VIII International Conference on Computational Plasticity
COMPLAS VIII
E. Oñate and D.R.J. Owen (Eds)

ON THE MODELING OF THE DYNAMIC CRACK PROPAGATION BY EXTENDED FINITE ELEMENT METHOD: NUMERICAL IMPLEMENTATION IN DYNELA CODE

I. Nistor⋆, O. Pantalé, and S. Caperaa

Laboratoire Génie de Production - Ecole Nationale d'Ingénieurs de Tarbes
47 av d'Azereix, 65016 Tarbes Cedex, France
e-mail: inistor@enit.fr, web page: http://www.enit.fr/

Key words: XFEM, Partition of Unity Method, Object-Oriented Prog., Crack Propagation

Summary. *The development of the computational techniques used in the analyzes of the dynamic fracture and their implementation in numerical codes takes a growing importance in last years. This interest derives from the necessity to predict the localisation of the initiations of cracks, when and in which direction this crack will propagate throw the material under dynamic loads. Several numerical approaches were proposed during few past decades in order to analyze discontinuous phenomena like cracks and shear bands occurring in structures under quasi-static or dynamic loads. In this paper we present the numerical implementation of the Extended Finite Element Method (XFEM), one of the latest approach developed in order to model the dynamic discontinuities. The crack representation in XFEM is based on the enrichment of the classical displacement-based finite element approximation through the framework of partition of unity method. In this approach, a crack is modeled introducing additional degrees of freedom to the nodes whose nodal shape function support intersects this one.The explicit dynamic FEM code DynELA developed in the LGP in Tarbes using an object-oriented framework is used to support the implementation of the XFEM as a new module called DynaCrack.*

1 XFEM DESCRIPTION FOR DYNAMIC CRACK ANALYSIS

Several approaches were proposed during last years in the field of computational fracture methods in order to avoid the re-meshing step in crack modeling. One of them is the eXtended Finite Element Method (XFEM): a new method based on Partition of Unity framework [1], developed firstly by Moes and al.[3]. The main idea of this method is that the approximation space spanned by standard finite element shape function is enriched by products of the standard basis function with special enrichment functions. Two types of functions were considered for quasi-static crack growth [3]: the Heaviside step function when the crack completely cuts the elements and Westergaard-type asymptotic functions for elements containing crack-tips. In our approach only the Heaviside step function was used for enriching discontinuous fields and the crack-tip passes from edge to edge. Depending on crack direction, Heaviside function H takes the value $+1$ for the points above the crack and -1 for them below the crack. If the support of nodal shape function for a node is intersected by a crack, the node is enriched with additional degrees of freedom (*dof*). The discontinuous displacement field for a N nodes mesh, including N^* enriched nodes, is approximated by:

$$u^h(X) = \sum_{I\in N} \phi_I(X)\, u_I + \sum_{I\in N^*} \phi_I(X) H(X)\, a_I \tag{1}$$

where ϕ_I represent the classical shape functions, u_I the classical *dof* and a_I the enriched ones. For a domain Ω presenting both classical and enriched *dof*, the equilibrium discrete equations for dynamic analysis with XFEM are:

$$\begin{bmatrix} M_{uu} & M_{ua} \\ M_{au} & M_{aa} \end{bmatrix} \begin{Bmatrix} \ddot{u} \\ \ddot{a} \end{Bmatrix} + \begin{bmatrix} K_{uu} & K_{ua} \\ K_{au} & K_{aa} \end{bmatrix} \begin{Bmatrix} u \\ a \end{Bmatrix} - \begin{Bmatrix} F^{ext} \\ Q^{ext} \end{Bmatrix} = 0 \tag{2}$$

where in the global mass and stiffness matrix the mixed and enriched terms are presented denoted by index *ua*, *au* and *aa*, respectively. The equilibrium equations system 2 must be completed with a crack evolution model providing the answer to three questions: when the crack advances, in which direction and how far it advances. In our implementation of XFEM two models are proposed.

