Numerical simulation of crack propagation in pressure equipments

Abdelhadi El Hakimi[a], Philippe Le Grognec[a], Saïd Hariri[a],
Mansour Afzali[b], Haïdar Jaffal[b]

[a] Ecole des Mines de Douai, 941 rue Charles Bourseul BP 10838, 59508 Douai Cedex, France
[b] CETIM, 52 avenue Félix Louat, 60300 Senlis, France

Abstract. The present study deals with the severity of cracks in pressure equipments, where such defects are often involved. The objective is to assess the growing harmfulness level of a given propagating defect, in order to predict the residual lifetime of a pressure equipment. The stress intensity factors are thus performed at each step during the crack front propagation. Our work is particularly concerned with the problems of shells including almost all sorts of practical defects, namely both internal and external, longitudinal or transverse semi-elliptic cracks. Use is made of a specific adjusted crack-block so as to mesh the vicinity of the crack front, in the framework of the finite element method. Moreover, the Paris' law is generalized to the case of 3D fatigue crack extension. Finally, a Python program is particularly developed to automate the calculation process, only involving a local remeshing and jointly using Abaqus and Castem softwares.

Simulation numérique de la propagation de fissures dans les structures sous pression

Résumé. *L'objectif de ce travail est de déterminer l'évolution du degré de nocivité d'un défaut dans un équipement sous pression durant sa propagation. L'estimation du degré de nocivité d'une fissure passe par le calcul des facteurs d'intensité de contrainte à chaque état d'avancement du front de fissure. Les fissures considérées sont semi-elliptiques. Les géométries et chargements peuvent être complexes de façon à couvrir la majorité des cas industriels. La modélisation numérique par éléments finis s'appuie sur la création d'un bloc-fissure, représentant le maillage optimisé du voisinage de la discontinuité. La loi de Paris permet de décrire le comportement en fatigue sous chargement cyclique. Un programme spécifique (Python), alliant les avantages des codes de calcul Castem et Abaqus, permet d'automatiser la démarche de propagation et facilite l'évaluation de la durée de vie résiduelle d'une structure sous pression fissurée.*

1 INTRODUCTION

Cracks are very often involved in pressure vessels. For obvious safety reasons, considering the important risk incurred, it is essential to assess the harmfulness level of such defects with a rather good accuracy. Furthermore, it is crucial to appraise the growth rate of this harmfulness degree, when the crack spreads over the thickness of the structure, giving rise to the residual lifetime of the pressure equipment.

Such problems are most often resolved numerically, using the classical finite element method, in the framework of fracture mechanics. But the numerical simulation of a three-dimensional propagating crack still represents a tedious task, as far as meshing is concerned. Indeed, a local refinement (and/or specific elements such as Barsoum ones) is required in the vicinity of the crack front in order to account for the asymptotic behaviour of the displacement field near the singularity. Moreover, remeshing will be needed to consider the propagation of the crack.

The total remeshing of the entire structure at each step of the front propagation is clearly prohibitive, so that a classical solution exists, consisting in isolating the discontinuity in a so-called "crack-block", describing the vicinity of the crack front. The crack-block mesh depends on the crack size, but it can be introduced anytime in the entire structure, whose mesh remains independent of the crack growth. One can mention, for instance, Dhondt [1] who suggested a particular solution of mesh local refinement with generation of hexahedral elements on the crack transition, in the context of the finite element method.

The meshing and remeshing techniques clearly represent the stumbling block of the numerical simulation in that context. As a consequence, the so-called "meshfree" methods have been developed, which allow theoretically to get free from all the mesh-related problems. The approximation lies now upon a set of points and corresponding shape functions, possibly discontinuous ones around the crack front. Of course, these methods require no more mesh refinement during the crack propagation. Nevertheless, the numerical integration calculations and the boundary conditions are much less trivial than in the context of the finite element method.

