EVALUATION OF ABAQUS XFEM CAPABILITIES FOR CRACK GROWTH ANALYSIS IN AERONAUTICAL STRUCTURES

Ismael Rivero Arévalo and Javier Gómez-Escalonilla Martín

Fatigue and Damage Tolerance Department (TAETS11) – Military Aircraft

Airbus Defence and Space

Abstract: *In the Fracture Mechanics field, standard industrial methods for crack growth analysis are mainly based on analytic calculations as classical Finite Element approaches are not practical to deal with discontinuities such as fatigue cracks due to the associated high computational costs. eXtended Finite Element Method (XFEM) is one of the methodologies that are being developed in the recent years in order to overcome the limitations associated to classical approaches, especially for complex analysis. However limited industrial experience is available to adopt this methodology as a standard practice in the aircraft industry.*

XFEM capabilities available in Abaqus 6.14 have been evaluated by the authors to assess the reliability and feasibility of the implementation of XFEM methodology to perform crack growth analyses in aeronautical structures. A set of different cases, common in aeronautical structures, have been selected for this purpose to check Abaqus capabilities on different configurations.
In the paper authors presents the results obtained for the evaluated cases and the validation of these results using alternative methodologies for comparison. Current limitations in Abaqus capabilities are also discussed and future developments are proposed for implementation in next Abaqus versions.

Keywords: *Aircraft, Fatigue, Crack, XFEM*

1. Introduction

Reduction of structural weight and maintenance requirements are two main goals for aircraft industry as these two facts directly penalize the aircraft operational costs. In the pursue of this target, the development of more accurate, robust and reliable methodologies for structural analysis has a key role as improved methods allows designing more optimized structural designs.

Focusing on the Fatigue and Damage Tolerance field, responsible of the sizing of many aircraft components and the definition of structural maintenance requirements, the use of numerical methodologies provides an important added value. For example the Finite Element Method (FEM) allows the accurate calculation of Stress Concentration Factors for complex configurations where analytic methodologies can only provide a conservative approach.

By the other hand, Crack Growth calculations are mainly based on analytic methodologies as no numerical approaches are consolidated as standard industrial practices. The XFEM (eXtended Finite Element Method) is one of the numerical methodologies that are appearing as an alternative to classical analytic methods for crack growth calculations. It combines the benefits of FEM to analyze complex structures with the capability of dealing with through-element discontinuities as fatigue cracks with acceptable computational cost.
The authors are working on new methodologies based on XFEM to evaluate its capabilities for crack growth calculations and its feasibility to become an industrial standard practice, the same way as FEMs are used for stress calculations.

XFEM capabilities available in Abaqus 6.14 have been evaluated by the authors to assess the reliability and feasibility of the implementation of XFEM methodology to perform crack growth analyses in aeronautical structures. A set of different cases, common in aeronautical structures, have been selected for this purpose to check Abaqus capabilities on different configurations.
In the paper authors presents the results obtained for the evaluated cases and the validation of these results using alternative methodologies for comparison. Current limitations in Abaqus capabilities are also discussed and future developments are proposed for implementation in next Abaqus versions.

2. XFEM capabilities in Abaqus

The eXtended Finite Element Method is an extension of the conventional Finite Element Method based on the concept of Partition of Unity (PUM). This methodology is able to deal with the presence of discontinuities in a finite element by enriching degrees of freedom with special displacement functions. Formulation and mathematical background of the XFEM approach implemented in Abaqus can be found in the literature, (Du) and (Park, 2012).

The displacement vector of a node included in the enriched domain can be expressed mathematically as follows:

$$u = \sum_{I=1}^{N} N_I(x) \left[u_I + H(x)\, a_I + \sum_{\propto=1}^{4} F_{\propto}(x) b_I^{\propto} \right] \quad (1)$$

where $N_I(x)$ are the shape functions affecting the entire set of nodes in the domain, u_i the nodal displacement vectors of the entire set of nodes, a_I the nodal enriched degrees of freedom for the set of nodes whose shape functions supports are fully cut by the crack (set J), $H(x)$ discontinuous or 'jump' functions, b_I^{α} the nodal enriched degrees of freedom for the set of nodes whose shape functions supports are cut by the crack tip and $F_{\alpha}(x)$ adequate asymptotic functions for the displacement field near the crack tip.

