waves and on the maximum shear stress. The need for a specific mesh size, depending on shear wavelength, to ensure an accurate FE model was demonstrated. The results are in agreement with previous studies analysing two-dimensional meshing.

**Keywords:** dynamic load, finite element analysis, meshing size, shear wave

## 1. INTRODUCTION

With the evolution of computer science, numerical approaches such as finite element methods are more and more used to understand physical mechanisms. These numerical tools are often used to extend and complete difficult and costly experimental methods and investigations. Finite element analysis is already widely used in fields such as mechanical and civil engineering [1, 2], understanding crashes [3, 4] and biomechanics [5, 6], and allows stresses and strains to be investigated at both local and global levels. In the field of biomechanics of impact, for example, FE codes are used for human body modelling. The human soft tissues are submitted to dynamic loadings and stress and strain computation enables one to define tolerance limits for the occurrence of injury. For this kind of dynamic simulation, in which waves are created and propagate in the field, the results, precision and stability of the FE analysis are dependent on parameters such as material properties and mesh size.

In general, the more refined the discretization, the more accurate the simulation. However, the number of elements in a finite element model is constrained because of the cost of CPU time and the duration of a simulation. It is therefore important to determine the appropriate number of elements in a FE model. Previous studies [7–11] investigated this point with two-dimensional simulations and led to the conclusion that one element should be smaller than a fraction (about 1/16) of the wavelength of the wave propagating in the FE model in order to minimize the error.

Considering either temporal or spatial discretization, the quality and stability of the FE model response are influenced by parameters such as the numerical stability, fineness of the mesh, numerical dispersion and spurious waves [7–9].

In typical FE computer code (e.g., Radioss, Altair HyperWorks 8.0, Michigan, USA) used for high speed simulations, temporal discretization is performed with explicit integration methods such as the central difference method (CDM). In such an explicit time integration method, exceeding a maximum time step compromises numerical stability. The maximum time step  $\Delta t_{\max}$  to be used is determined by the Courant number  $C$  [7]

$$C = \frac{c\Delta t_{\max}}{\Delta x} \leq 1 ; \quad (1)$$

that is, during one time step the distance  $c\Delta t_{\max}$ , where  $c$  is the wave velocity, travelled by the fastest wave in the model should be smaller than the smallest characteristic element size  $\Delta x$  in the mesh, representing the shortest length needed for a wave arriving at a node to cross the element.

Other effects can influence the response of a FE model. Numerical dispersion and spurious waves are sources of errors directly linked to the spatial discretization (element formulation and mesh density) [8–10]. Numerical dispersion may occur due to waves of different wavelengths propagating at different phase velocities due to numerical artefacts. In the specific case of a one-dimensional uniform grid, the phase velocity  $c_{\text{FEM}}$  in the mesh is related to the theoretical phase velocity  $c$  (eqn 2a) as described in previous papers [8–10]: the wave velocity; is dependent on the wavelength  $\lambda$  and the element size  $\Delta x$ :

\* Corresponding author. E-mail: sebastien.roth@utbm.fr  
Tel.: +33 3 84 58 39 01

$$\frac{c_{\text{FEM}}}{c} = \frac{2}{\Phi} \left[ \left( \sin \frac{\Phi}{2} \right)^{-2} - \frac{2}{3} m \right]^{-1/2} \quad \text{with}$$

$$\Phi = \frac{\omega \Delta x}{c} = \frac{2\pi \Delta x}{\lambda} \quad (2a)$$

where  $\omega$  is the angular frequency and  $m$  is the mass matrix. For a lumped mass matrix,  $m$  is equal to zero. Then, eqn 2a yields:

$$\frac{c_{\text{FEM}}}{c} = \sin \left( \pi \frac{\Delta x}{\lambda} \right) / \pi \frac{\Delta x}{\lambda}. \quad (2b)$$

In the particular case of  $\lambda = \Delta x$ , no wave propagation is found, which can be interpreted in terms of boundary frequency [8, 11] by eqn 3:

$$f_{\text{cutoff}} = c / \Delta x. \quad (3)$$

Fig. 1 shows that for a dispersion ratio  $c_{\text{FEM}}/c$  equal to 99%, the relative wave number  $\lambda / \Delta x$  (the number of elements per wavelength) has to be close to 16.

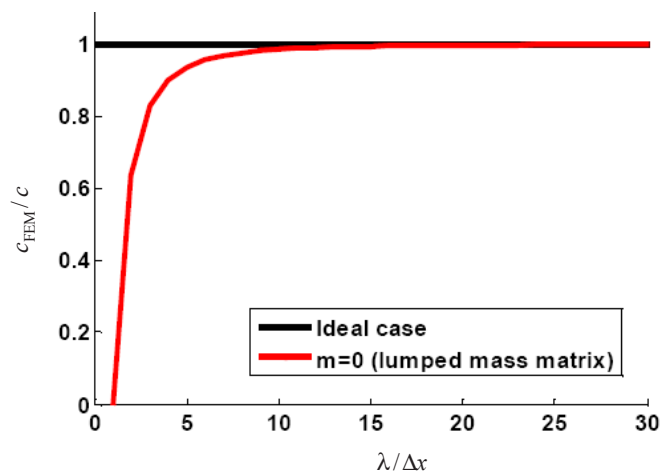


Figure 1. Wave dispersion due to the dependence of wave velocity on wavelength. Adapted from Bazant (1978) [10].

An extension of this analysis has been carried out for a two-dimensional finite element model [9]. For square elements, that study investigated the evolution of the dispersion  $c_{\text{FEM}}/c$  as a function of the relative mesh density  $\Delta x/\lambda$ , for different angles  $\theta$  between the element coordinate system ( $x$  axis) and the wave propagation direction. The numerical dispersion in the mesh (see Fig. 2) increased when  $\theta$  increased from 0 to 45°.

