COMBINING MESH REFINEMENT AND XFEM FOR FRACTURE MECHANICS SIMULATIONS: CONTRADICTION OR STRENGTH?

Dr. C. Henrard, Dr. M. Bruyneel, Ir. J.-P. Delsemme (LMS Samtech, Belgium)

Dr. C. Henrard, XFEM Project Leader and SAMCEF R&D Engineer

THEME

Fracture Mechanics using XFEM combined with mesh refinement.

SUMMARY

The goal of this article is to evaluate the interest of combining a so-called *mesh-independent* extended finite element method (XFEM) for fracture mechanics with dynamic mesh refinement.

Since its beginning in 1999, XFEM has been considered a revolution in fracture mechanics simulation thanks to the fact that it does not require any modification to the geometry or the mesh in order to model a crack. It remains, nevertheless, a finite element method for which the mesh needs to be fine enough to capture the evolution of the fields (stress and displacement) it tries to model.

In this article, the dynamic mesh refinement functionality available in the Morfeo/Crack plugin for SAMCEF will be assessed. In particular, the apparent contradiction of using a mesh-independent method in combination with dynamic mesh refinement will be explained. The interest of this method will be justified both in terms of accuracy and computation time.

KEYWORDS

SAMCEF, Fracture Mechanics, XFEM, Mesh Refinement.

COMBINING MESH REFINEMENT AND XFEM FOR FRACTURE MECHANICS SIMULATIONS: CONTRADICTION OR STRENGTH?

1: Introduction

SAMCEF is a general-purpose finite element software (LMS Samtech; Nyssen, 1979; SAMCEF, 1974). It is mainly used in the aeronautical, automotive and energy sectors. Its strength stems from its abundant element library—containing, among others, beam, shell, volume, contact, assembly, kinematic joints, and post-processing elements—and its wide variety of boundary conditions and material laws going from the simplest linear elastic law to the most complex multi-layered composite material law or the non-local damage model.

In terms of fracture mechanics, SAMCEF has been used for several decades by large aeronautical companies to model complex 2D and 3D problems using crack boxes including Barsoum elements (Barsoum, 1976). This allowed a user to take into account the effect of fatigue loading into its design by computing the stress intensity factors. More recently, in 2005, the eXtended Finite Element Method (XFEM) was introduced along with the level set technique to be able to insert one or several cracks in a part without having to modify the CAD or the mesh to represent the crack lips and tips. This brought a significant improvement in terms of reduction of the *modeling time* for the user with a small cost of computation time. This method was however limited to 2D computations and crack propagation still remained a manual task.

Since 2010, the XFEM method has also been available in 3D thanks to the use of the Morfeo/Crack plugin for SAMCEF developed by Cenaero (Wyart, et al., 2008). This plug-in features fully-automatic crack propagation under fatigue loading with a choice of propagation law and dynamic mesh refinement. It has been successfully applied to advanced industrial problems (Henrard, et al., 2011).

Within this context, this article will focus on the use of the dynamic mesh refinement technique combined with XFEM. After a brief description of XFEM and the mesh refinement technique, the article will evaluate this method in terms of the accuracy of the stress intensity factors (SIF), the propagation path and the computation time.

2: XFEM method

The XFEM method used inside SAMCEF and its Morfeo/Crack plugin is based on the original principles as developed by (Moës, et al., 1999). It uses a specific integration scheme for the element cut by the crack or containing the crack tip. Moreover, SAMCEF uses an implicit crack representation with level-sets (Sukumar, et al., 2001). After computing the J-integral, it is decomposed into the three stress-intensity factors (SIFs) K_1, K_2 and K_3 using the interaction integral method (Yau, et al., 1980).

More information about all these principles can be found in the literature (Wyart, et al., 2008).

3: Dynamic mesh refinement

XFEM is intrinsically a mesh-independent method. In the context of fracture mechanics, this means that the crack can cut through any element so that the mesh does not need to conform to the crack position. Anyone who ever had to create a mesh with crack boxes would know that this can lead to a tremendous reduction of the model creation time.