To mathematically describe the crack and track its growth the Levet Set Method is used. Two level sets are required to model the crack:

Φ: describes the crack surface
Ψ: it is built so that its intersection with Φ is the crack front

The use of these two level sets allows the representation of the crack using only nodal data (no explicit representation of the crack is required).

The combination of this methodology with the enriched shape of functions allows the modelling of arbitrary crack growth without the necessity of remeshing.

Due to its high potential, this methodology has been implemented in several commercial codes such as Abaqus (Du, Z.). In this work, authors explore the performance of the XFEM code implemented in Abaqus 6.14 in fracture analysis of typical aeronautical structures.

Among the different XFEM approaches included in Abaqus, this paper will be focused on two of them:

1) XFEM Stationary Crack

XFEM Stationary Crack approach implemented in Abaqus considers full element enrichment (all the terms included in Equation 2 are taken into account). This formulation allows dealing with the asymptotic stress fields that appears near the crack tip.

This methodology can be used to calculate Stress Intensity Factors (SIFs) for arbitrary cracks without adapting the mesh to the crack path (cracks can cross finite elements and crack tip can be located inside a finite element).

The domain of the elements affected by the enriched formulation is defined by the following Abaqus keyword:

```
*ENRICHMENT, ELSET=Solid-1, NAME=Crack-1, TYPE=STATIONARY CRACK,
```

Crack position is defined as an Initial Condition:

```
*INITIAL CONDITIONS, TYPE=ENRICHMENT
```

SIF evaluation is based on contour integration around crack tip. This analysis is requested using the following keyword:

```
*CONTOUR INTEGRAL, CRACK NAME=Crack-1, CONTOURS=5, TYPE=K FACTORS, XFEM
```

The use of this formulation will result in obtaining a Stress Intensity Factor value for each integration contour (the user can define the number of contours to be evaluated) at each node located in the crack tip. In the examples included in the following sections this point will be discussed with more detail.

For result post-processing, level sets Φ and Ψ (define the crack surface and tip) and the XFEM status (intact, broken or partially broken) of the enrichment elements have to be requested:

```
*NODE OUTPUT
PHILSM, PSILSM
*ELEMENT OUTPUT
STATUSXFEM
```

2) XFEM Crack Propagation using Low-cycle Fatigue Analysis based on Linear Elastic Fracture Mechanics

The crack growth analyses included in this paper have been performed using the Low-cycle Fatigue Analysis based on Linear Elastic Fracture Mechanics (LEFM) implemented in Abaqus.

This methodology has been selected due to the following reasons:

- Low-cycle Fatigue Analysis is performed using Abaqus Direct-Cyclic approach what significantly reduces the computational costs of the analysis, as the damage growth can be extrapolated through several load cycles avoiding the necessity of evaluating cycle-by-cycle the full model.

- Classical analytic methodologies for crack growth analysis are mainly based on Linear Elastic Fracture Mechanics, so a great amount of material data is available to be used with LEFM approach.

By the other hand the use of this methodology has certain limitations:

- A reduced element enrichment formulation is used in the analysis as the term that takes into account the asymptotic stress field at the crack tip is not taken into account. As a consequence crack tip cannot be located inside an element.

- Linear elastic material behavior as the analysis is based on LEFM.

- Direct-Cyclic approach is not able to deal with problems in which changes in the status of the contacts between the different components are produced. This limitation also affects the contact between the crack surfaces (it can be very important if compression load states are applied)

- The efficiency of the Direct-Cyclic approach is highly reduced if complex loading spectra are applied as damage growth extrapolation through load cycles is not possible.

