Finite Element Technology

On the propagation of pyrotechnical shocks into complex structures, taking medium frequencies into account
Pierre LADEVÈZE*, Guillaume BÉZIER, Hervé RIOU, Hugo LECLERC (ENS Cachan)

Non-intrusive coupled global/local analysis of localized plasticity problems
Lionel GENDRE*, Olivier ALLIX, Pierre GOSSELET (ENS Cachan), François COMTE (Snecma Villaroche)

Re-triangulation of existing surface meshes with high curvatures
Antonio Carlos MIRANDA, Luiz Fernando MARTHA* (PUC-Rio), Paul WAWRZYNEK, Anthony INGRAFFEA (Cornell University)

Mapping functions in the eight node elastodynamic infinite element with union shape function (EIEUSF)
Konstantin Savkov KAZAKOV (VSU "Luben Karavelov")
On the Propagation of Pyrotechnical Shocks into Complex Structures, taking Medium Frequencies into account

Pierre Ladevèze *, Guillaume Bézier, Hervé Riou, and Hugo Leclerc

* Laboratoire de Mécanique et Technologie
ENS Cachan / CNRS / Université Paris 6 / PRES UniverSud Paris
61, avenue du Président Wilson, F-94230 CACHAN CEDEX
ladeveze@lmt.ens-cachan.fr

Abstract

The design of industrial structures requires the knowledge of their dynamic behaviour. We propose here a computational method for transient dynamic analysis which enables one to cover both the low- and medium-frequency ranges, this last frequency range being often ignored. It is a frequency approach in which the low-frequency part is obtained through a classical technique (FEM for example) while the medium-frequency part is handled through the Variational Theory of Complex Ray (VTCR) initially introduced for vibrations at fixed frequency. The aim is to show the capabilities of the method for the study of pyrotechnical propagating waves in launchers.

1 Introduction

The design of industrial structures requires the knowledge of their dynamic behavior and of their environment. However, the response of such structures cannot be described completely by using current tools based on classical finite elements techniques. The medium-frequency range is often ignored because calculations in that range would require a very refined spatial mesh, leading to prohibitive calculation costs (Bathe [1, 2]) and numerical difficulties (F. Ihlenburg and I. Babuška [3], A. Derämaeker et al. [4]). Yet, accounting for the medium-frequency content may be necessary: while the displacements in this frequency range are small the velocity (and, therefore, the kinetic energy) can be significant. Thus, the transient dynamic analysis of complex engineering structures in the low- and medium-frequency range is an important challenge and would have useful applications, especially in the study of pyrotechnical shocks.

The present work is an attempt to meet this challenge for the transient part of the solution, introduced in Ladevèze and Chevreuil [14]. It uses a frequency approach in which the low-frequency part is obtained through the classical techniques and presents no major difficulty. Conversely, calculation of the medium-frequency part still raises problems, the difficulty residing in the fact that the wavelengths of the studied phenomena are very small. It follows that extending low-frequency methods using finite element calculation would require an unreasonable number of degrees of freedom. Actually we use an alternative approach named the Variational Theory of Complex Rays, a dedicated approach for the calculation of medium frequency vibration problems, initially introduced in Ladevèze [8]. The objective of this paper is to describe the capabilities of this approach for the study of pyrotechnical propagating waves in launchers.

2 Resolution of the transient dynamic problem

The problem to solve is a transient dynamic problem under the assumption of small perturbations. The structure is defined in the space domain $\Omega$ and its boundary is $\partial \Omega$. At each time $t$ of the interval $[0, T]$, the structure is subjected to the following solicitations:

- a displacement field $\mathbf{U}_d$ on a portion $\partial_1 \Omega$ of the boundary $\partial \Omega$,
- a force density $\mathbf{F}_d$ on the portion $\partial_2 \Omega$ of $\partial \Omega$ which is the complementary part of $\partial_1 \Omega$,
- a force density $\mathbf{f}_d$ on the whole domain $\Omega$.

The transient dynamic problem can be written as a global variational problem over the frequency-space domain. The Fourier transform is applied to every time-dependent quantities, yielding frequency-dependent functions:

$$\hat{f}(\omega) = \int_{-\infty}^{+\infty} f(t) e^{-i\omega t} dt$$

(1)

Putting the accent on the displacement, the reference problem can be rewritten as: find $\hat{\mathbf{U}}(M, \omega)$, with $M \in \Omega$, such that:
\[ \vec{U}_{\partial \Omega} = \vec{U}_d \]
\[ \forall \omega \in \mathbb{R}, \forall \hat{U}^* \in U_0, \]
\[ \int_{\Omega} \left\{ -\rho \omega^2 \hat{U} \hat{U}^* + (1 + i \eta) \text{Tr}[K \epsilon(\hat{U}) \epsilon(\hat{U}^*)] \right\} d\Omega = \int_{\Omega} \hat{F} d\Omega + \int_{\partial \Omega} \hat{F}_d \hat{U}^* d\Omega \]

The constitutive relation is:
\[ \sigma = K \epsilon(U) + \eta K \dot{\epsilon}(U) \] (3)

\( U_0 \) is the subspace of \( U^{[0,T]} \) associated with a zero value of \( \vec{U}_d \) on the boundary \( \partial \Omega \). \( K \) is the Hooke’s operator, \( \sigma \) is the stress operator, \( \epsilon \) is the strain operator with \( \epsilon(U) = [\text{Grad}(U)]_{\text{sym}} \), and \( \rho \) is the density which is assumed constant. In the present work, damping is introduced classically in terms of the frequency with \( \eta > 0 \), so the reference problem has a unique solution. More complex constitutive relations could also be taken into account.