Nevertheless, XFEM remains a finite element method for which the size of the mesh plays a crucial role in the accuracy of the solution. Without remeshing, performing a crack propagation analysis for which the crack path is not known in advance requires the use of one of the two following methods:

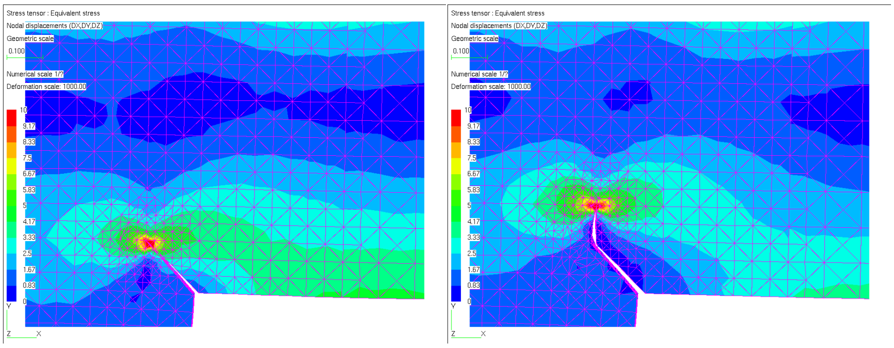
- refine a large region containing all the finite elements in which the crack could possibly grow, which could be of a substantial size;
- refine locally around the initial crack tip and perform a few propagation steps to see where the crack propagates. Then, create a new mesh which is *also* fine in the new crack tip location and restart the computation from the beginning. Using this procedure iteratively, the mesh will be progressively refined along the whole crack propagation path.

This second method is not only inefficient in terms of computation time — the computation needs to be restarted multiple times from the beginning and the mesh will end up to be fine along the whole crack path instead of just the crack tip — but creating such meshes is difficult and time consuming.

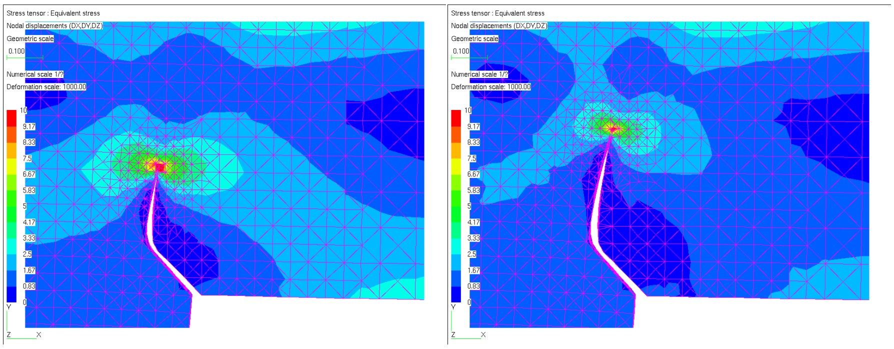
For all these reasons, SAMCEF released end of 2012 version 15.0-04 enhanced with dynamic mesh refinement. The idea behind this feature is to automatically refine the mesh at each propagation step so that:

- the mesh is sufficiently fine around the crack tip for the SIFs to be accurately computed;
- the mesh is coarse far away from the crack tip and, in particular, along the crack surface.

Refining and coarsening the mesh is not an easy task. The method implemented in SAMCEF is based on MaDLib, the open-source Mesh Adaptation Library (UCL, 2009). First, the user must create a mesh which is at least *fine enough* to perform an accurate simulation without crack but *as coarse* *as possible* to limit the computation time. Then, at each propagation step, SAMCEF estimates the desired mesh size around the crack tip(s) and crack surface. MaDLib will then automatically refine the mesh and SAMCEF will update the model (mesh, materials and boundary conditions). When the crack propagates, the zone with the finest desired size will follow the crack. Since the new meshes are recomputed for every step from the initial coarse mesh, the mesh will be refined when the crack tip gets closer and coarsened with it moves away, as shown in Figure 1.



(a) Step 0 (b) Step 2



(c) Step 4 (d) Step 6

Figure 1: Dynamic mesh refinement in 3D during crack propagation

If this fully-automatic refinement feature does not yield to satisfactory results or if the user wants to have more control about the mesh size, it is also possible to manually specify:

- the desired mesh size around the crack *tip* (and the radius of the zone in which this size should be enforced)
- the desired mesh size in the vicinity of the crack *surface* (and the width of this zone).

4: Problem description

Throughout the rest of this article, the evaluation of the mesh refinement method will be based on a thermo-mechanical problem found in the literature

(Prasad, et al., 1994). In this article, this problem was solved in 2D using a boundary element method (BEM). It was also examined more recently using XFEM in (Duflot, 2008).