The domain of the elements affected by the enriched formulation is defined by the following Abaqus keyword:

```
*ENRICHMENT, NAME=Crack-1, ELSET=Solid-1.Solid, [TYPE=PROPAGATION
CRACK], INTERACTION=Crack
*SURFACE INTERACTION, NAME=Crack
```

Crack initial position is defined as an Initial Condition:

```
*INITIAL CONDITIONS, TYPE=ENRICHMENT
```

The LEFM criterion is defined by the following keyword. Paris' law in terms of strain energy release has to be defined by the user for the analyzed material in this keyword (Equation 2).

```
*FRACTURE CRITERION, TYPE=FATIGUE, [NORMAL DIRECTION=1]
```

$$\frac{da}{dN} = c_3 \Delta G^{c_4} \qquad (2)$$

where da/dN is the increment of crack length in the applied load cycle, ΔG is the incremental strain energy release and c_3, c_4 are material parameters.

The Low-cycle Fatigue Analysis (Direct-Cyclic) is defined by the following keyword:

```
*STEP
*DIRECT CYCLIC, FATIGUE
```

For result post-processing, additionally to the outputs requested for Stationary Crack approach, the incremental strain energy release is requested.

```
*ELEMENT OUTPUT
ENRRTXFEM
```

3. Practical applications

In this section the different cases that have been studied by the authors to evaluate Abaqus XFEM capabilities will be presented. A set of representative cases in aeronautical structures is selected to perform this evaluation. To evaluate the reliability of the method, the results obtained using XFEM approach will be compared with other methodologies such as analytic calculations.

3.1 Edge cracks

Cracks growing from the edge of a plate are a very common scenario in the Fracture Mechanics analyses performed for aeronautical structures. The configuration selected for the analysis is shown in the next figure:

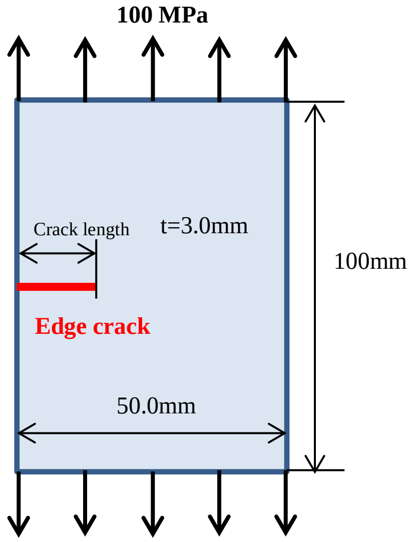

Figure 1: Edge crack scenario

For this configuration SIF calculation and crack growth analysis have been performed.
The model generated using Abaqus/CAE for the XFEM analysis of this configuration is shown in the next figure. A max principal stress plot is shown for a certain crack length.

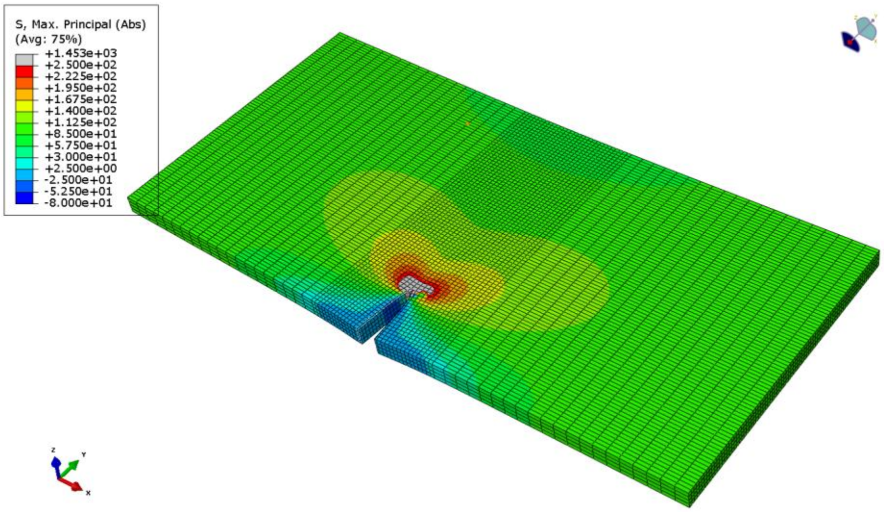

Figure 2: Edge crack. Abaqus model

Stress Intensity Factor calculation

SIF values have been calculated for several crack length / panel width ratios (from 0.1 to 0.6). The next table shows the detailed results obtained for a crack length / panel width ratio of 0.2. Results are shown for each through-thickness node and each integration contour.