For a given \( \omega \), the previous problem is a forced vibration problem whose solution can easily be shown to be unique. Finally, one must calculate the frequency response function \( \hat{U}(M, \omega) \) of the system for \( \omega \in [0, \omega_c'] \).

The present approach considers a partition of the studied frequency range \( [0, \omega_c'] \) into two parts: a low-frequency part \( [0, \omega_c] \) and a medium-frequency part \( [\omega_c, \omega_c'] \). The Variational Theory of Complex Rays (VTCR) is a suitable computational method for the medium-frequency range \( [\omega_c, \omega_c'] \).

3 Outline of the VCTR

The Variational Theory of Complex Rays (VCTR), introduced in Ladevèze [8], is a dedicated approach for the calculation of medium frequency vibration problem. The state-of-the-art can be found in Ladevèze and Riou [13]. It can be summarized in two major points:

– the first characteristic of the VCTR is the use of a new variational formulation of the forced vibration problem which enables one to use a priori independent approximations within each substructure.

– the second characteristic of the VTCR is the introduction within each substructure of two scale approximations \( \hat{U}^h \) with a strong mechanical meaning, defined by identifying three zones: the interior zone, the edge zone and the corner zone. For example, in the neighborhood of a point \( X \) of the interior zone, the solution is assumed to be properly described locally as the superposition of an infinite number of local vibration modes which can be written in the following manner:

\[ \hat{U}^h(X, Y, P) = W^h(X, Y, P) e^{i \omega k P Y} \] (4)

where both \( X \) and \( Y \) represent the position vector, \( X \) being associated with slow variations and \( Y \) with rapid variations. More precisely, the terms related to the position vector \( X \) vary slowly when \( X \) moves along the structure, whereas the terms related to the position vector \( Y \) vary rapidly when \( Y \) moves along the structure. \( P \) is a vector characterizing the local vibration mode and \( k^2 = (1 + i \eta) \). In order for these local modes \( \hat{U}^h, \sigma^h \) to be admissible, they must exactly verify the dynamic equations, i.e. the dynamic equilibrium equation and the constitutive relation. Thus we get some properties of \( P \). All directions of propagation are taken into account and unknowns are discretized amplitudes with relatively long wavelength. Examples of local modes are shown in Figure 1.

The capabilities of the method have been demonstrated on complex assemblies of plates (Ladevèze et al. [9]). Comparisons with industrial finite element codes showed that the VTCR is able to predict the response at a very low cost. The method was extended to shells in Riou et al. [12]. Each development of the VTCR was implemented in the dedicated software CORAY MF (COmplex RAYs for Medium Frequencies) for the study of medium frequency forced vibration problems.
In our proposed approach for the transient dynamic response analysis, the Frequency Response Function (FRF) over a very wide frequency range needs to be calculated. Therefore the ability to use the VTCR over a relatively wide frequency range is advantageous (Ladevèze and Riou [13]). The idea is to introduce a two-scale approximation in terms of \( \omega \) and distinguish slow and fast coefficients varying with respect to the frequency. An *ad hoc* strategy then uses the mean quantities of the solution over the frequency band and their complementary part in order to predict the behavior of the quantities of interest.

4 Application of the proposed method

The proposed method has already been applied to a plate subjected to a moving impact loading with a speed of 7100 m.s\(^{-1}\), (Ladevèze and Chevreuil [14]). This approach is now being applied on assemblies of orthotropic shells in order to simulate the pyrotechnical waves within space structures. The last example is the pyrotechnical test which consists of a structure made of a honeycomb composite cone (ACU) and an aluminium cylinder subjected to a moving impact loading. Coupling between out-of-plane, pressure and shear waves induced by the curvature within orthotropic shells is taken into account.

![Simulated structure](image)

Figure 2 – Simulated structure

For the prediction of the FRF over the medium frequency range by the VTCR, the structure was divided into 48 substructures and 64 interior modes and 16 modes per edge were used for each substructure.

Références


Non-intrusive coupled global/local analysis of localized plasticity problems

Lionel GENDRE*, Olivier ALLIX1, Pierre GOSSELET1, François COMTE2

1 LMT-Cachan (ENS Cachan/CNRS/UPMC/PRES UniverSud Paris)
61, av. du Président Wilson – F-94230 Cachan – FRANCE
* gendre@lmt.ens-cachan.fr
2 Snecma Villaroche – Rond-point René Ravaud – Réau – F-77550 Moissy-Cramayel – FRANCE

Abstract
This paper presents a computational strategy to solve structural problems featuring nonlinear phenomena that occur within a small area of interest, while the rest of the structure retains a linear elastic behavior. Two finite element models are involved: a global linear model of the whole structure, and a local nonlinear “submodel” meant to replace the global model in the area of interest. An iterative exchange technique is then used to perform this replacement in an exact but non-intrusive way, which means the model data sets are never modified and the computations can be carried out with standard finite element software. Several ways of exchanging data between the models are investigated and convergence rates are presented on simple academic cases.