It can be concluded from these studies that for a lumped mass matrix a minimum number of elements per wavelength should be used to reduce the dispersion error to less than 1%, independent of incident angle [9].

Spurious reflexions can occur in the computed solution due to nonuniformity in the discrete mesh. Indeed, in a regular and uniform finite element mesh,

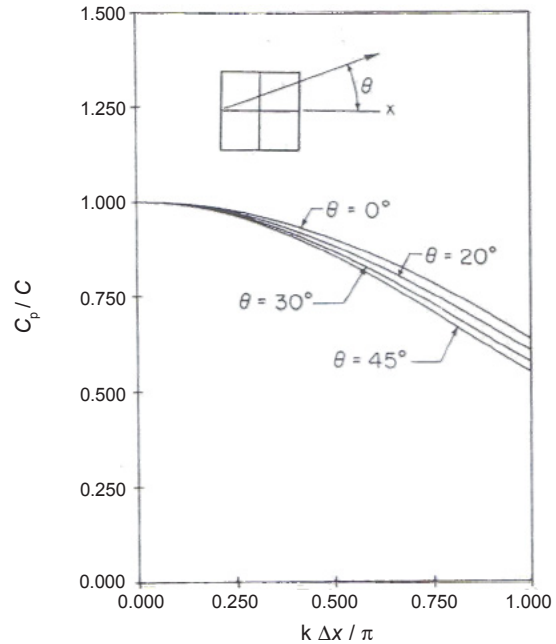


Figure 2. Dispersion of bilinear quadrilateral element with lumped mass matrix as a function of relative wavenumber for various wave propagation directions. Adapted from Mullen and Belytschko (1982) [9].

elastic waves travel without reflexion whereas variations in the mesh size induce spurious reflexions [10–12]. Bazant (1978) [10] described the amplitude of the transmitted and reflected waves at the transition plane between mesh parts in a one-dimensional mesh consisting of two uniform parts with different element sizes. Influences of the element size ratio  $\Delta x_2 / \Delta x_1$  and the number of elements per wavelength of the second mesh part  $\lambda / \Delta x_2$  are shown in Fig. 3.

In the ideal case (uniform mesh) there would be no reflexion and the amplitude of the normalized diffracted wave would be equal to one. Spurious wave reflexion at boundaries of finite elements of different sizes is significant for wavelengths  $\lambda$  that are not much greater than the size of the larger finite element. The heuristic limit is  $\lambda = 10\Delta x_2$  for lumped mass, where  $\Delta x_2$  is the greatest element size.

These studies have shown that several parameters influence the quality of the results in a finite element analysis based on the CDM. As this numerical method is widely used in mechanical engineering, it is of a great interest to analyse the consistency of the results of such a simulation, which is tightly influenced by the discrete mesh density.

In this paper, we propose to illustrate the sensitivity of a numerical model in terms of spatial discretization and precision. Three- and two-dimensional finite element models were developed with different element sizes and were subjected to a dynamic load to generate propagating

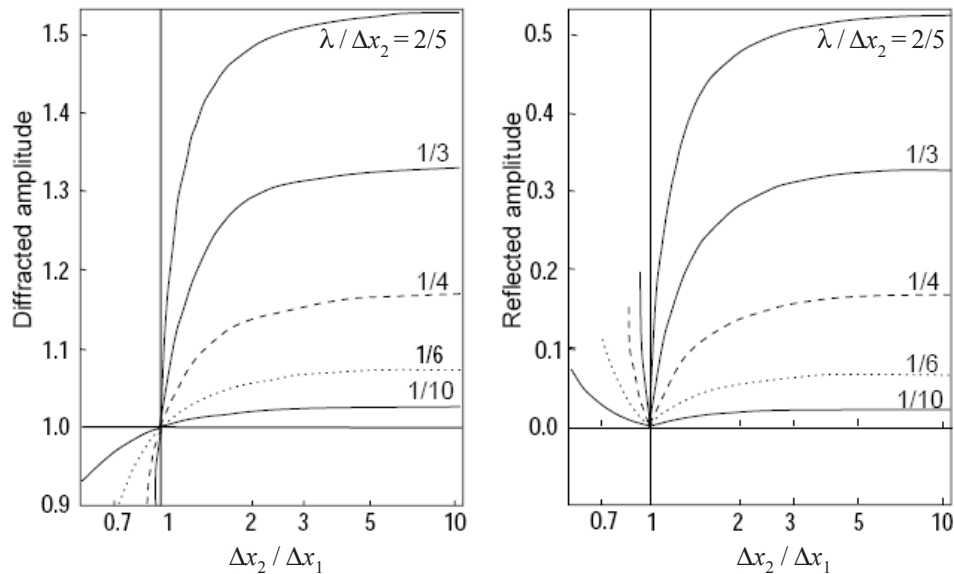


Figure 3. Amplitude of diffracted and reflected wave normalized to the amplitude of the incident wave, as a function of mesh size ratio  $\Delta x_2 / \Delta x_1$  and relative mesh size  $\lambda / \Delta x_2$ . Extract from Bazant (1978) [10] and Brands (2002) [12].

elastic waves in the model. Results in terms of displacement, shear stress and velocity were recorded in order to elucidate the mesh dependence of a FE model under dynamic loading.

**2. METHODS**

**2.1 Basic considerations**

Several finite element models (FEM) in two or three dimensions were developed in a human modelling framework using Hypermesh software (Altair Hyperworks 8.0, Michigan, USA). Radioss crash (Altair Hyperworks 8.0, Michigan) finite element code was used for the simulation. Different types of elements were used in the simulations: 4-node “quad” and “shell” and 8-node “brick” elements for the 2D and 3D simulations respectively.