The geometry of the part can be found in Figure 2. It is composed of a cruciform plate (cross with four arms). The ratio *a/L* is equal to 0.2 and the initial crack forms an angle of 45 degrees with the vertical axis.

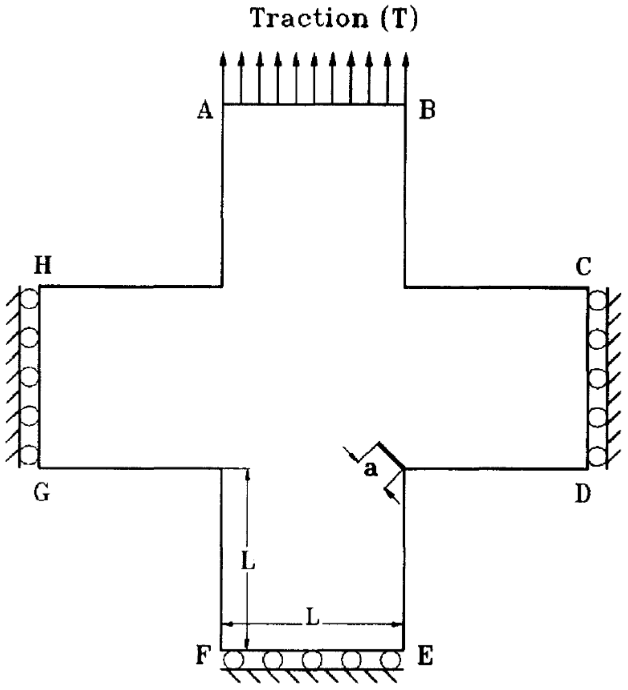


Figure 2: Geometry of the thermo-mechanical problem (Prasad, et al., 1994)

Load Case	Temperature [°C] AB	CD	EF	GH	Traction T [*Pa*] AB	Description
(1)	10	0	-10	0	0	Thermal only
(2)	0	0	0	0	10	Mechanical only
(3)	10	0	-10	0	10	Mechanical + thermal
(4)	20	0	-20	0	10	Mechanical + 2 * thermal
(5)	10	-5	-10	-5	10	Idem (3) -5°C on horiz. arms

Table 1: Boundary conditions of the thermo-mechanical problem

The five sets of boundary conditions (load cases) applied on the part are shown in Table 1. They are composed of various combinations of thermal and mechanical boundary conditions applied on the extremities of the four arms.

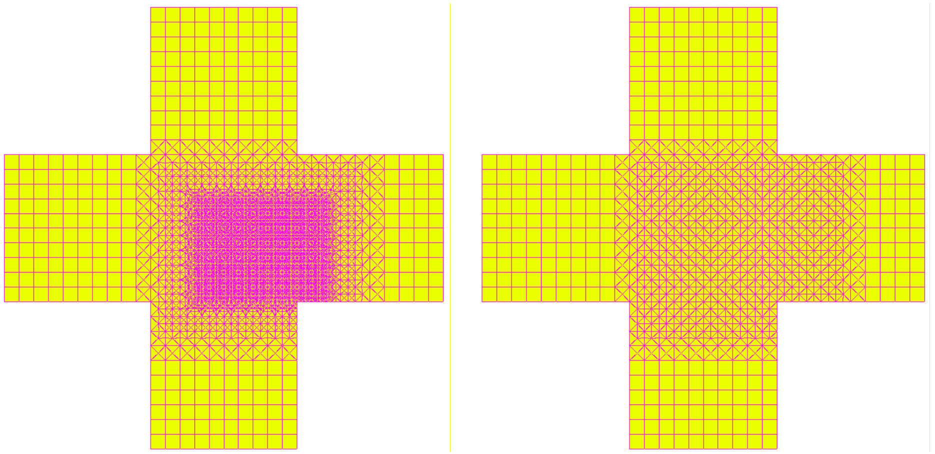
In the present article, the problem was solved in 3D. The thickness of the plate was kept constant and equal to 0.4*L* to be as close as possible to a plane-strain state. The computation was done in two steps. First, using Mecano/Thermal

(SAMCEF, 1974), the thermal field was computed on the crack-free structure based on the imposed temperatures on the extremities of the four arms. Secondly, these temperatures were projected on a new mesh and used as boundary conditions for the mechanical computation. This computation was performed taking the crack into account using XFEM elements.