1. Introduction
In the aircraft industry, it is a common task to perform a finite element analysis on a complex structure that mostly evolves in a linear elastic way, but exhibits confined plasticity (or other nonlinear phenomena) in a small critical region. Such an analysis is usually carried out using a one-way submodeling technique, consisting in a global linear computation followed by local nonlinear “zooms” centered on critical zones and driven by the global displacements (as presented by Kelley in [6]) or stresses (Jara-Almonte et al., [5]). Although numerically efficient, those methods have a strong limitation: since they ignore the global influence of local plasticity, they cannot assess phenomena such as large-scale stress redistributions, and tend to introduce uncontrolled errors due to the inaccuracy of local boundary conditions.

On the other hand, several nonlinear multiscale strategies have been recently designed, such as the micro/macro approach [4,7] or the works of Cresta et al. [2]. Those methods provide robust and efficient ways to solve large nonlinear problems, but they suffer from a drawback: they are intrusive with respect to traditional finite element (FE) formulations. In other words, they cannot be carried out using commercial FE software and industrial model data files, and it would be long and difficult to implement them into an industrial environment.

This paper presents a non-intrusive and exact analysis strategy for finite element problems with localized plasticity. Much like submodeling techniques, it involves two FE models and can be carried out using standard FE software; however, it is built around an iterative exchange procedure designed to converge to the exact FE elastic-plastic solution. A brief description of the method and some results are proposed here.

2. Essential ideas
2.1 The elastic-plastic reference problem
Let us consider an elastic-plastic problem, where all plasticity is supposed to be confined within a small region ΩI of the structure, called the domain of interest (see Figure 1). In the remaining region ΩC, called the complement domain, we assume the constitutive law is purely linear elastic. Moreover, small perturbation theory is used; therefore the only source of nonlinearity is the plasticity in ΩI.
2.2 Global and local finite element models
As explained before, we intend to solve the elastic-plastic problem with two FE models. The first one, known as the global model, describes the whole structure and is purely linear elastic. We assume it to represent the complement domain's mechanical behavior accurately, however there can be several inaccuracies in the domain of interest: plasticity may occur, the FE mesh may be too coarse with respect to the expected gradients, and some small geometric details such as holes or cracks may be absent from the mesh.

Therefore, a local model is used to provide an accurate representation of the area of interest, with a “correct” mesh and an elastic-plastic constitutive law. The two meshes are assumed to match on \( \Gamma \), the boundary between the two domains, but can be completely different inside \( \Omega^I \) as shown on Figure 1.

Finally, the reference mesh is the mesh one would obtain by substituting the local mesh into the global mesh, and the reference solution is the FE solution to the elastic-plastic problem discretized onto the reference mesh.

![Figure 1: Global (left), local (center) and reference (right) FE models. The dark area is elastic-plastic.](image)

2.3 Solution substitution
In many global/local approaches, the solution is searched as the sum of the global solution and a local enrichment term (as in the works of Ben Dhia [1], Duester et al. [3] or Mao & Sun [8]). This is not the case here, and we define the global-local solution by substituting the local FE solution (with superscript \( L \)) into the global solution (\( G \)). Denoting displacements by \( u \) and stresses by \( \sigma \), this is written:

\[
\begin{align*}
    s(\vec{x}) &= (u(\vec{x}), \sigma(\vec{x})) \\
    &= \begin{cases} 
        s^L(\vec{x}) & \text{if } \vec{x} \in \Omega^I \\
        s^G(\vec{x}) & \text{if } \vec{x} \in \Omega^C 
    \end{cases}
\end{align*}
\]

There are two advantages in doing this. First, the two models can be defined quite independently: there is no compatibility constraint inside \( \Omega^I \). In other words, several geometry, mesh and/or constitutive law modifications can be introduced into the local model, and their exact influence on the whole solution can be assessed without changing anything in the global model, which could be helpful during design processes. Second, this form enables to handle local nonlinearity at the level where it actually occurs, ie. within the local model ; Cresta's works [2] suggest that this idea can save computational costs compared to expensive global iterations.

2.4 Iterative exchange technique
Finally, an iterative exchange technique is used to link both models and to have the global-local solution (defined by (1)) converge to the reference solution (defined in Section 2.2). The method is initialized by a global analysis that can be used to estimate the location of plastic areas. Then each iteration goes as follows:

1. **Local analysis**: a nonlinear analysis is performed on the local model, with part of the previous global solution specified as a boundary condition on \( \Gamma \) (displacement, traction or mixed).

2. **Residual computation**: the residual, a load vector measuring the discontinuity of the global-local solution across \( \Gamma \), is formed. If its magnitude is “too large”, we then proceed to the

3. **Global correction**: the residual is injected into the global problem as an additional load, in order to have the global model deform as if local details were present ; after updating the global solution, the process is repeated from Step 1 until the residual is small enough.