In order to illustrate the influence of the mesh density, the element size of each FE model was varied from 0.25 mm to 3 mm for “quad” and “shell” elements, and from 0.5 mm to 3 mm for “brick” elements.

The two-dimensional and three-dimensional models were assumed purely elastic, linear, homogeneous and isotropic. In the context of soft tissue modelling, material properties of a liver were considered [13]: a Young’s modulus  $E$  of 4 kPa, a Poisson’s ratio  $\nu$  of 0.49 (quasi-incompressibility of the model) and a mass density  $\rho$  of 1 g/cm<sup>3</sup>.

With the assumption of pure elasticity, the theoretical shear wave velocity  $V_s$  is given by the following equation:

$$V_s = \sqrt{\frac{E}{3\rho}} \tag{5}$$

Therefore, in our case,  $V_s$  was equal to 1.1 m/s and the wavelength  $\lambda_s = V_s / F_0$ , where  $F_0$  is the frequency of

the vibration, was 23 mm. In soft tissues a shear wave propagates at a velocity of a few m/s; thus, a small time step is necessary to monitor displacements induced by the shear wave in the millimetre scale. In addition to a shear wave, the input generates a compressional wave in the environment. In soft tissues, the compressional wave velocity is 1000 times greater than the shear wave velocity, which means that the wave propagates quasi-instantaneously, contrariwise to traditional solids where the shear wave velocity is of the same order as the compressional wave velocity. Indeed, compressional wave velocity in soft tissues is about 1500 m/s, with a corresponding wavelength of 30 m (i.e., significantly greater than the 23 mm wavelength of the shear wave).

**2.2 Two-dimensional numerical simulation**

The basic geometry was a plane (90 × 60 mm). In order to demonstrate the influence of the mesh in a finite element model and to show the importance of mesh size, six FE models were developed with five different mesh sizes: 0.25, 0.5, 1, 2 and 3 mm and one with a varying size (“bias”), from 0.5 to 3 mm. This bidimensional analysis implies planar propagation. The characteristics of the developed models are given in Table 1. For these models at a given wavelength, the number of elements per wavelength varied from 92 to 7.

The dynamic load was applied to one side of the plane making a 45° angle with the  $z$  direction in order to obtain propagation of the wave in a nonperpendicular direction. The applied vibration was a 50 Hz sine wave with an amplitude of 1 mm. An aggregated boundary condition was imposed on the opposite side, as illustrated in Fig. 4.

Table 1. Material properties, dimensions and mesh sizes of the 2D FE models.

Models	Mechanical properties			Geometry of the mesh		
	Young's modulus / kPa	Poisson's ratio	Density / g cm <sup>-3</sup>	Plane dimension	Element size / mm	Element type
2D SHELL 1	4	0.49	1	Width 90 mm Height 60 mm	0.25	4-node quad elements
2D SHELL 2					0.5	
2D SHELL 3					1	
2D SHELL 4					2	
2D SHELL 5					3	
2D SHELL 6					Bias	

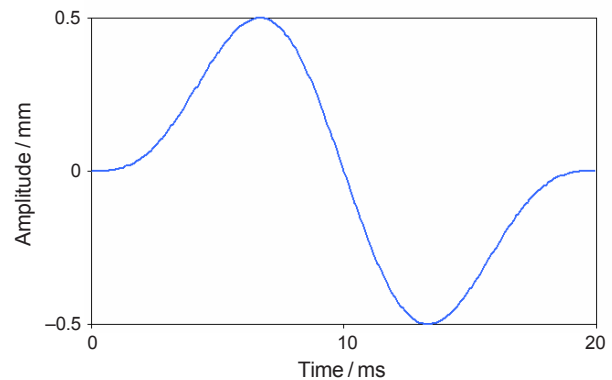
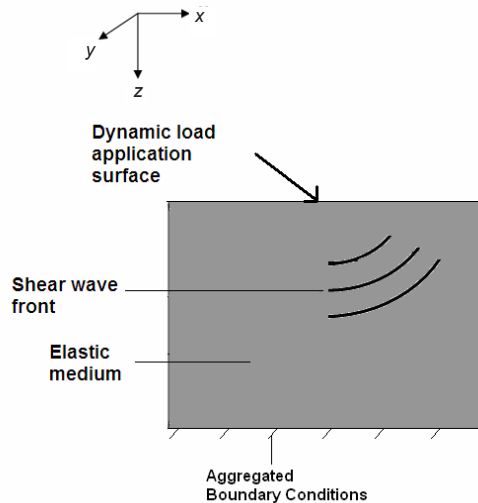


Figure 4. Input dynamic load applied to the plane as a 50 Hz sinusoidal vibration.

### 2.3 Three-dimensional numerical simulation

This study of wave propagation in a 2D FE grid can be extended to a three-dimensional configuration. Two different formulations of three-dimensional elements were used in the simulations.

#### 2.3.1 4-node shell elements

The simulations were conducted in the same way as for the two-dimensional simulations. A plane (90 × 60 mm) was meshed with “shell” elements of 0.1 mm thickness, with six different mesh sizes, as illustrated in Table 2. Input configuration was equivalent to that for the two-dimensional simulations illustrated in Fig. 4.

#### 2.3.2 8-node brick elements

For these 3D simulations, several FE models of a cylinder were developed with different mesh sizes. The material properties used in these models are described in §2.1. Dimensions and material properties of the cylinder are in Table 3.

The dynamic load was applied on the top surface of the cylinder (Fig. 5). The applied vibration was a 50 Hz sine wave with 1 mm amplitude. The application (upper) surface was a 6 mm diameter disc centred according to the symmetry axis. The bottom surface was constrained (no degree of freedom). The dynamic load applied to the finite element model simulates a transient elastography experiment [13].