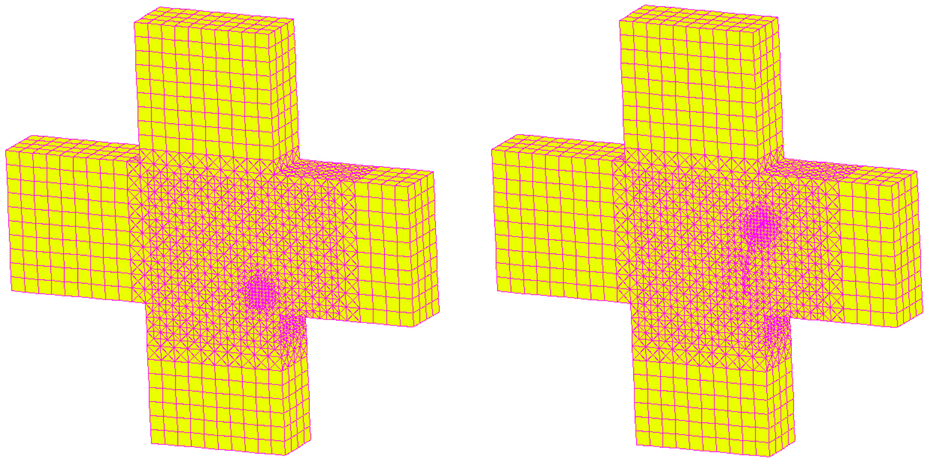
The interesting feature about this test is that the crack path varies greatly from one load case to the other as shown in Figure 4, making it a good validation problem.

5: Meshes

Two different meshes were used for the mechanical computation. On the one hand, the mesh shown in Figure 3(a) uses a fixed size of 0.02mm on the whole central region. The goal is to allow the crack to propagate using the same mesh for all of the 5 load cases while always keep a fine mesh at the crack tip.



(a) Fixed mesh (b) Coarse mesh for dynamic refinement



(c) Dynamic mesh at propagation step 0 (d) Dynamic mesh at propagation step 10

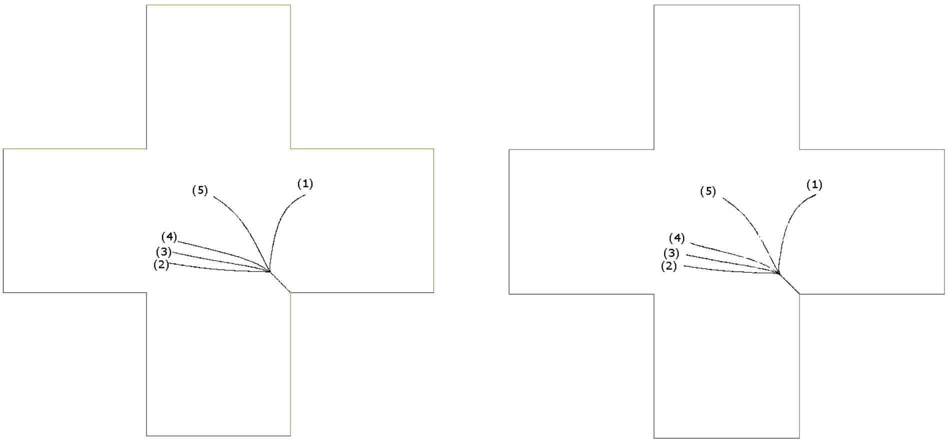
Figure 3: Meshes used for the simulations

On the other hand, the mesh shown in Figure 3(b) is relatively coarse but will be dynamically refined at each propagation step so as to impose the same element size around the crack tip as the fixed mesh. As examples, Figure 3(c) and (d) show the refined mesh used for load case 1 at propagation steps 0 and 10.

The fixed mesh contains slightly more than 605,000 elements throughout the simulation and 591,000 XFEM elements. The initial coarse mesh used for the dynamic refinement contains 71,000 elements. During crack propagation, the number of XFEM elements increases from 75,000 to almost 85,000 elements, which is about 7 times less than for the fixed mesh.