3. Discussion and results
According to the type of boundary condition chosen for the local analysis, the implementation and convergence rate of the method can be very different. Two possible choices are discussed below.
3.1 Displacement-based zooming

Displacement-based zooming is obtained when the global displacement field is specified as a boundary condition on $\Gamma$ during the local analysis. Assuming this choice, once Step 1 is complete, the global-local solution (defined by (1)) verifies every equation of the reference problem, except traction equilibrium on $\Gamma$. Therefore the residual (Step 2) is defined as the virtual work of global-local traction discontinuity across $\Gamma$,

$$ r^G = -\int_\Gamma (\sigma^G u^C + \sigma^L u^L) \cdot N^G d\Gamma \tag{2} $$

To avoid stress interpolation errors, this vector is directly assembled from nodal reaction forces on $\Gamma$ (conjugate to the prescribed displacements). Then global correction (Step 3) can be simply performed by solving an additional global problem loaded by the residual alone, and then updating the global solution:

$$ u^G \leftarrow u^G + [K^G]^{-1} r^G \tag{3} $$

where $K^G$ is the global stiffness matrix or operator. This approach is similar to the “Iterative Global-Local” technique proposed by Whitcomb [9] and is basically a modified Newton method formulated on surface traction equilibrium. Therefore, it tends to converge slowly when plasticity occurs because $K^G$ is then too stiff. This can be improved by replacing $K^G$ with a tangent operator $K^G$ obtained from tangent moduli:

$$ u^G \leftarrow u^G + [\tilde{K}^G]^{-1} r^G \tag{4} $$

To avoid modifying the global model, $\tilde{K}^G$ is never assembled. Instead, since $\tilde{K}^G$ and $K^G$ differ only by a few terms (actually by a small rank matrix), (4) is addressed by an iterative algorithm that uses $[\tilde{K}^G]^{-1}$ as a preconditioner by sending load vectors to the global problem. This is demonstrated on Figure 2, where the convergence rates of three different operators are compared on a simple 1D test case – the initial global operator, a “true” tangent operator which can be proved to be optimal (ie. fastest possible), and an approximate tangent operator. A significant improvement in convergence rates can be noticed (4 iterations with the optimal operator versus 30 with the basic method, to get a $10^{-4}$ residual error), although the computational cost of the extra calls must also be assessed.

![Global: 5 elements Local: 20 elements at the left of the load (circled zone)](image)

**Figure 2:** 1D test case (left) and convergence rates for the displacement-based zooming (right), using three different operators for global correction

3.2 Mixed zooming

Even with an optimal global correction step, it takes several iterations to link the two models properly: as shown in Cresta’s works [2], a displacement boundary condition isn’t the most relevant choice from a mechanical point of view. To achieve a better convergence rate, one can use a mixed Robin boundary condition on $\Gamma$:

$$ (\sigma^L u^L + \sigma^G u^C) = \tilde{K}^{G \rightarrow L}(u^G - u^L) \tag{5} $$
where $K^{G\rightarrow L}$, the FE discretization of $\Delta^{G\rightarrow L}$, is a positive matrix defining an additional stiffness on $\Gamma$. This kind of condition can be very accurate if $K^{G\rightarrow L}$ is a good representation of the stiffness of the complement domain $\Omega^C$; in particular, one can prove that if $K^{G\rightarrow L}$ is the exact Schur complement of $\Omega^C$, then the local solution equals the reference solution after the first local analysis, no matter how inaccurate the global solution is. This may be particularly interesting for applications where the only quantities of concern are in the local model.

With this method, neither the displacements nor the tractions are continuous across $\Gamma$ at the end of Step 1. Therefore the residual must measure both discontinuities, and contains an additional term with respect to (2):

$$r^{G} = \int_{\Gamma} \left[ - \left( \sigma^{G} \epsilon^{G} + \sigma^{L} \epsilon^{L} \right) + K^{L-G} (\mathbf{u}^{L} - \mathbf{u}^{G}) \right] \cdot N^{G} \, d\Gamma$$  \hspace{1cm} (6)

$K^{L-G}$, the FE discretization of $\Delta^{L-G}$, is another parameter similar to $K^{G\rightarrow L}$. Global correction is then performed as previously, by applying the residual as an additional global load. It was proven that if $K^{L-G}$ is the exact Schur complement of the local model, then the optimal global correction operators are the same as those defined for the displacement-based zoom. Thus, if $K^{L-G}$, $K^{G\rightarrow L}$ and $K^{G}$ all match their optimal values, then the whole global-local solution reaches the reference solution in just one iteration.

Of course, those Schur complements cannot be computed exactly at realistic costs for large industrial models. Therefore we have tested algebraic approximations to them; when using only the matrix blocks relative to the boundary ($\Gamma$) degrees of freedom, it only took 3 global computations to attain complete convergence for the 1D test case on Figure 2, and only 2 iterations to get an accurate local solution (with a relative error near $10^{-4}$).

### 3.3 More results

As of March 2008, the strategy is being implemented with Abaqus/Standard as a master script submitting and post-processing analysis jobs. Results on 2D and/or 3D test cases will be presented at the conference.