Table 2. Material properties, dimensions and mesh sizes of the 3D FE models.

Models	Mechanical properties			Geometry of the mesh		
	Young's modulus / kPa	Poisson's ratio	Density / g cm <sup>-3</sup>	Plane dimension	Element size / mm	Element type
3D SHELL 1	4	0.49	1	Width 90 mm Height 60 mm	0.25	4-node shell elements
3D SHELL 2					0.5	
3D SHELL 3					1	
3D SHELL 4					2	
3D SHELL 5					3	
3D SHELL 6					Bias	

Table 3. Material properties, dimensions and mesh sizes of the cylindrical FE models.

Models	Mechanical properties			Geometry of the mesh		
	Young's modulus / kPa	Poisson's ratio	Density / g cm <sup>-3</sup>	Cylinder dimensions	Element size / mm	Element type
3D_BRICK_1	4	0.49	1	Diameter = 90 mm Height = 60 mm	0.5	8-node brick element
3D_BRICK_2					1	
3D_BRICK_3					2	
3D_BRICK_4					3	

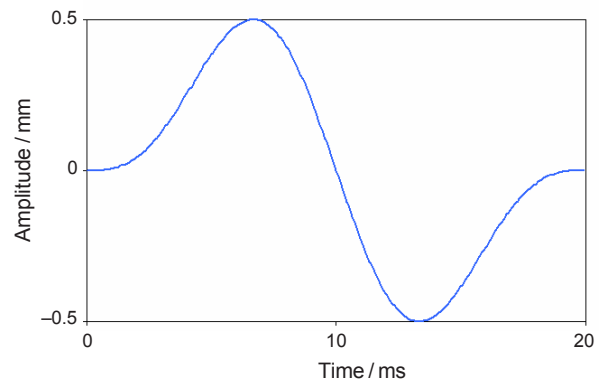
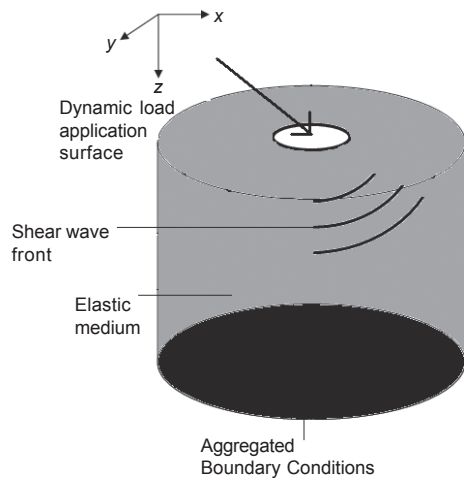


Figure 5. Input dynamic load applied to the cylinder as a 50 Hz sinusoidal vibration.

### 2.4 Variation of loading frequency

In order to investigate the influence of the loading frequency on the mesh density, simulations with 4-node shell elements were run with different loading curves. The 50 Hz sinusoidal vibration was replaced by a 25 Hz sinusoidal vibration as illustrated in Fig. 6.

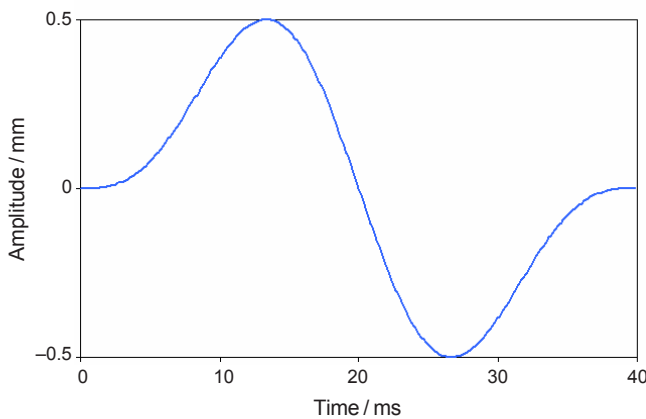


Figure 6. 25 Hz sinusoidal vibration applied in the plane for investigating the frequency dependence of the response.

## 3. RESULTS

### 3.1 Two-dimensional simulations

Results in terms of shearing stress, velocity, displacement and pressure were recorded and compared for the six models at different times as illustrated in Figs 7–10.

Results in terms of shearing stress, velocity and displacement for the two-dimensional simulations show discrepancies in terms of wave propagation. In addition to the size of the elements, which does not give a precise distribution of mechanical parameters, it can be observed that maximum stress, velocity and displacement are not equivalent for meshes greater than 2 mm. The greater the mesh size, the more information in terms of wave front is lost. Differences between the models reached 50%, 55% and 30% in terms of shearing stress, velocity and displacement respectively for two-dimensional simulations. However pressure, wave front and maximum pressure were equivalent for the six models.

### 3.2 Three-dimensional simulations

#### 3.2.1 4-node shell elements

Figs 11–13 give the results of three-dimensional simulations performed with shell elements. Again, great discrepancies can be observed in terms of the velocity and von Mises stress.

For velocity and von Mises stress a critical mesh size can be observed: discrepancies are observed for 1 mm compared to 2 mm and greater. However, decreasing mesh density (0.5 mm and 0.25 mm) increases CPU time and hence duration of a simulation, but does not seem to yield more information.