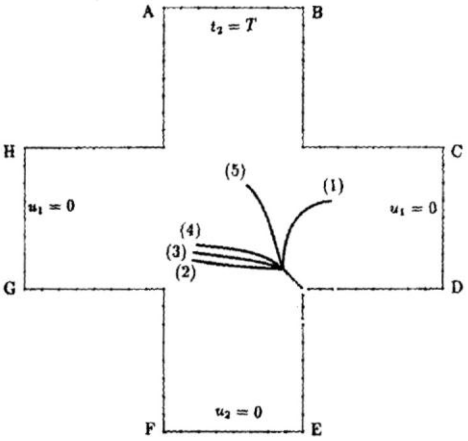
6: Crack propagation path

The crack path is shown after 11 propagation steps in Figure 4 for both meshes and compared to the reference solution found in literature. Both computations yield to identical paths which are very close to the reference.



(a) Crack path with fixed mesh

(b) Crack path with dynamic mesh

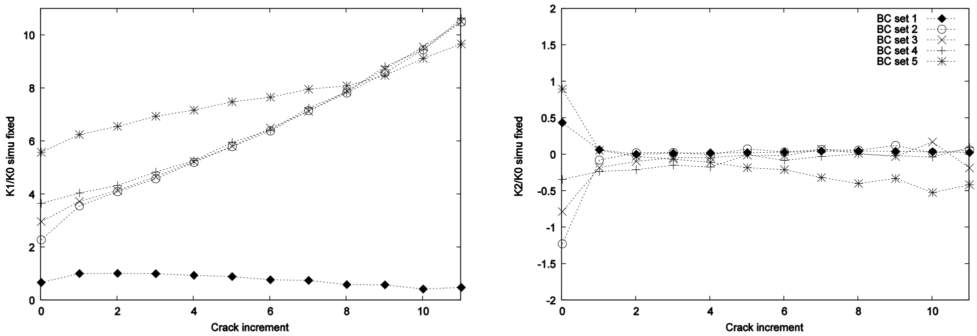


(c) Reference path (Prasad, et al., 1994)

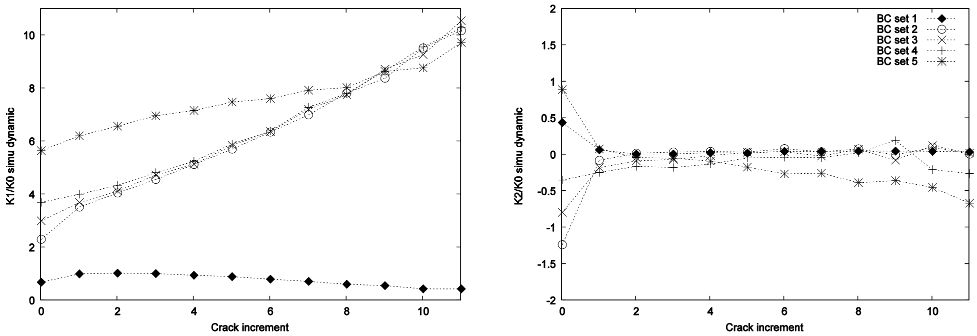
Figure 4: Crack path after 11 propagation steps

7: The evolution of the first stress intensity factor K_1 during crack propagation is Stress Intensity Factors

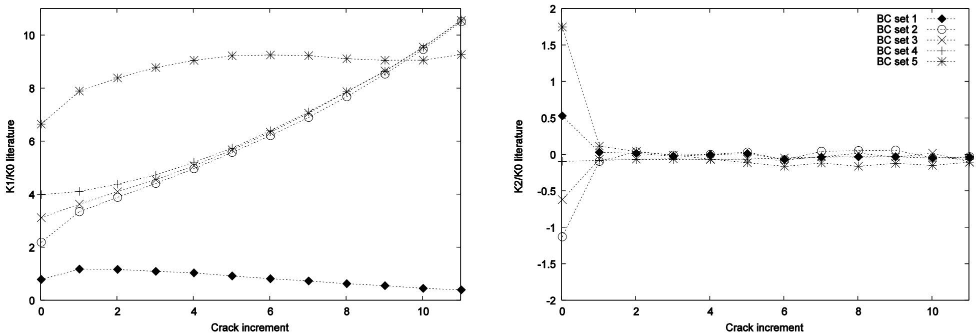
The evolution of the first and second stress intensity factors K_1 and K_2 during crack propagation are shown in Figure 5 for both meshes and for the reference solution of the literature. Since the computations are performed in 3D, the value of K_1 is averaged along the center portion of the thickness to limit the potential inaccuracy close to the free surfaces.