Table 1: Detailed SIF results for crack length / panel width = 0.2

SIF – mode I (MPa√mm)		**Integration Contour**			
		1	**2**	**3**	**4**
Node	**1**	1288	852	890	910
	2	1261	755	807	824
	3	1270	769	816	834
	4	1261	755	807	824
	5	1288	852	890	910

As it can be seen, obtained values in each integration contour differ. So comparison with other approaches is required to determine which contour provides the best results.

XFEM results will be compared with the analytic results obtained from Rooke and Cartwrigth solution 1.1.20 (Rooke and Cartwright, 1974).

The analytic SIF result for the crack length / panel width ratio of 0.2 is 768 MPa√mm, which is very similar to the results provided by contours 2 and 3. The next figure shows a comparison between analytic values and XFEM results for contours 2 and 3 and the average between both contours for the different crack lengths.

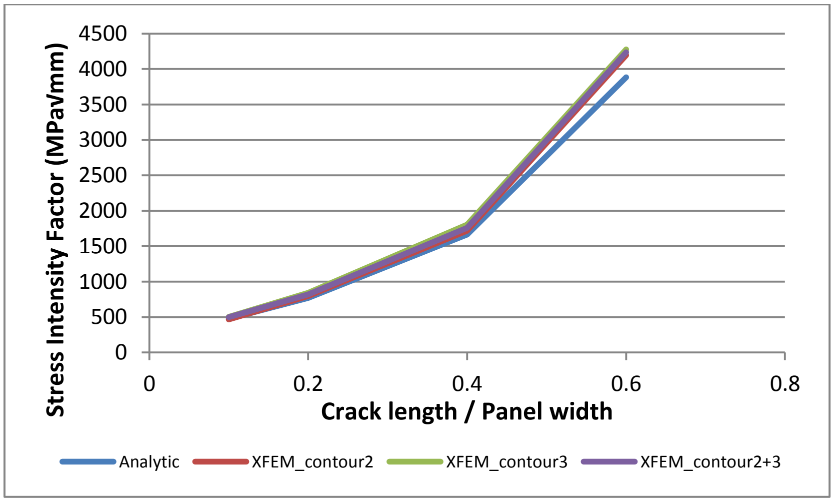

Figure 3: Edge crack. SIF results comparison

As it can be seen in the previous figure very accurate SIF results are obtained in contours 2 and 3, so the average value between both contours can be taken as reference result.

Crack Growth analysis

The next figure shows the comparison of the crack growth results for XFEM and analytic Linear Elastic Fracture Mechanics calculation.

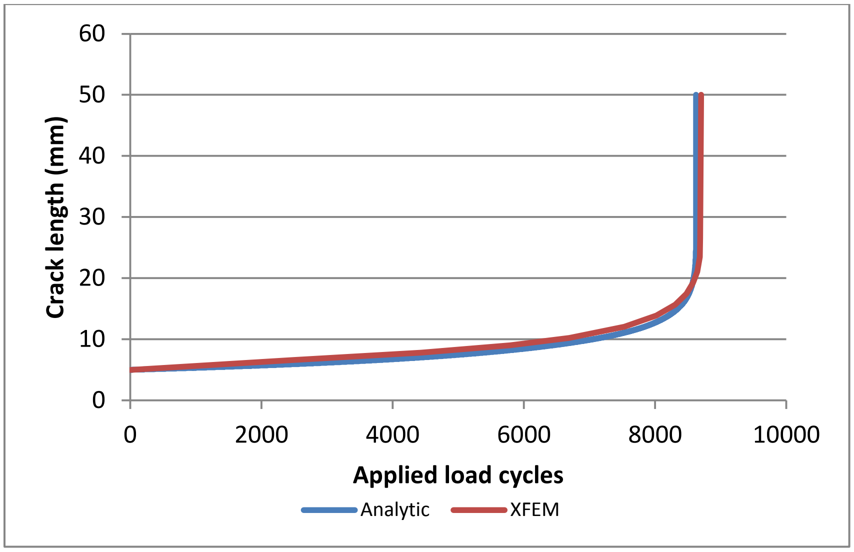

Figure 4: Edge crack. Crack Growth results comparison

A very good correlation is observed between both methodologies.