The first one is the physical model based on energy release rate calculation. The evaluation of the the energy release rate is achieved using the far-fields, through the calculation of the path-independent dynamic J-integral [2] along a contour Γ surrounding the crack-tip:

$$J_k = \int_\Gamma \left[(W+U)\, n_k - \sigma_{ij} \frac{\partial u_i}{\partial x_1} n_j \right] d\Gamma \tag{3}$$

where W and U are the strain and kinetic energy densities respectively, and n_i is the unit normal vector to Γ. The value of this integral gives the propagation criterion: the crack will advance if the current energy release rate exceeds a critical limit. The advancement direction of the crack is given by:

$$\theta_c = 2\arctan\left\{ 1/4 \left(K_I/K_{II} - sign(K_{II}) \sqrt{(K_I/K_{II})^2 + 8} \right) \right\} \tag{4}$$

where K_M are dynamic stress intensity factors extracted from J-integral components. The crack speed is provided by the numerical propagation algorithm, since the crack-tip advances one edge at a time.

The second crack evolution model is based on the evaluation of the crack opening displacement (COD) at the crack-tip. The crack will advance when the COD exceed a critical limit given as a parameter of the cohesive zone. The advancement direction is given by the maximum circumferential stress criterion.

2 NUMERICAL PROCEDURES FOR XFEM IMPLEMENTATION IN DYNACRACK

The numerical implementation of XFEM is achieved on behalf of the explicit FEM code **DynELA** [4] in a new module intended for XFEM, called **DynaCrack**. The main numerical

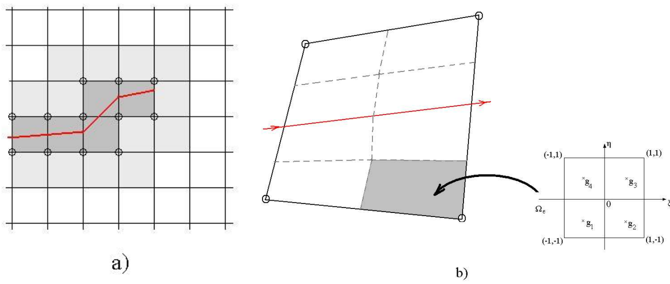

Figure 1: a) XFEM enriched mesh; b)Partition of a cutted element

Parameter	Fine mesh	Middle mesh	Coarse mesh	Abaqus
$G\ [J/m^2]$	$1.18 \cdot 10^{-3}$	$1.26 \cdot 10^{-3}$	$1.36 \cdot 10^{-3}$	$1.23 \cdot 10^{-3}$
$K_I\ [Pa\sqrt{m}]$	1521.7	1617.9	1624.2	1480.4
$K_{II}\ [Pa\sqrt{m}]$	-404.4	-460.9	-493.8	-722.4

Table 1: Fracture parameters values from DynaCrack and Abaqus analysis

algorithms programmed for XFEM implementation in **DynaCrack** concern the enrichment of nodes and elements, the partition of cutted elements, assembly procedure of global matrix, explicit integration of governing equations and crack evolution models.

For introducing the additional *dof*, an algorithm was programmed to identify the enriched nodes and the value of the Heaviside function. In Figure 1a the enriched nodes are encircled, the cutted enriched elements are shaded in dark grey and the others enriched elements (having at least one enriched node) are shaded in light grey. For the enriched elements, the specifically algorithms were developed in order to compute the integrating terms as mass and stiffness matrix, handling variable number of *dof*.

Cutted elements impose the partitioning of the integration domaine for the mass and stiffness matrix using quadrilateral partitions as shown in Figure 1b. Computation of the physical crack evolution model is based on the evaluation of the J-integral using a square domain around the crack-tip. The dynamic stress intensity factors are extracted by direct method.

3 VALIDATION EXAMPLE

We consider the cracked panel shown in Figure 2a for analysing the mixed-mode fracture. The reference length $L = 1\,m$ and the distribution traction $\sigma = 1000\,Pa$. The properties of the material are: $E = 2 \cdot 10^{11}\,Pa$, $\nu = 0.3$ and $\rho = 7833\,kg/m^3$.