(a) K1/K0 fixed mesh (b) K2/K0 fixed mesh



(c) K1/K0 dynamic mesh (d) K2/K0 dynamic mesh



(e) K1/K0 reference (f) K2/K0 reference

Figure 5: Evolution of K_1 and K_2 during crack propagation

In these figures, the SIFs values are normalized to a value K_0 equal to:

$$K_0 = \alpha E \sqrt{\pi a_0} \tag{1}$$

Similarly to the crack paths results, the values of the SIFs obtained with both simulations are identical and very close to the reference solution.

8: Computation times

The computation times of load case 1 are given for both meshes in Table 2.

	Fixed mesh		Dynamic refinement	
Global simulation time	27039 *s*	100%	5944 *s*	22%
Avg. propagation step time	1405 *s*	100%	308 *s*	22%
- remeshing time	-	-	123 *s*	-
- resolution time	215 *s*	100%	32 *s*	15%
- post-processing time	250 *s*	100%	72 *s*	29%

Table 2: Computation times of load case 1

Despite the fact that refining the mesh and projecting the level sets of the crack on the new mesh at every propagation step takes a significant amount of time (one third of the step time on average), the global computation time is nevertheless much faster and approximately divided by five. In addition, the

9: Conclusion

In this article, a thermo-mechanical fracture mechanics problem was analyzed using SAMCEF and compared to a reference solution found in literature. The simulation was performed using XFEM elements.

Even though XFEM is said to be mesh-independent, it remains nevertheless a finite element method for which the mesh needs to be fine enough to get accurate results. This is especially true in the vicinity of the crack tip where the stresses are singular.

Throughout this article, the thermo-mechanical problem was simulated either with a fixed mesh or using a new feature called dynamic mesh refinement. This feature considerably reduces the amount of work necessary to prepare the meshes and allows the user to ensure that the mesh is always fine enough in the vicinity of the crack tip.

The conclusion of this study is that to get equally accurate results, dynamic refinement drastically reduces the computation time and the model creation time.

COMBINING MESH REFINEMENT AND XFEM FOR FRACTURE MECHANICS SIMULATIONS: CONTRADICTION OR STRENGTH?

REFERENCES

Barsoum, R., 1976. On the use of isoparametric finite elements in linear fracture mechanics. *Int. J. Num. Meth. Eng.,* 10(1), pp. 25-37.

Duflot, M., 2008. The extended finite element method in thermoelastic fracture mechanics. *Int. J. Num. Meth. Eng.,* 74(5), pp. 827-847.

Henrard, C. et al., 2011. *Computational Fracture Mechanics using XFEM in SAMCEF: Validation on a Industrial Test Case.* Cardiff, UK, Cardiff University, In The 2nd International Conference on the eXtended Finite Element.

LMS Samtech, n.d. *More information about SAMCEF can be found on the LMS Samtech website.* [Online]
Available at: http://www.lmsintl.com/simulation/samtech
[Accessed February 2013].

Moës, N., Dolbow, J. & Belytschko, T., 1999. A finite element method for crack growth without remeshing. *Int. J. Num. Meth. Eng.,* 46(1), pp. 133-150.

Nyssen, C., 1979. *Modélisation par élements finis du comportement non linéraire de structures aérospatiales,* Liege, Belgium: University of Liege.

Prasad, N. N. V., Aliabadi, M. H. & Rooke, D. P., 1994. Incremental Crack Growth in Thermoelastic Problems. *Int. J. Fract.,* Volume 66, pp. R45-R50.

SAMCEF, 1974. *Système d'analyse des milieux continus par éléments finis. Theory and documentation of the finite element code (in french),* Liege, Belgium: LTAS (University of Liege).

Sukumar, N., Chopp, D., Moës, N. & Belytschko, T., 2001. Modeling holes and inclusions by level sets in the extended finite element method. *Comput. Meth. Appl. Mech. Eng.,* 190(46), pp. 6183-6200.

UCL, 2009. *MAdLib.* [Online]
Available at: http://sites.uclouvain.be/madlib
[Accessed 02 2013].

Wyart, E. et al., 2008. Substructuring FE-XFE approaches applied to threedimensional. *J. Comp. Appl. Math.,* 215(2), pp. 626-628.

Yau, J., Wang, S. & Corten, H., 1980. A mixed-mode crack analysis of isotropic solids using conservation laws of elasticity. *J. Appl. Mech.,* Volume 47, pp. 335-341.