### Acknowledgment

This work is part of the MAIA-MM1 research program and is supported by Snecma.

### References


Re-triangulation of existing surface meshes with high curvatures

Antonio Carlos MIRANDA, Luiz MARTHA*, Paul WAWRZYNEK, Anthony INGRAFFEA

*Department of Civil Engineering and Tecgraf – Computer Graphics Technology Group
Pontifical Catholic University of Rio de Janeiro (PUC-Rio), Rio de Janeiro RJ 22453-900, Brazil
lfm@tecgraf.puc-rio.br

Abstract
This work describes an automatic algorithm for unstructured mesh regeneration on arbitrarily shaped three-dimensional surfaces. The arbitrary surface may be: a triangulated mesh, a set of points, or an analytical surface (such as a collection of NURBS patches). To be generic, the algorithm requires the implementation of three abstract methods. The first, given a point location, returns the desired characteristic size of a triangular element at this position. The second method, given the current edge in the boundary contraction algorithm, locates the ideal apex point that forms a triangle with this edge. And the third method, given a point in space and a projection direction, returns the closest point on the geometrical supporting surface. This work also describes the implementation of these three methods to re-mesh an existing triangulated mesh that might present regions of high curvature. In order to test the efficiency of the proposed algorithm of surface mesh generation and implementation of the three abstract methods, results of performance and quality of generated triangular element examples are presented.

1. Introduction
This paper describes an algorithm for generating triangle finite-element meshes on surfaces of arbitrary shape and curvatures. It is an extension of a previously proposed algorithm (advancing-front technique) for generating unstructured meshes in three-dimensional [1, 2] and two-dimensional [3, 4] domains. Algorithms have been described in the literature previously that generate 3D surface meshes by re-meshing existing triangulations [5-7], or by using analytical surface descriptions for geometrical support [8-10]. The present algorithm generalizes the type of geometrical support that may be used by making use of three generic functions that abstractly represent the supporting surface. In a particular case, this paper also describes an implementation of the three generic methods for re-meshing existing surface triangulations and demonstrates the quality of the generated meshes.

To facilitate a smooth transition between regions and regions of high curvatures, the present algorithm employs an octree data structure. However, unlike some authors, e.g. Rassineux [11], who use a quadtree/octree procedure to generate internal nodes prior to element generation, here it is used to provide local policies used to define the discretization of the surface mesh and to define the sizes of triangular elements to be generated during the advancing-front procedure. The authors feel that this approach tends to provide better control over the quality of the generated mesh and to decrease the number of heuristic, clean-up procedures.

2. Description of the algorithm
The input data for the present algorithm is a polygonal description of the boundary of the surface patch to be meshed and a 3D supporting surface that is abstractly represented by three generic methods defined as follow:

1st Method - Given a point location, the method returns the desired characteristic size of an ideal equilateral triangular element at this position. The length of the triangle’s side is considered as its characteristic size.

2nd Method - Given the current base edge in the boundary contraction algorithm, the method locates the ideal apex point that forms a new triangle. The method has two input arguments: the height of the candidate equilateral triangle and a unit vector in the edge perpendicular direction. This unit vector is a “surface intersection direction” that defines a plane perpendicular to the base edge at its mid point. These arguments are used to determine an optimal triangle apex point location.

1
3rd Method - Given a point in space, the method returns the closest point on the geometrical supporting surface. The method also receives a surface projection vector as additional information. This method is used only in the final stage of the surface mesh generation for local mesh improvement.

2.1 Advancing-front procedure

In this algorithm, as in its ancestors [1–4], the advancing-front process is divided into two phases to ensure the generation of valid triangulations. In the first phase, a geometry-based element generation is pursued to generate elements of optimal shapes. After this ideal phase is exhausted and no more optimal elements can be generated, a topology-based element generation takes place, creating valid, but not necessarily well shaped, elements in the remaining region. The required steps for the advancing-front procedure are as follows.

2.1.1 Front contraction (geometry-based element generation)

Ideally, the entire mesh will be generated in the geometry-based phase. In this phase, for each base edge on the advancing front, the following is performed:

- The optimal location \( N_1 \) for the vertex of an equilateral triangle to be formed is determined employing the 1st and 2nd Methods. Using the base edge middle point \( M \) as input, 1st Method returns the target triangle characteristic size (the length of the equilateral triangle side) at this point location. With this size, the height of the candidate triangle is obtained. Then, using the Normal x Tangent vector at the middle point \( M \) as the required unit vector (surface intersection direction) in the edge perpendicular direction and the triangle height, 2nd Method returns the desired \( N_1 \) location on the support surface.

- If no existing node is inside the optimal region, a new node is inserted at the optimal location \( N_1 \) and an element is generated using this node. If only one node exists in the region, this node is used to generate the element.

- Additional geometric checks are performed to ensure that the edges of the new triangular element do not intersect any existing edge of the advancing front and that the new triangle apex does not lie inside any other existing triangle. In both cases, the new element is rejected. These checks are not trivial in 3D space. However, they are performed in a local 2D system on the plane of the new triangular element, avoiding complex geometry checks in 3D space. The new edges and adjacent elements edges are transformed to the local 2D system on the plane of the new triangular element and all checks are made in this local system.