3.2 Hole cracks

Riveted joints are the most common configuration for structural attachments between aircraft components. Due to this fact rivet holes are present in most aeronautical structures. The high stress concentrations that appears at holes makes them susceptible to crack initiation what makes necessary to perform analyses of cracks growing from holes.

In this case two different configurations will be analyzed: unloaded and loaded hole.

Crack growing from unloaded hole

An open hole that is not transmitting load will be analyzed. The selected configuration is shown in the next figure:

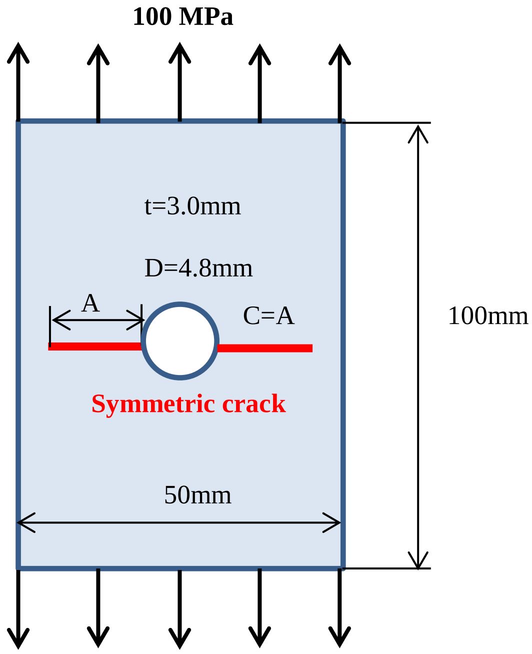

Figure 5: Unloaded hole crack scenario

The model generated using Abaqus/CAE for the XFEM analysis of this configuration is shown in the next figure. A max principal stress plot is shown for a certain crack length.

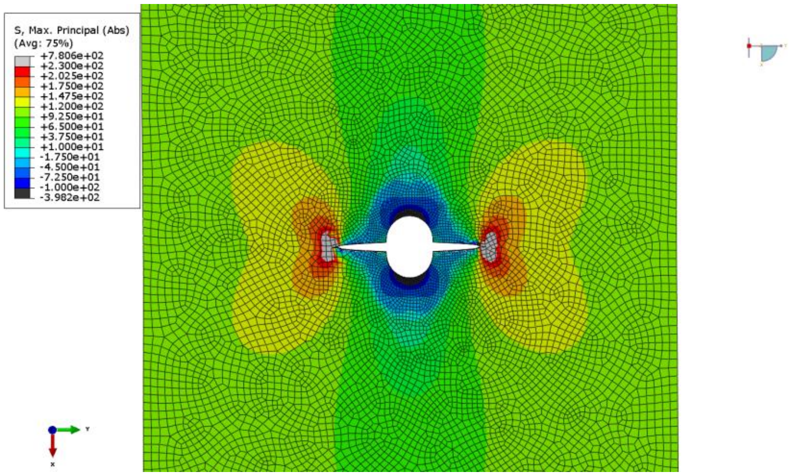

Figure 6: Unloaded hole crack. Abaqus model

SIF values have been calculated for several crack length / hole diameter ratios (from 0.5 to 3.0).

The next table shows the detailed results obtained for a crack length / hole diameter ratio of 1.0. Results are shown for each through-thickness node and each integration contour.

Table 2: Detailed SIF results for crack length / hole diameter = 1.0

SIF – mode I (MPa√mm)		Integration Contour				
		1	2	3	4	5
Node	1	786	518	552	561	564
	2	790	506	537	546	550
	3	802	520	549	558	560
	4	790	506	537	546	550
	5	786	518	552	561	564

XFEM results will be compared with the analytic results obtained from NASA/FLAGRO Hole Crack solution (JSC-22267, 1986) with the superposition of Rooke and Cartwright solution 1.1.5 (Rooke and Cartwright, 1974) to take into account the finite plate width effect.