For a few decades, the eXtended-Finite Element Method has been considered as a compromise between the previous extreme solutions. In fact, it preserves the mesh advantages without geometrical constraints, by improving specifically the finite element shape functions using the partition of unity method. Nevertheless, all these methods can hardly apply in our industrial context.

In this study, a modeling tool is performed for the determination of crack propagation in pressure equipments. It is based on the previous concept of a crack-block in order to minimize the paramount cost of meshing operations. Shell-type structures are considered, involving semi-elliptic surface flaws.

The main task consists in the achievement of a Python program, implying advantages of both Castem and Abaqus softwares. These two finite element codes have been retained for their facilities in terms of meshing and geometrical modeling, respectively, in the absence of a versatile one. The computer program completely automate the crack propagation calculations for any geometries and loadings, while ensuring a constant accuracy in the computation of the stress intensity factors along the crack front. As an example, the Paris' law is used to illustrate the fatigue behaviour of a cylindrical shell under axial tensile loading with an external circumferential semi-elliptic defect.

2 IMPLEMENTATION OF THE FINITE ELEMENT SIMULATION TOOL

2.1 Crack-block

A plane-parallel crack-block is designed in the framework of the finite element method, encompassing the crack front and the corresponding discontinuity surface, which represents the vicinity of any semi-elliptic defect to be included in any shell-type structure. This crack-block is big enough to contain the deepest and largest crack considered with its influence zone [2] [3] [4] [5] (see Figure 1).

From one side, the crack-block finite element mesh is optimized and specifically adapted to the size and aspect ratio of the actual crack in order to reproduce proper values of the stress intensity factors (for the opening mode) or the contour J integrals (see [6] for a precise definition) in comparison with reference solutions from CEA for various geometries of cracked cylinders [7].

>From another side, the external envelope of the crack-block is meshed very carefully, in order to join more easily the corresponding surfaces in the future connection between the crack-block and the remainder.

Use is made of the finite element Castem software to build the crack-block geometry and mesh, taking advantage of its meshing facilities based on geometrical transformations. Twenty-noded hexahedral elements (CU20) and, to a smaller extent, fifteen-noded prismatic ones (PR15) are employed, all of them involving intermediate nodes for a quadratic interpolation.

Before the crack-block is inserted in the shell body, it may be bended according to the proper radius of curvature, depending on the local geometrical shape of the structure and especially on the crack place in the shell (for instance, the crack may be internal or external, circumferential or longitudinal, in the case of a cylindrical shell).

2.2 Data transfer

The meshing of the entire structure, except the crack-block, and the finite element calculations will be performed using Abaqus software, allowing for more complex geometries and making easier the creation or the importation of all sorts of meshes. For the time being, it is also necessary to dump a lot of data concerning the crack-block from Castem to Abaqus.

The crack-block finite element mesh is first saved in a file with extension *.sauv*, namely the nodes coordinates and the elements connection table (orphan mesh), together with geometrical entities such as the crack front and the crack lips. This Castem file is then converted into an input file for Abaqus with extension *.inp*, by means of a script program in Perl language and an intermediate data file.

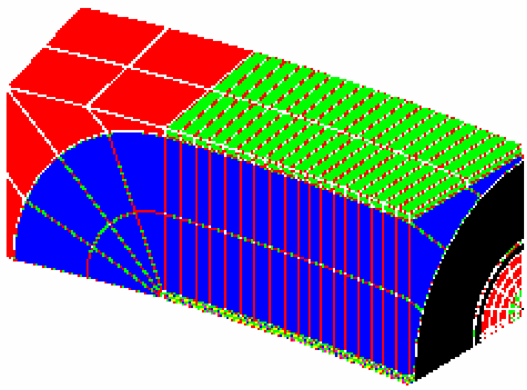

Figure 1. Crack-block.