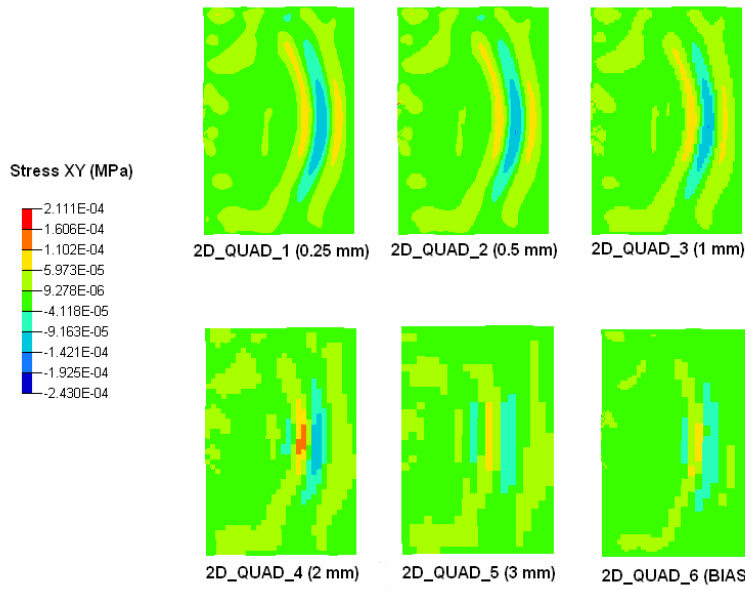


Figure 7. Shearing stress (MPa).

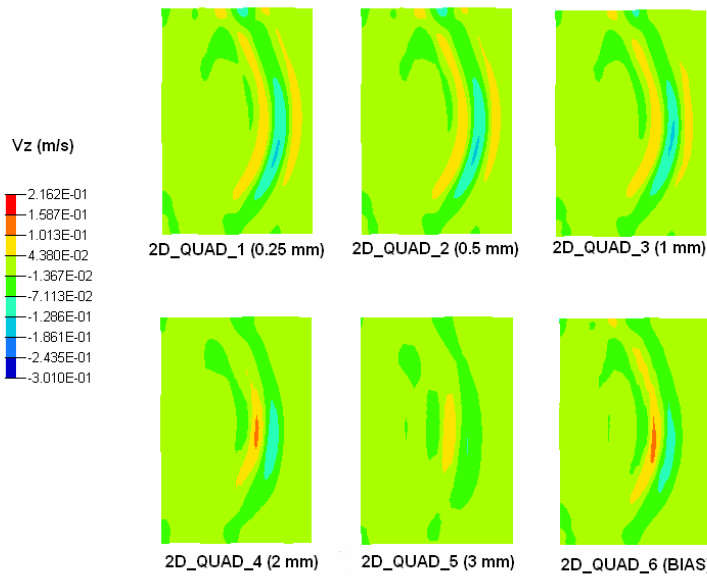


Figure 8. Velocity (m/s) along the z axis.

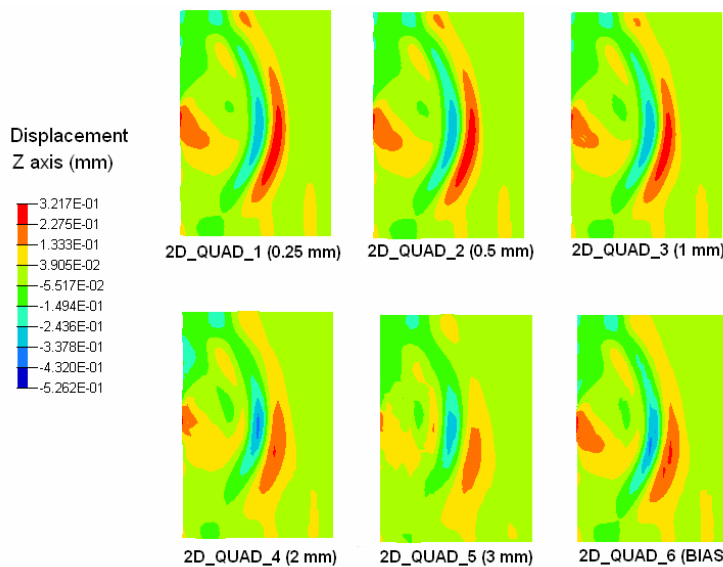


Figure 9. Displacement (mm) along the z axis .

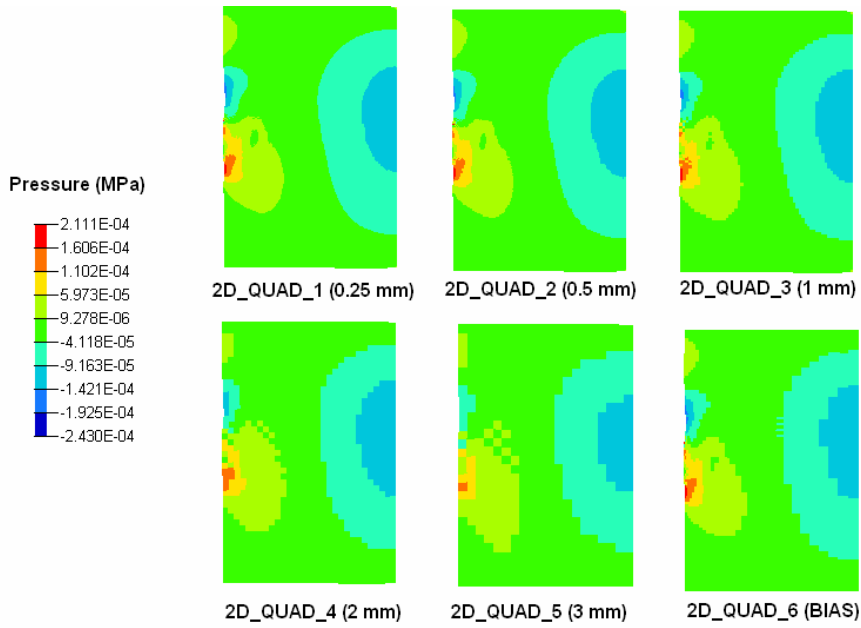


Figure 10. Pressure (MPa).