The analytic SIF result for the crack length / hole diameter ratio of 1.0 is 518 MPa√mm, which is again very similar to the results provided by contours 2 and 3.
The next figure shows a comparison between analytic values and XFEM results for contours 2 and 3 and the average between both contours for the different crack lengths.

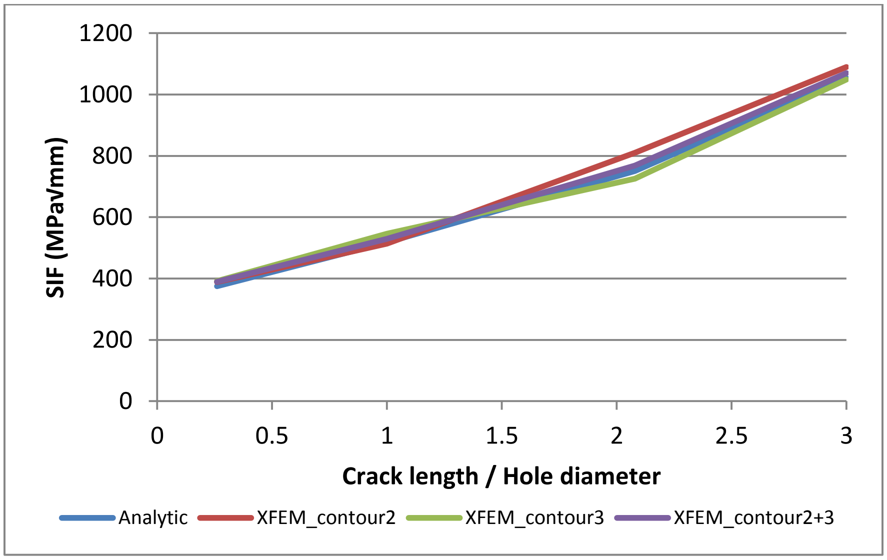

Figure 7: Unloaded hole crack. SIF results comparison

Average value between contours 2 and 3 provides the most accurate solution.

Crack growing from loaded hole

A hole that is transmitting load from a rivet will be analyzed. The selected configuration is the same that was analyzed for unload hole but adding the rivet load. Rivet load will be applied to the hole surface through a pressure distribution.

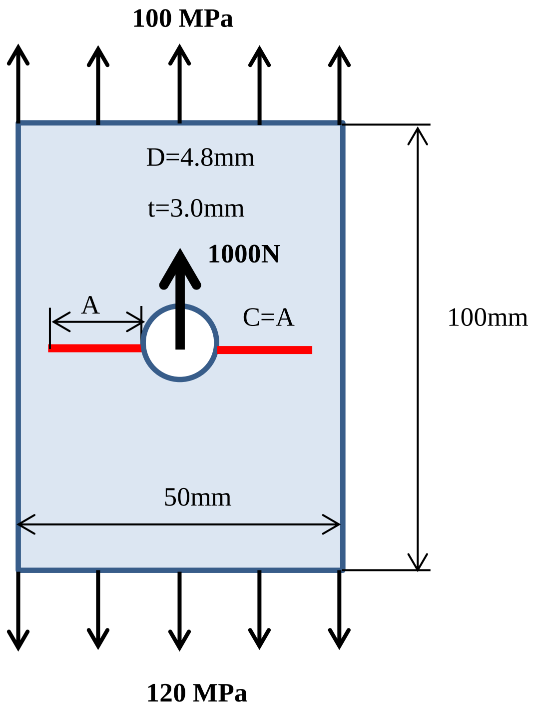

Figure 8: Loaded hole crack scenario

The model generated using Abaqus/CAE for the XFEM analysis of this configuration is shown in the next figure. A max principal stress plot is shown for a certain crack length.