2.3 Modeling and calculus

A specific program in Python language is developed within the Abaqus script interface to automate the calculation process. This main program executes Castem and then the Perl transfer program. After that, it defines the geometry, the mesh, the material properties, and the loading and boundary conditions on the whole structure. Then, the stress intensity factors are computed and smoothed along the crack front, using the virtual crack extension method.

The main problem concerns the use of script modeling and meshing operations in the case of complex geometries. For simple cylindrical or spherical shells, it is possible to build the crack-block counterpart using easy geometrical transformations such as extrusions or revolutions, and the mesh is straightforward. But the problem is much more intricate for current industrial appliances.

The optimal solution consists in importing the geometry of the structure from any CAO tool (e.g. using Abaqus Graphical User Interface). The meshing of the remaining structure (except the crack-block) is then performed just once (possibly with the same tool) in compliance with the crack-block invariant external surfaces, whose mesh is sufficiently simple to be "manually" reproduced by any Graphical User Interface. As a result, just a few geometrical and meshing operations remain in the Python program, namely assembling the two imported parts and merging the face to face nodes.

2.4 Fatigue propagation law

At the end of each step of the calculation process, the crack growth has to be evaluated. In this work, we only focus on plane crack propagation, given that the crack-block involves a plane semi-elliptic crack front. The Paris' law [8], usually written in a two-dimensional framework, is applied here, in the case of 3D fatigue crack extension, at each node of the crack front. The crack advance in the plane orthogonal to the front at each point is governed by the following relation, only considering the paramount opening mode effect:

$$\frac{da}{dN} = C(\Delta K)^m \quad (1)$$

In the expression above, a represents the crack depth, N the number of cycles, and $\Delta K = K_{\max} - K_{\min}$ stands for the cyclic variation of the stress intensity factor. C and m are constant material parameters. It should be mentioned that the previous law has already been used by other authors in a modified way, taking into account plasticity effects or loading amplitude changes.

>From a practical point of view, the crack front propagation is calculated every ΔN cycles. At each new step, the Paris' law is discretized, giving rise to the incremental displacement Δa of any node along the crack front. In another way, it would be possible to define the size increment as the maximum advance $\Delta a_{\max}$ at the nodal point where the stress intensity factor reaches its maximum value. The number of cycles is deduced by applying the Paris' law at this point, providing then all the other local advances along the crack front. In order to comply with our crack-block use, including a semi-elliptic defect, the new crack front, initially obtained numerically (node by node), is replaced by a semi-elliptic one where the new dimensions a and c are approximated by the least mean square method.

Finally, the flowchart of Figure 2 summarizes the whole calculation process, including at each step of crack propagation the crack-block updating, the stress intensity factors evaluation and the crack front advance.