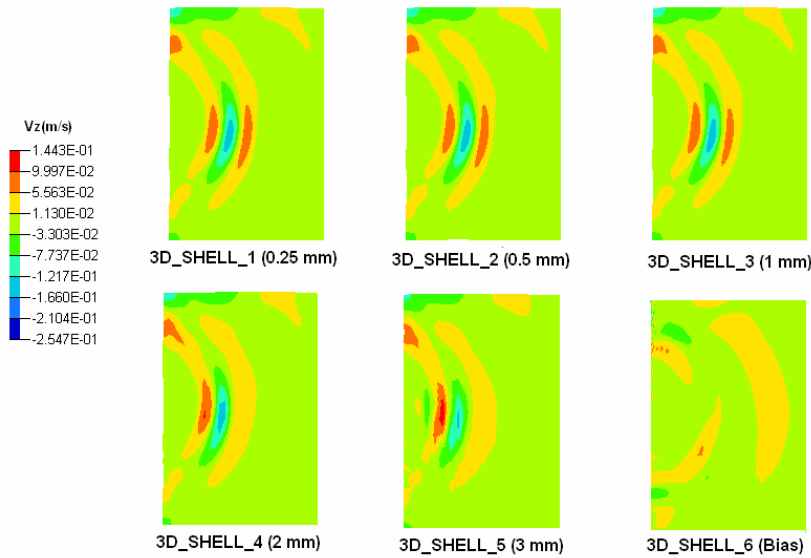


Figure 11. Velocity (m/s) along the z axis.

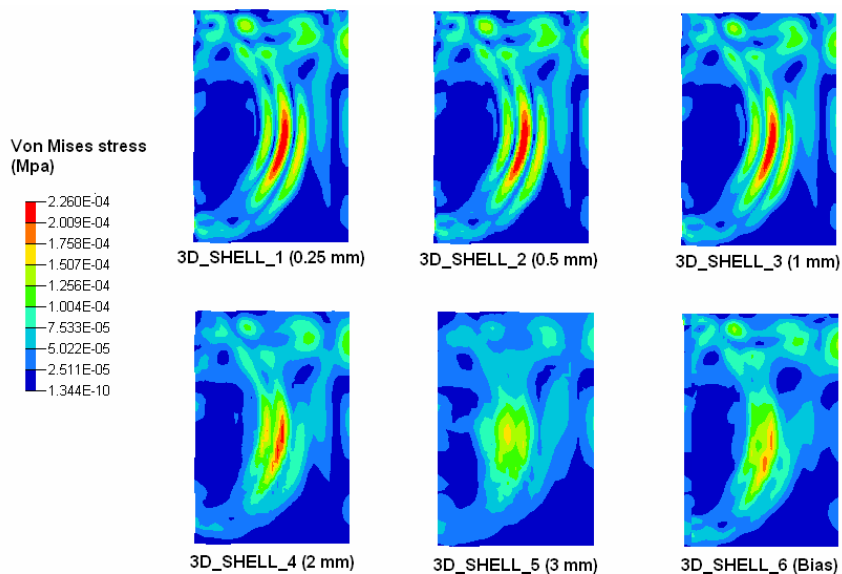


Figure 12. Von Mises stress (MPa).



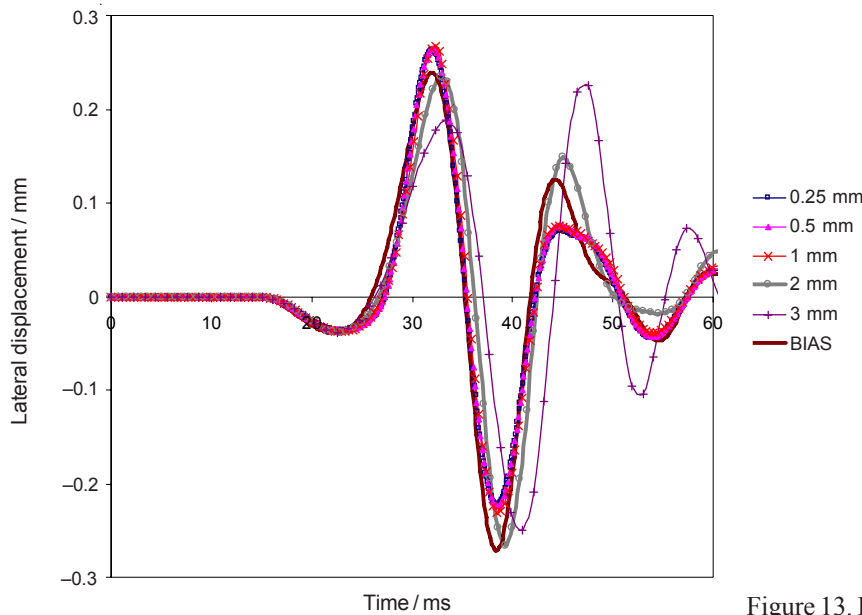


Figure 13. Lateral displacement in the middle of the plane.

Displacement time history has also been investigated in the middle of the solid. Similar displacement curves were obtained for the 0.25 mm, 0.5 mm and 1 mm mesh sizes. However, for 2 mm, 3 mm and bias discrepancies appear, which demonstrate the influence of the mesh on the response of a finite element model.

### 3.2.2 8-node brick elements

The distribution of mechanical parameters was investigated and observed in the middle of the cylinder, as illustrated in Fig. 14. von Mises stress is different in terms of distribution and maximum value.

### 3.3 Frequency dependence

In order to investigate the frequency dependence of the mesh, the same simulations have been performed with 4-node shell elements by decreasing the frequency of the vibration to 25 Hz. For this frequency, and for the material used in the study, the wavelength is 46 mm, which corresponds to almost 16 or more elements per wavelength for all models. Figs 15 and 16 illustrate results in terms of von Mises stress and velocity.