2.1.2. Front contraction (topology-based element generation)

The objective of this phase of the algorithm is to force the generation of valid triangles, even if a measure of the quality of the shape of the element does not fall within the allowable range used in the previous phase. The topology-based element-generation phase starts when a boundary edge fails twice in trying to generate an optimal element. The list of rejected edges of the previous phase is transformed into a list of active edges and, similarly to the geometry-based phase, a list of rejected edges is created for edges that eventually fail in generating valid triangles.

In the topology-based element-generation phase, any node close to the current base edge is selected and stored in a priority queue of candidate nodes. The node that has the maximum included angle with respect to the base edge is chosen for the generation of the new triangle. If the edges of this triangle do not intersect any other edge of the current advancing front, the element is created and the boundary is contracted accordingly. The topology-based phase ends when the lists of active and rejected edges are empty. This phase always generates a valid, even though non-optimal, mesh.

2.2 Local mesh improvement

A smoothing technique is used to improve mesh quality by relocating nodes within a patch. A general formulation for this technique is given by equation (1), which is a generic form of a weighted Laplacian function. The smoothing procedure is repeated twice for all internal nodes. In general, the smoothing of surface mesh equation will move nodes to a position off of the surface. The 3rd Method is employed after each smoothing procedure as a "pull back" operation, moving the target node back to the geometric supporting surface. In theory, Laplace smoothing and the "pull back" procedure can cause mesh "folding". A check is made and a node is not moved if doing so will cause an invalid mesh. In practice, there has been no need to enforce this restriction for any of the test cases.
3. Application: re-meshing
To demonstrate the performance and the efficiency of the proposed surface meshing algorithm, this section describes an implementation of the three generic methods for re-meshing existing surface triangulations.

3.1. First generic method implementation
The first generic method of the proposed surface mesh generation obtains the desired characteristic size of a triangular element given the position of a point on the surface. It is important that the algorithm presents a smooth transition between regions with elements of highly varying sizes. Therefore, an auxiliary background data structure (octree) is used to store the distribution of characteristic triangle sizes in space.

In the current application the size of the leaf cells in a region of a domain are used as the size of the desired characteristic element size in this region. The creation of the background octree from an input surface follows five steps: (a) Octree initialization based on given boundary edges. (b) Refinement to force maximum cell size. (c) Refinement to provide minimum size disparity for adjacent cells. (d) Refinement to account for surface curvatures. (e) Refinement to provide minimum size disparity for adjacent cells for the updated octree.

3.2. Second generic method implementation
Two different approaches to this task are described in the literature. Lohner [6] tries to find a host triangle, where the ideal node lies, using a neighbor-to-neighbor search. If this fails, octrees are employed. Finally, if this approach fails again, a brute force search over all the surface elements is performed. Carlos et al. [5] use a procedure that guarantees that the new triangle sides have exactly the desired length. This procedure considers the equation of a sphere, and finds the intersections between the sphere and the mesh. The approach used in this work is similar to this one, employing the circle equation in a 2D coordinate system.

3.3. Third generic method implementation
The third generic method returns the point on the geometric supporting surface that is closest to a given point near to, but not necessarily on, the surface. During the nodal smoothing phase, points may be moved off the surface. This method is used to move the points back to the surface. As input, the method receives the current node location and its current normal vector. The implementation of this function is quite simple: First, a local search based on the auxiliary R-Tree structure creates a list of all neighborhood triangles of the geometric supporting surface around the given node. Second, the node is projected on each neighbor triangle plane, using the given projection direction. Finally, the selected new node location is the one that lies inside one of the neighbor triangles.

3.4 Example: performance mesh and quality
This section presents additional examples of finite-element meshes generated on 3D surfaces using the proposed algorithm. The main objectives of this section are to estimate the expected performance of the surface mesh generation algorithm and assess the quality of generated triangular elements.

Four examples of triangle surface meshes to be re-meshed with the present algorithm. The first one is an open cylinder. The second is a geological salt-dome surface. The third is a geological surface with four internal loops. The final example is a surface with many irregularities in its curvatures.

Figure 1(a) shows a plot of the elapsed processing time as a function of the number of elements generated. Using a fitting equation, one may infer that the algorithm’s performance is close to $O(N \log N)$. The quality of generated meshes is presented in the form of a histogram such as the one shown in Figure 1(b). In this histogram, the horizontal axis corresponds to the $\gamma^p/\gamma^*$ quality measure in intervals represented by triangular shapes that are shown below the histogram. The vertical axis corresponds to the percentage of elements in each interval of the quality measure. These results demonstrate that the proposed algorithm generates meshes with good quality for the great majority of elements. Note that the quality of meshes in all examples is below 1.5, an indication of very well shaped elements.