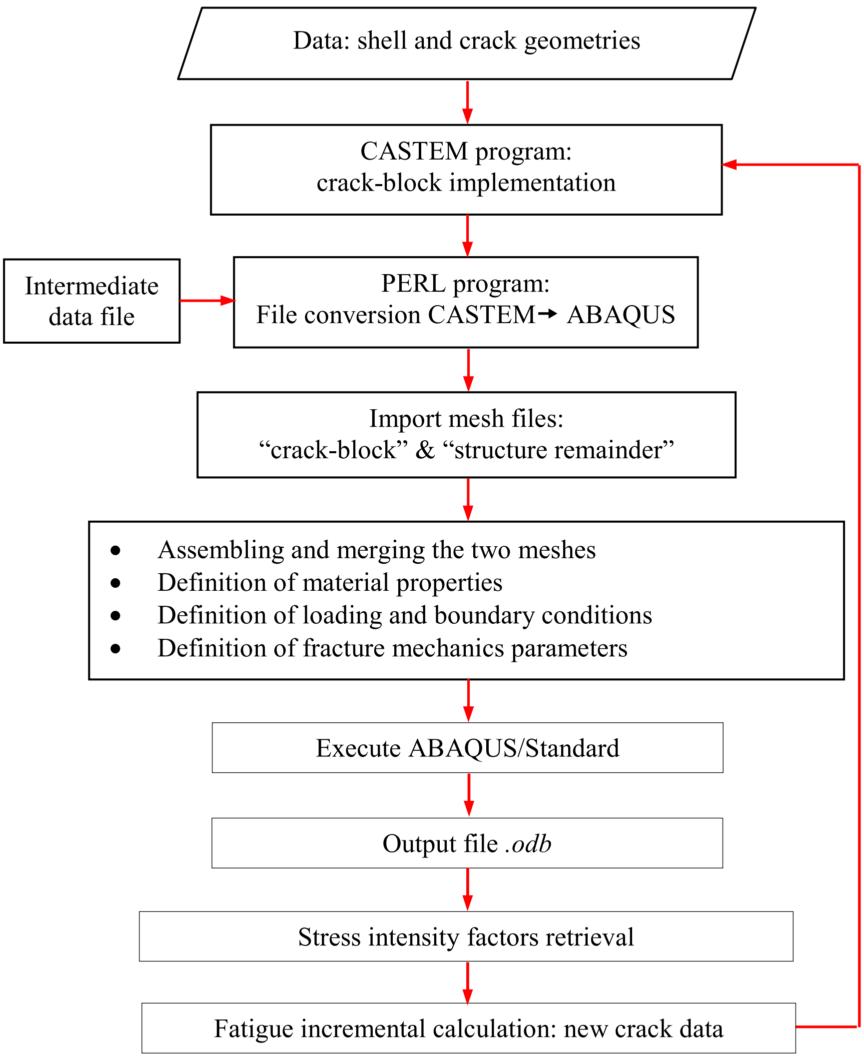

Figure 2. Python program.

3 VALIDATION EXAMPLE

As an example, let us consider the case of a cylinder with a semi-elliptic external circumferential crack, subjected to an axial tensile loading. The internal radius of the cylinder is $R = 600\ mm$, its thickness $t = 60\ mm$ and its length $L = 4\sqrt{Rt}$. The initial crack is defined by its relative depth $a/t = 0.2$ and its aspect ratio $c/a = 2$.

The material is supposed to be linear elastic with Young's modulus $E = 200000\ MPa$ and Poisson's ratio $\nu = 0.3$. Additional material parameters are needed in the definition of the Paris' law: $C = 1.64\,10^{-10}\ MPa^{-2}$ and $m = 2$. The cylindrical shell is submitted to an axial tensile stress of $100\ MPa$, so that the opening mode of the crack is involved. Only one quarter of the structure is modeled, taking into account both axial and transverse symmetries.

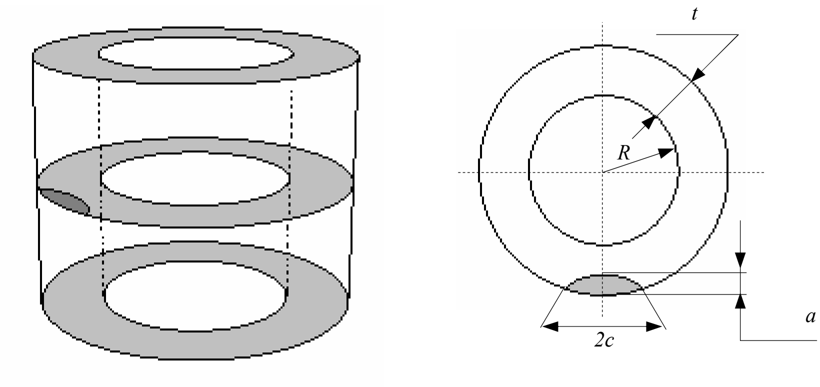

Figure 3. Geometrical parameters.