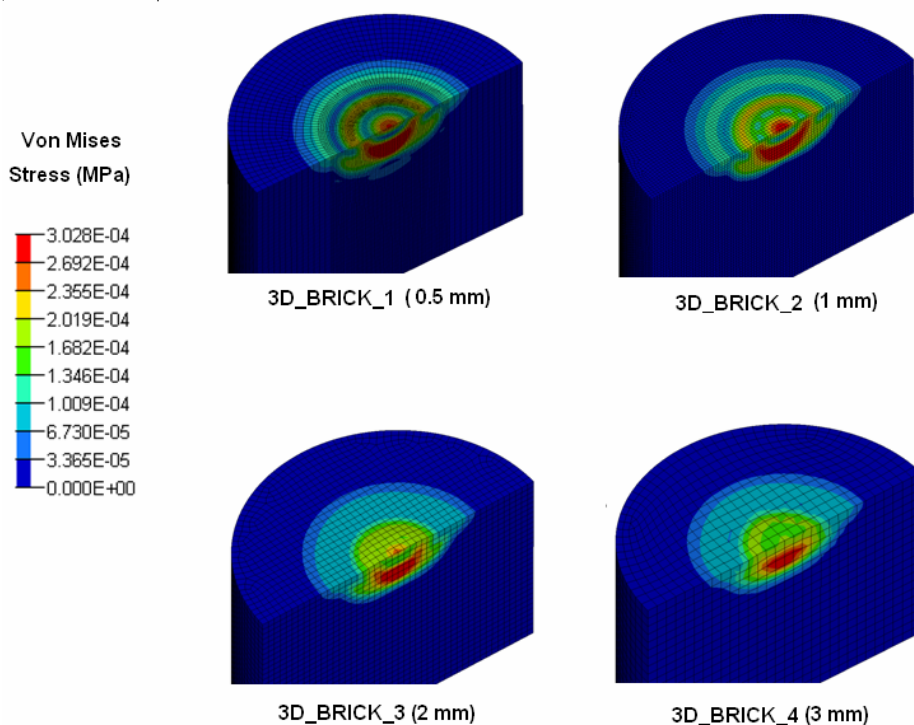


Figure 14. Von Mises stress in the cylinder (MPa).

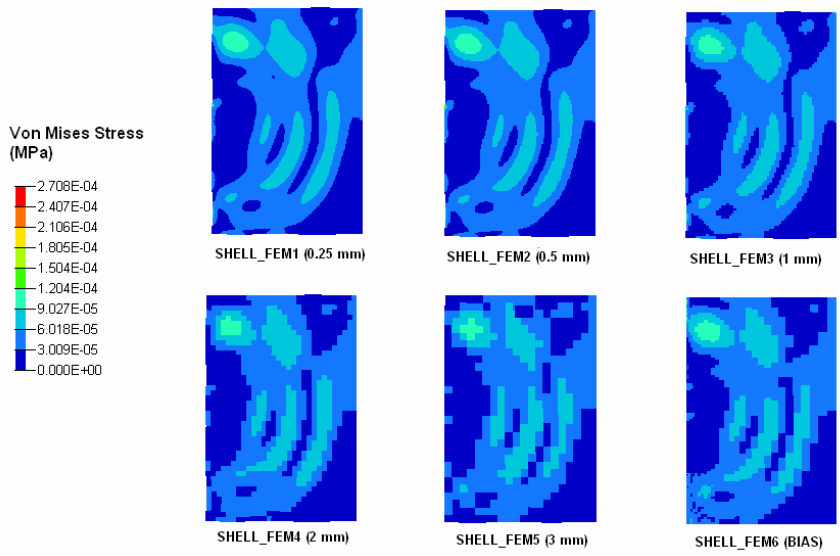


Figure 15. Von Mises stress (MPa) for a 25 Hz sinusoidal vibration.

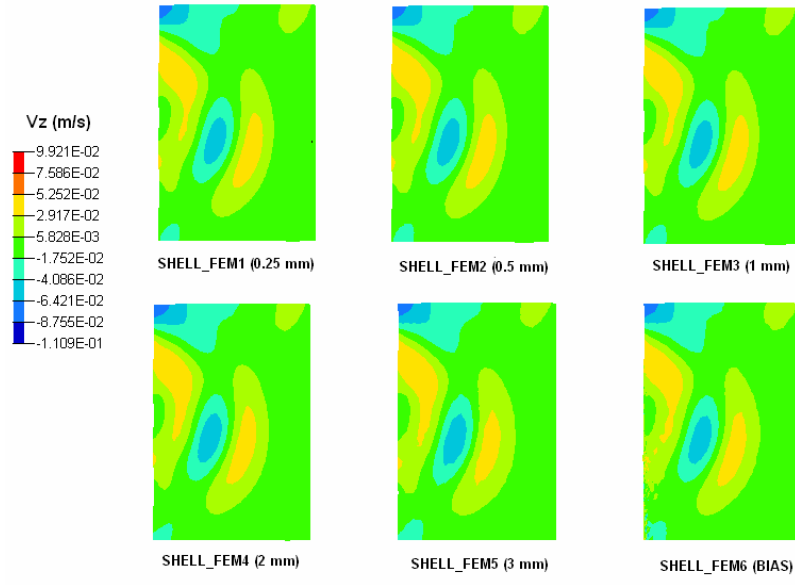


Figure 16. Velocity (m/s) along the z axis for a 25 Hz sinusoidal vibration.

**4. DISCUSSION**

Based on two- and three-dimensional simulations, our results show that for a given elasticity and density almost 16 elements per wavelength is necessary in order to reach an asymptote in terms of mechanical parameters, especially for shear waves (Bazant et al., 1978). Our simulations lead to the same conclusion as Bazant’s study concerning mesh density as a function of frequency.