Three different regular meshes were considered: (20x35, 25x44 and 30x53 elements). A comparison was made with the results obtained by the **Abaqus** simulation of the same model with a mesh of 911 elements, aligned with the crack geometry. As one can see in Figure 2 b) and c), the distribution field of von Mises stress is similar for both codes and has the expected analytically shape. The fracture parameters for mixed mode are presented in Table 1.

A quite good agreement was found between **DynaCrack** and **Abaqus** analysis for the energy

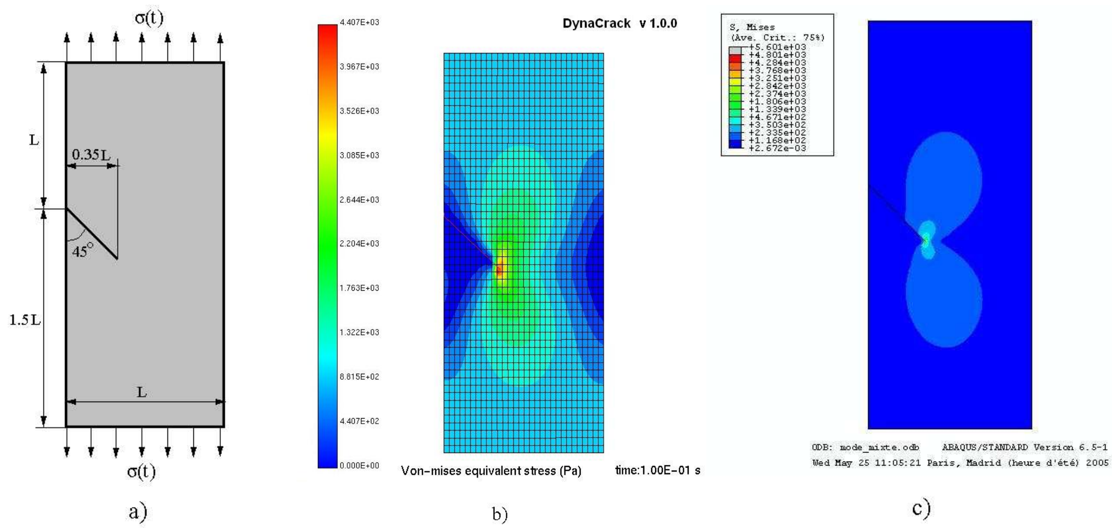

Figure 2: Mixed_mode crack problem. a) The geometry of the model; b) DynaCrack von Mises stress distribution; c) Abaqus von Mises stress distribution;

release rate G and the dynamic stress intensity factor K_I. Concerning the K_{II} values, the quite important differences could be explained by the crack-tip modeling in XFEM with the Heaviside step function. The displacement field was also compared and the relative difference between **DynaCrack** and **Abaqus** analysis for the vertical displacements of upper and lower panel edges are less of 8%.

4 CONCLUSION

The numerical implementation of XFEM in an explicit FEM code has been achieved for treating dynamic crack propagation. Several algorithm were programmed in order to realise the crack implementation, the enrichment of mesh, the numerical integration of mass and stiffness matrix over cutted elements, the explicit integration of governing equation and the implementation of two crack evolution models. An numerical example involving a mixed-mode fracture analysis proves the robustness of the implemented algorithms.

REFERENCES

[1] I. Babuska and J. Melenk. The partition of unity method. *IJNME*, 40:727–758, 1997.

[2] L. B. Freund. *Dynamic Fracture Mechanics*. Cambridge University Press, 1998.

[3] N. Moës, J. Dolbow, and T. Belytschko. A finite element method for crack growth without remeshing. *IJNME*, 46:131–150, 1999.

[4] O. Pantalé, S. Caperaa, and R. Rakotomalala. Development of an object-oriented finite element program: application to metal forming and impact simulation. *J-CAM*, 168(1-2):341–351, 2004.