4. Conclusion
This paper has described an algorithm for regenerating triangle finite-element meshes on surfaces of arbitrary shape and with varying curvature. The input data for the present algorithm is a generic supporting surface and a polygonal description of the boundary on the surface patch to be meshed. The supporting surface is represented abstractly by three generic methods: The first, given a point location, returns the desired characteristic size of a triangular element at this position; The second, given the current edge in the boundary contraction algorithm, locates the ideal apex point that forms a triangle with this edge; and The third, given a point in space and a projection direction, returns the closest point on the geometrical supporting surface.
These three generic methods are used in the proposed two-pass advancing-front procedure to generate elements on the supporting surface. In the first pass, elements are generated based on geometrical criteria, which produces well-shaped elements. In the second pass, triangular elements are generated based solely on topology criteria.

The three generic methods were implemented to regenerate triangle mesh surfaces. This approach is very useful when a mesh needs to be refined, coarsened or improved. Some examples have demonstrated the quality of the generated meshes and the importance of considering surface curvatures in local mesh refinement.

Figure 1 – (a) Generation times for re-meshing the example surfaces and (b) Histogram of element quality for re-meshing the example surfaces.

References

Mapping functions in the eight node elastodynamic infinite element with union shape function (EIEUSF)

Konstantin Savkov Kazakov *

*Structural mechanics dept., VSU "Luben Karavelov"
175' Suhodolska str., 1373 Sofia, Bulgaria
kazakov@vsu.bg

Abstract

Many engineering problems can be regarded as unbounded. In structural mechanics Soil-Structure Interaction problems can be modeled more efficiently using infinite elements on the same boundary of the studied system. Generally, the domain of the problem is divided into two parts: near and far field. The solution is based on finite elements to the near field and infinite elements for the far field.

This paper is devoted to the development of mapped infinite element formulations directly applicable in the Finite element method. Basic advantages of such elements are demonstrated and discussed. Computational techniques based on the infinite element are often called infinite element method. However the infinite element computational models of unbounded problems usually contain finite and infinite elements in conjunction. By reason of that, in the paper, this technique is called Infinite elements in the Finite element method.

The main aspects of the $C_1$ and the $C_n$ continuity along the finite/infinite element (artificial boundary) line in two-dimensional substructure Soil-Structure Interaction problems are also discussed in brief. In this class of models such a line marks artificial boundary between the near and the far field of the model.

An extension of dynamic infinite elements for transient dynamic analysis is also presented in the paper. The realistic simulation of the transient Soil-Structure Interaction in a time domain is compared with a formulation in frequency domain.

1. Introduction

The main purpose of the work is to demonstrate new form of serendipity type mapping functions, applicable in the formulation of the elastodynamic infinite element with union shape functions (EIEUSF). Recently, such an infinite element was proposed by the author in [2] and [3] and can be successfully used in wave propagation computational models. The basic idea is the construction of union shape basis, including union shape functions. The union shape function, corresponding node $i$ can be written as

$$N_i(x, z) = \sum_{q=1}^{m} N_{iq}(x, z, \omega) = M(x, \xi) L_i(\eta) R e W(\xi).$$

Here, $N_{iq}(x, z, \omega)$ are the functions included in the wave basis, $M(x, \xi)$ is mapping function, assuring the geometrical transformation if local to global coordinates. Functions $L_i(\eta)$ and $R e W(\xi)$ give the distribution of the basic variable in local coordinate $\eta$ and $\xi$ directions, respectively. $L_i(\eta)$ are Lagrange polynomials, linear or quadratic, in the case of two or three nodes in $\eta$ direction, respectively. Functions $R e W(\xi)$, called union wave functions, given in the form
\[ \text{Re} W(\xi) = \sum_{q=1}^{m} A_q \text{Re} W(\xi, \omega_q). \]  

\[ \text{Re} W(\xi, \omega_q) = \cos \left( \frac{i \omega_q}{c_p} \xi \right) \exp(\alpha \xi), \]  

where \( \omega_q, c_p, \) and \( \alpha \) are frequencies, wave velocities (for \( P \)-waves) and attenuation factor, respectively. Such functions can be constructed also for \( S \)-waves and Rayleigh waves, using velocities \( c_s \) and \( c_r \). The coefficients \( A_q \) can be treated as weight wave coefficients. In more details, the idea of the elastodynamic infinite element with union shape functions (EIEUSF) is given in the works of the author [2] and [3]. This element can directly be used in the FEM concept, Hughes [1].

In the paper a new form of the function \( M(x, \xi) \), based on two mid-points is demonstrated and discussed in brief. In general form, the mapping of local to global coordinates along the infinite element domain can be written as

\[ x = T \xi \quad \text{or} \quad \{x\} = \{T\} \{\xi\} \]  

where \( x \) and \( \xi \) are vectors, containing components of the two coordinate systems. Such vectors can be one, two or three-dimensional. The matrix \( T \) is a matrix which accomplishes unique transformation of a point from the local to global domain. If reverse transformation exists than

\[ \{\xi\} = \{T\}^{-1}\{x\} \]  

An alternative geometrical transformation can be written in the form

\[ x = \sum_{i=1}^{n} M_i x_i, \quad y = \sum_{i=1}^{m} M_i y_i \quad \text{and} \quad z = \sum_{i=1}^{r} M_i z_i \]  

where \( M_i \) are mapping functions, and \( n, m \) and \( r \) are the number of node points used respectively in every one of the directions. The two forms give identical transformation in many cases. The mapping in much number of infinite element formulations is based on the special form of equations (5), Zhao and Valliappan [5].

\[ x = \sum_{i=1}^{4} M_i x_i, \quad y = \sum_{i=1}^{4} M_i y_i \quad \text{and} \quad z = \sum_{i=1}^{4} M_i z_i. \]  

In this paper the transformation given by equations (7) is used. This form leads to some conveniences in the formulation.

