Finite Elements in Analysis and Design 86 (2014) 23–33

Contents lists available at ScienceDirect

Finite Elements in Analysis and Design

journal homepage: www.elsevier.com/locate/finel

Low intrusive coupling of implicit and explicit time integration schemes for structural dynamics: Application to low energy impacts on composite structures

T. Chantrait [a], J. Rannou [a,*], A. Gravouil [b,c]

[a] Onera – The French Aerospace Lab, 29 avenue de la Division Leclerc, 92320 Chatillon cedex, France
[b] Université de Lyon, CNRS Insa-Lyon, LaMCoS UMR5259, 20 Av Albert Einstein, FR69621 Villeurbanne cedex, France
[c] Institut Universitaire de France, France

ARTICLE INFO

Article history:
Received 4 December 2013
Received in revised form
13 March 2014
Accepted 25 March 2014
Available online 17 April 2014

Keywords:
Implicit dynamics
Explicit dynamics
Code coupling
GC method
Composite materials

ABSTRACT

Simulation of low energy impacts on composite structures is a key feature in aeronautics. Unfortunately it involves very expensive numerical simulations: on the one side, the structures of interest have large dimensions and need fine volumic meshes (at least locally) in order to properly capture damage. On the other side, explicit simulations are commonly used to lead this kind of simulations (Lopes et al., 2009 [1]; Bouvet, 2009 [2]), which results in very small time steps to ensure the CFL condition (Courant et al., 1967 [3]). Implicit algorithms are actually more difficult to use in this situation because of the lack of smoothness of the solution that can lead to prohibitive number of time steps or even to non-convergence of Newton-like iterative processes. It is also observed that non-smooth phenomena are localized in space and time (near the impacted zone). It may therefore be advantageous to adopt a multiscale space/time approach by splitting the structure into several substructures with their own space/time discretization and their own integration scheme. The purpose of this decomposition is to take advantage of the specificities of both algorithms families: explicit scheme focuses on non-smooth areas while smoother parts (actually linear in this work) of the solutions are computed with larger time steps with an implicit scheme. We propose here an implementation of the Gravouil–Combescure method (GC) (Combescure and Gravouil, 2002 [4]) by the mean of low intrusive coupling between the implicit finite element analysis (FEA) code *Zset/Zébulon* (Z-set official website, 2013 [5]) and the explicit FEA code Europlexus (Europlexus official website, 2013 [6]). Simulations of low energy impacts on composite stiffened panels are presented. It is shown on this application that large time step ratios can be reached, thus saving computation time.

1. Introduction

Low energy impacts can be very harmful for composite laminates used in the aerospace industry. They can actually cause significant damages (matrix cracking, delamination, fiber failure, etc.) inside the composite or on the side opposite to the impact. However, the residual print left on the impacted side can be almost undetectable to the naked eye. Induced damages can therefore lead to early failure of the structure while they can be unnoticed during a visual inspection, this is related to the concept of BVID (Barely Visible Impact Damage). Controlling such situations is essential for manufacturers. Numerical simulations of this phenomenon could be really helpful to orient and to rationalize test campaigns by the use of virtual testing as well as to understand scale effects. Various researches are led in the scientific community to simulate these impacts which is actually very difficult to carry out at industrial level. Implicit solvers can be used to deal with this type of problem with satisfactory results [7–9]. However non-smooth sources like contact, softening damage laws or cohesive zone models are often introduced in the models which can make convergence of implicit solvers difficult to achieve. Using explicit solvers to simulate low-energy impacts is an alternative way to handle this difficulty despite the non-dominance of high frequency terms or wave propagation in this context [1,2,10,11]. Explicit solvers are indeed more suitable to solve non-smooth problems. However, stability requires the use of very small time steps which linearly decrease with the characteristic length of the smallest mesh element. Moreover, very fine meshes are usually required (at least locally) to capture the non-linear phenomena occurring during an impact. This thus leads to a very large number of increments which can be prohibitive.

* Corresponding author.
E-mail addresses: Teddy.Chantrait@onera.fr (T. Chantrait), johann.Rannou@onera.fr (J. Rannou), Anthony.Gravouil@insa-lyon.fr (A. Gravouil).

http://dx.doi.org/10.1016/j.finel.2014.03.011
0168-874X/

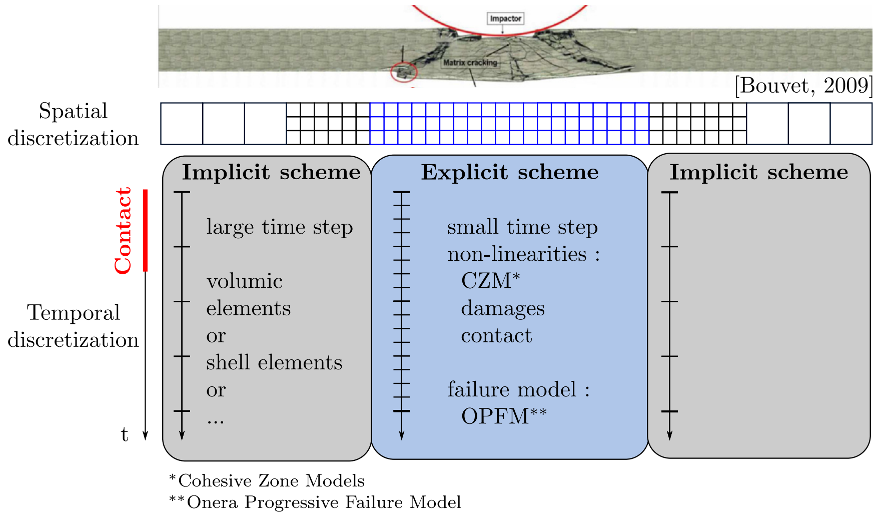

Fig. 1. Overview of a multiscale space/time approach applied in the context of low-energy impact on composite structures.