For pressure, the conclusion is different. Indeed, as far as the pressure distribution is concerned, FE models give almost the same solution for all mesh sizes. This point illustrates the theoretical view of §3.1 concerning the wavelength of the compression wave, which is much greater (1000 fold) than the characteristic mesh size.

Concerning the frequency dependence, simulations with a 25 Hz sine vibration lead to results that are in accordance with previous mesh density studies. Indeed,

the theoretical critical mesh size proposed by previous studies [8–11] changes with wavelength. For a frequency of 25 Hz and for the material used in the present study, the wavelength is 46 mm, which corresponds to almost 16 elements per wavelength for the model 3D\_SHELL\_5 (for 3 mm, the greatest mesh size), which is enough to obtain a correct response from the model according to Bazant’s curve [10]. Thus, results with 3D\_SHELL simulations are equivalent since every model contains almost 16 elements or more per wavelength. Even if the distribution of mechanical parameters (i.e. von Mises stress) would be better described with a fine mesh, no discrepancies can be observed in terms of wave front and maximum stress, as illustrated in Figs 15 and 16.

Finally, when the critical number of elements per wavelength is reached, a further increase of the number of elements is not necessary and even useless: an asymptote

in terms of precision and space discretization is reached: a more precise discretization would considerably increase the CPU time with no significant improvement of the accuracy of the results.

These results impact the development of complex finite element models such as biomechanical ones. The complexity of the geometry can lead to inhomogeneous meshing in turn leading to numerical errors when the mesh is not sufficiently refined. When looking at shear stress or shear strain in biological tissues, one should be aware of the need for an accurate mesh size to correctly simulate the propagation of shearing parameters for a given loading case.

## 5. CRITIQUE

With finite element software like the one used in his study (Radioss, Altair), several formulations of elements are available: anti-hourglass, full integration or partially integrated element. Whereas the mass distribution in an element does not change (lumped mass matrix), the rigidity matrix changes according to the formulation. Then, the response of an element in terms of frequency can change. Other simulations are needed (and planned) in order to investigate the influence of the number of integration points and element formulation on the FE response.

The amplitude of the sinusoidal vibration used in our simulation was quite low. Simulations with higher amplitudes are planned in order to investigate the loading amplitude dependence.

Furthermore, the present simulations investigated the response of a model submitted to a single loading frequency. In crashes or impact biomechanics, models are subjected to multifrequency loadings. This point should be investigated in further simulations.

Finally, simulations have been performed with a linear elastic material law; other studies with more complex material laws are needed.

## 6. CONCLUSIONS

This paper illustrates the mesh dependence of finite element models. The number of elements (mesh density) can significantly influence the response to loadings and hence the ability of the model to reproduce the physical phenomenon studied. Numerical dispersion is a kind of numerical artefact, which can lead to erroneous results in a dynamic simulation. Even with simple models like planes or cylinders, where the mesh is homogenous, this

study has shown that the mesh density influences the results. For more complex models, in which the elements are not necessarily homogeneous, spurious waves appear and can increase the numerical artefacts. For biomechanical models, where shearing parameters tend to be the cause of injuries, the number of elements per wavelength should be kept in mind, as was suggested by Brands et al. [12].

## REFERENCES

1. Cho, T. and Kim, T.S. Probabilistic risk assessment for the construction phases of a bridge construction based on finite element analysis. *Finite Element in Analysis and Design* **44** (2008) 383–400.
2. Mackerle, J. Finite element and boundary element analysis of bridges, roads and pavements bibliography (1994–1997). *Finite Element in Analysis and Design* **29** (1998) 65–73.
3. Eskandarian, A., Marzougui, D. and Bedewi, N.E. Finite element model and validation of a surrogate crash test vehicle for impacts with roadside objects. *Int. J. Crashworthiness* **2** (1997) 239–258
4. Whitworth, H.A. et al. Finite element modeling of the crash performance of roadside barriers. *Int. J. Crashworthiness* **9** (2004) 35–43.
5. Deck, C. and Willinger, R. Improved head injury criteria based on head FE model. *Int. J. Crashworthiness* **13** (2008) 667–678.
6. Roth, S., Vappou, J., Raul, J.S. and Willinger, R. Child head injury criteria investigation through numerical simulation of real world trauma. *Computer Methods and Programs in Biomedicine* **93** (2009) 32–45.
7. Courant, R., Friedrichs K. and Lewy, H. On the partial difference equations of mathematical physics. *IBM J.* **11** (1967) 215–234.
8. Belytschko, T., Guo, Y., Liu, W.K. and Xiao, S.P. A unified stability analysis of meshless particle methods. *International J. Numerical Methods in Engineering* **48** (2000) 1359–1400.
9. Mullen, R. and Belytschko, T. Dispersion analysis of finite element semidiscretizations of the two-dimensional wave equation. *International J. Numerical Methods in Engineering* **18** (1982) 11–29.
10. Bazant, Z.P. Spurious reflection of elastic waves in nonuniform finite element grids. *Computer Methods in Applied Mechanics and Engineering* **16** (1978) 91–100.
11. Bazant, Z.P. and Celep, Z. Spurious reflection of elastic waves in nonuniform meshes of constant and linear strain unite elements. *Computers & Structures* **15** (1982) 451–459.
12. Brands, D.W.A. Predicting brain mechanics during closed head impact—numerical and constitutive aspects, PhD thesis, Eindhoven: Technische Universiteit (2002).
13. Bastard, C., Oudry, J., Mofid, Y. and Sandrin, L. Transient elastography in heterogeneous tissues. *J. Acoustical Soc. Am.* **123** (2008) 3000.