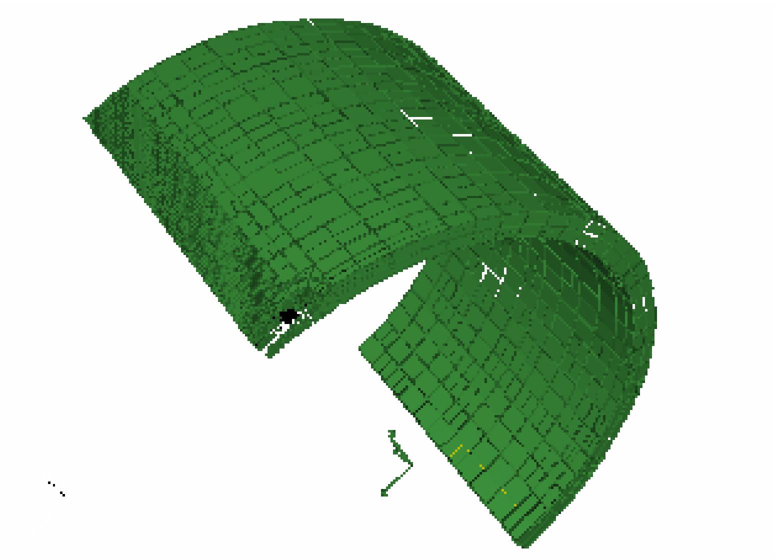

Figure 4. Mesh of a quarter of the cracked cylinder.

Figure 5 shows the growth of the relative depth of the crack vs. the number of loading cycles, until the crack almost passes through the thickness of the shell. In the practical case of pressure equipments, the situation becomes critical well before the crack emerges from the inner surface. Here, no more criteria are used to complete the propagation process. Finally, this solution is in good accordance with similar results from Carpinteri and Brighenti [9].