Note however that these non-linear phenomena occur on a very localized area around the impact point.

Adopting a space/time multiscale strategy thus appears to be advantageous to solve this kind of multiscale problems. This can be performed through domain decomposition where each sub-domain owns its time discretization. The purpose of this decomposition is to focus on numerical computation where non-linear phenomena appear [12]. Explicit resolution in the area close to the impact is required because of the lack of smoothness of the solution. However, on the complementary area where the solution is smoother, implicit integration is appropriate. Larger time steps can then be used to save CPU time. The present work is based on the GC method [4] and aims at coupling semi-industrial finite elements codes (FEA) *Zset/Zébulon*[1] and *Europlexus*.[2] Fig. 1 illustrates the computation strategy. It shows one section of an impacted plate and a typical mesh. This mesh is divided into two domains: an impacted domain (center) which is processed by the explicit code *Europlexus* with a fine time step and a complementary domain which is processed by the implicit code *Zset/Zébulon* with larger time steps. In addition, low intrusivity is a key feature of the implementation when industrial applications are aimed [13,14]. In the present work this goal has been achieved through `Python/NumPy` high level scripting so that no coding was required into the native programming languages of *Zset/Zébulon* and *Europlexus* to implement the coupling algorithm itself.

2. The algorithmic framework

We describe in this section the algorithmic framework of this study. Space and time discretization of structural dynamics problems are firstly presented. Both implicit and explicit algorithms are expressed in terms of velocity unknowns. This unusual form is useful to handle both implicit and explicit schemes equations within the GC domain decomposition framework (see Section 2.3) which requires continuity of interfacial velocity. Gravouil and Combescure [4] indeed showed that imposing interfacial velocity continuity is mandatory to ensure stability when coupling arbitrary Newmark schemes with different time steps. Domain decomposition method written under this constraint is then presented for two domains. Extension of domain decomposition with different time steps within each sub-domain through the GC method is finally described.

2.1. Space and time discretization of structural dynamics problems

We consider here the finite element discretization of the principle of virtual power which leads to the semi-discretized equilibrium system of Eqs. (1) for an undamped structure [15]:

$$\begin{cases} \forall t \in [t_0, t_f], & \mathbf{M}\ddot{\mathbf{u}}(t)+\mathbf{f}^{int}(\mathbf{u}(t)) = \mathbf{f}^{ext}(t) \\ \mathbf{u}(t_0)=\mathbf{u}_0, & \dot{\mathbf{u}}(t_0)=\dot{\mathbf{u}}_0 \end{cases} \quad (1)$$

where $\mathbf{M}$ is the symmetric definite-positive consistent mass matrix, $\mathbf{u}$ is the nodal displacements vector, $\mathbf{f}^{int}$ is the internal forces vector ($\mathbf{f}^{int}(t)=\mathbf{K}\mathbf{u}(t)$ for linear elasticity with $\mathbf{K}$ the stiffness matrix) and $\mathbf{f}^{ext}$ is the external forces vector. Single and double superposed dots over a quantity denote respectively its first and second time derivatives. The initial displacements and velocities vectors are respectively denoted as $\mathbf{u}_0$ and $\dot{\mathbf{u}}_0$. t_0 and t_f denote respectively the beginning and the end of the time domain of interest.

The system (1) is then discretized in time to be solved numerically. A lot of time integrators can be found in the literature, see for instance [16–19]. However, due to GC method constraints, the present work is restricted to Newmark schemes [20]. A Newmark temporal integrator is defined by two parameters γ and β and involves relations (2) among displacement, velocity and acceleration vectors from time t_n to time $t_{n+1}=t_n+\Delta t$. Δt is the time step, $n \in [\![0, n_{step}-1]\!]$ and $n_{step} \in \mathbb{N}^*$ is the number of time step:

$$\begin{cases} \mathbf{u}_{n+1} = {}^p\mathbf{u}_n + \beta \Delta t^2 \ddot{\mathbf{u}}_{n+1} \\ \dot{\mathbf{u}}_{n+1} = {}^p\dot{\mathbf{u}}_n + \gamma \Delta t \ddot{\mathbf{u}}_{n+1} \end{cases}. \quad (2)$$

with displacement predictors ${}^p\mathbf{u}_n$ and velocity predictor ${}^p\dot{\mathbf{u}}_n$ given by the following relations:

$$\begin{cases} {}^p\mathbf{u}_n = \mathbf{u}_n + \Delta t \dot{\mathbf{u}}_n + (1-2\beta)\dfrac{\Delta t^2}{2}\ddot{\mathbf{u}}_n \\ {}^p\dot{\mathbf{u}}_n = \dot{\mathbf{u}}_n + (1-\gamma)\Delta t^p \ddot{\mathbf{u}}_n. \end{cases} \quad (3)$$

where subscripted quantities $\mathbf{x}_n$ are the approximations of $\mathbf{x}(t_n)$ at time t_n.

Introducing Newmark relations (2) in Eq. (1) leads to the following system of equations as long as γ is not equal to zero which never happens in practice for stability reasons [15]:

$$\ddot{\mathbf{u}}_0 = \mathbf{M}^{-1}(\mathbf{f}_0^{ext} - \mathbf{f}^{int}(\mathbf{u}_0)) \forall n \in [\![0, n_{step}-1]\!],$$

[1] *Zset/Zébulon*is an implicit FEA code developed by Mines ParisTech, Onera and NW Numerics & Modeling, http://zset-software.com/

[2] *Europlexus* is an explicit FEA code developed by *Commissariat à l'énergie atomique* (CEA) and the Joint Research Centre (JRC) in Ispra, Italy.