2. Mapping functions

First one dimensional mapping functions are proposed in the work of Zienkiewicz [6]. The proposed in this paper mapping in global \( x \) (local \( \xi \)) direction can be given in two forms: mapping without “pole” and mapping with “pole”. Only the first form will be demonstrated here. Corresponding global and local coordinates of the nodes, see figure 1, are

\[ J(x_l) \leftrightarrow J(-1), \quad K(x_k) \leftrightarrow K\left(-\frac{1}{3}\right), \quad L(x_k) \leftrightarrow L\left(\frac{1}{3}\right) \quad \text{and} \quad M(x_M = \infty) \leftrightarrow M(+1) \]
The mapping of local to global coordinates is based on the equations (6), depending on the number of used mapping functions. Here equations

\[ \sum_{i=1}^{4} M_i x_i = x \quad \text{or} \quad x = M_1 x_J + M_2 x_K + M_3 x_L + M_4 x_M \]  

are used.

The functions \( M_i \) must be chosen in such a way that a point from the local domain \( \xi \in [-1;1] \) is transformed to a point in the global domain \( x \in [x_J;\infty) \) and every local node coordinate is mapped to the correct value of the same node in global coordinates. Here, serendipity mapping functions are given, written in two different forms

- **Variant I**
  \[
  M_1 = \frac{-2\xi}{1-\xi}, \quad M_2 = (1+\xi) \left(\frac{1}{3} - \frac{\xi}{2}\right) \left(\frac{1}{3} + \frac{\xi}{2}\right), \\
  M_3 = \frac{3}{4} (1+\xi), \quad M_4 = (1+\xi) \left(\frac{1}{3} - \frac{\xi}{2}\right) \left(\frac{1}{3} + \frac{\xi}{2}\right),
  \]
  \( \xi = -1, \quad \xi = -1/3, \quad \xi = 1/3, \quad \xi = 1 \)

- **Variant II**
  \[
  M_1 = \frac{-2\xi}{1-\xi} + \frac{9}{8} (1+\xi) \left(\frac{1}{3} + \frac{\xi}{2}\right), \quad M_2 = (1+\xi) \left(\frac{1}{3} - \frac{\xi}{2}\right) \left(\frac{1}{3} + \frac{\xi}{2}\right), \\
  M_3 = \frac{3}{4} (1+\xi), \quad M_4 = (1+\xi) \left(\frac{1}{3} - \frac{\xi}{2}\right) \left(\frac{1}{3} + \frac{\xi}{2}\right).
  \]

Substituting mapping functions from **Variant I** and **Variant II** in equations (8) leads to equations, respectively.
\[
x = -\frac{2\xi}{1-\xi} x_j + (1+\xi) \left( \frac{1}{3} - \frac{\xi}{1+\xi} \right) x_k + \frac{3}{4} (1+\xi) x_L + \left(1+\xi\right) \left( \frac{1}{3} - \frac{\xi}{1+\xi} \right) x_m
\]

or

\[
x = \left[ -\frac{2\xi}{1-\xi} + \frac{9}{8} (1+\xi) \left( \frac{1}{3} + \xi \right) \right] x_j + (1+\xi) \left( \frac{1}{3} - \frac{\xi}{1+\xi} \right) x_k + \frac{3}{4} (1+\xi) x_L + \left(1+\xi\right) \left( \frac{1}{3} + \xi \right) x_m
\]

(11)

Suppose now that \( JK = KL = a \), and taking into consideration that

\[ x_k = x_j + a , \]

\[ x_L = x_j + 2a = x_K + a , \]

using eq. (12) the results are

- when \( \xi = -1 \) \( x = x_j \),
- when \( \xi = -1/3 \) \( x = x_j + a = x_K \),
- when \( \xi = +1/3 \) \( x = x_j + 2a = x_K + a = x_L \),
- when \( \xi = +1 \) \( x = x_M = \infty \).

(13)

The mapping functions, given by eqs. (9), are appropriate in the case when \( JK = KL \). In this case the geometrical parameter \( JK = KL = a \) can be treated as geometrical characteristics of the infinite element. Such a condition does not lead to any computational and application restrictions. In the case when \( JK \neq KL \), eqs. (10), must be applied. The geometrical transformation, based on eqs. (9), does not lead to correct value at point \( \xi = 1/3 \). If the node at point \( x_L \) in global coordinates or \( \xi = 1/3 \) in local coordinates is omitted, such mapping functions can be used in a formulation with three nodes on \( x \) direction of the element. This point however, point \( x_L \), is indispensable to be used in the mapping. In the case of rapidly attenuating field variable function the parameter \( a \) must be chosen as a small number.

References