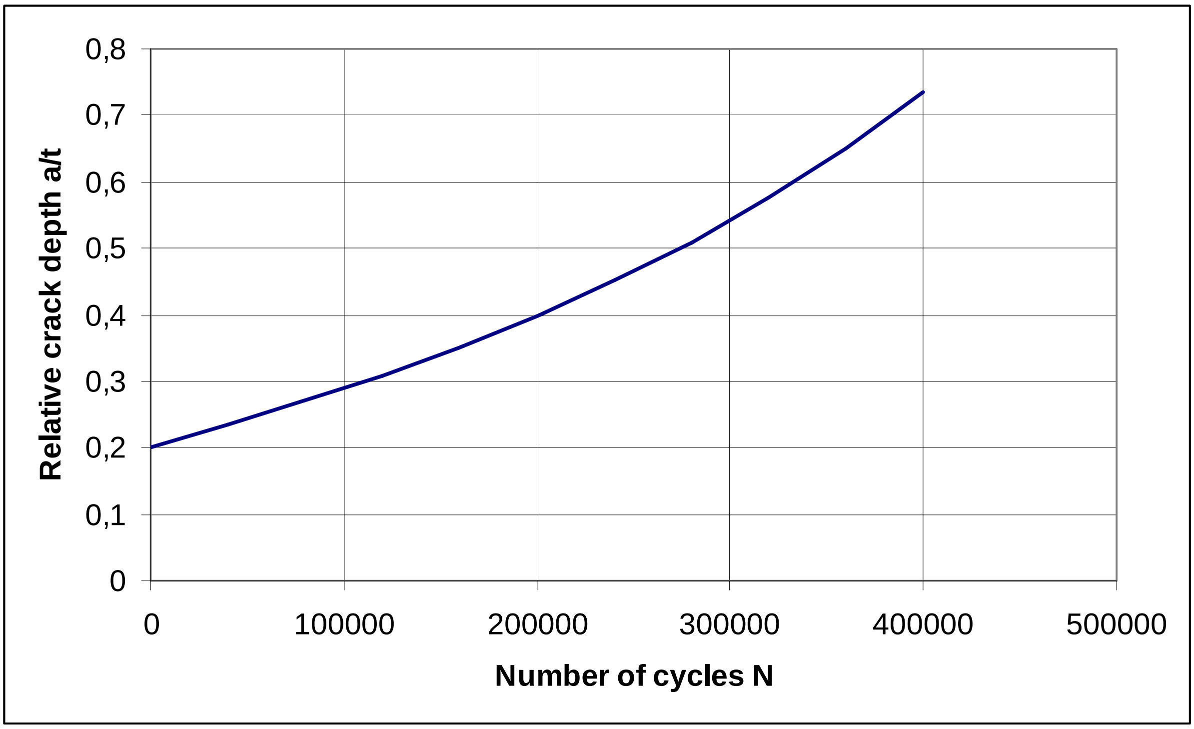

Figure 5. Semi-elliptic crack propagation.

4 CONCLUSION

This study deals with the numerical simulation of the crack propagation in pressure vessels and especially the development of a specific computer tool. In the framework of the finite element method, a so-called "crack-block" is used, displaying a calibrated and optimized mesh in the vicinity of the singularity and confining the mesh refinement around the crack zone. Only semi-elliptic cracks are considered. Castem software is used to build the crack-block, whose data are transferred in an input file for Abaqus software, through a Perl program.

The remaining steps, namely inserting the crack-block in the whole structure (whose mesh could have been achieved using the Graphical User Interface ABAQUS/CAE), defining data such as material properties, loading and boundary conditions, and calculating stress intensity factors, are performed with Abaqus software. All these tasks are put together in a script Python program. Finally, in the context of fatigue crack propagation, a specific subroutine is dedicated to the approximation of the crack evolution, at each increment, using the Paris' law (or equivalent) and satisfying the geometrical condition that the crack front remains semi-elliptic during all the process.

A particular attention is devoted to the transitions between the different numerical tools, in order to perfectly automate the calculation process and dispose of a robust and efficient program, quickly providing accurate values for the residual lifetime of cracked equipments under pressure.

5 REFERENCES

1. G. Dhondt, Cutting of a 3D finite element mesh for automatic mode I crack propagation calculations, *International Journal for Numerical Methods in Engineering*, 42, 1998, 749-772.
2. A. Saffih, Etude de la nocivité des fissures dans un cylindre comportant une transition d'épaisseur : comparaison avec un cylindre droit, PhD thesis, University of Sciences and Technologies of Lille, Ecole des Mines de Douai, France, 2003.
3. A. Saffih and S. Hariri, Numerical study of elliptical cracks in cylinders with a thickness transition, *International Journal of Pressure Vessels and Piping*, 83, 2006, 35-41.
4. A. Saffih and S. Hariri, Comparison of semi-elliptical cracks in cylinders with a thickness transition and in straight cylinders – Elastic-plastic behaviour, *Engineering Fracture Mechanics*, 73(17), 2006, 2685-2697.
5. A. El Hakimi, S. Hariri, A. Laksimi, P. Le Grognec and Z. Azari, Etude numérique et semi-analytique de la nocivité de défauts dans les coques sphériques, *European Journal of Computational Mechanics*, 16(1), 2007, 79-101.
6. J. R. Rice, A path-independent integral and the approximate analysis of strain concentration by notches and cracks, *Journal of Applied Mechanics*, 35, 1968, 379-386.
7. S. Chapuliot, Formulaire de K_I pour les tubes avec un défaut de surface semi-elliptique longitudinal ou circonférentiel, interne ou externe, *Research report* – CEA, R-5900, 2000.
8. P. C. Paris and F. A. Erdogan, A critical analysis of crack propagation laws, *Journal of Basic Engineering*, 85, 1963, 528-534.
9. A. Carpinteri and R. Brighenti, Circumferential surface flaws in pipes under cyclic axial loading, *Engineering Fracture Mechanics*, 60(4), 1998, 383-396.