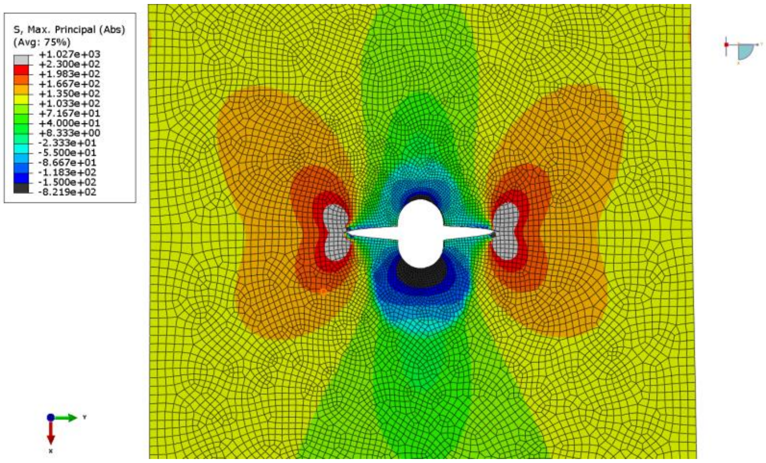

Figure 9: Loaded hole crack. Abaqus model

SIF values have been calculated for several crack length / hole diameter ratios (from 0.5 to 3.0). The next table shows the detailed results obtained for a crack length / hole diameter ratio of 1.0. Results are shown for each through-thickness node and each integration contour.

Table 3: Detailed SIF results for crack length / hole diameter = 1.0

SIF – mode I (MPa√mm)		Integration Contour				
		1	**2**	**3**	**4**	**5**
Node	**1**	806	679	717	697	712
	2	825	687	728	711	729
	3	844	707	744	728	744
	4	825	687	728	711	729
	5	806	679	717	697	712

The analytic SIF result for the crack length / hole diameter ratio of 1.0 is 672 MPa√mm (obtained using the same methodology that was used for unloaded hole), which is very similar to the results provided by contour 2.

The next figure shows a comparison between analytic values and XFEM results for contours 2 and 3 and the average between both contours for the different crack lengths.

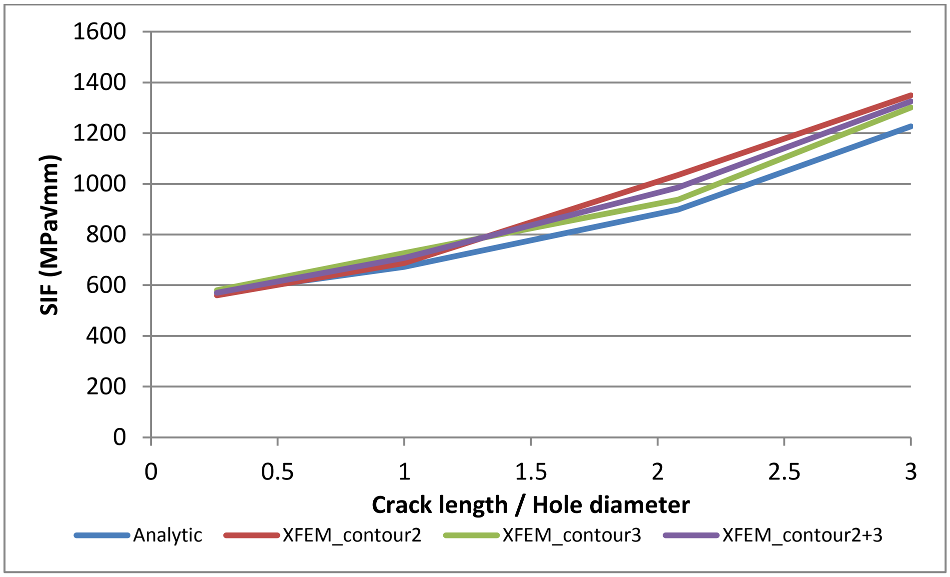

Figure 10: Loaded hole crack. SIF results comparison

Good correlation is observed, especially for small crack sizes. For large cracks XFEM results are more conservative than analytic approach.

Multicrack scenario

Riveted structural attachments are done through a set of fasteners, so several holes are made at the same structural component. To prevent widespread fatigue damage issues, scenarios with several cracks growing from different holes have to be evaluated. An evaluation of XFEM capabilities to deal with multicrack scenarios will be done in this section. A scenario consisting on two embedded cracks on a plate will be analyzed (this is a simplification of a scenario with cracks growing from two different holes when cracks are big enough to ignore the effect of the hole).

The next figure shows the analyzed configuration.

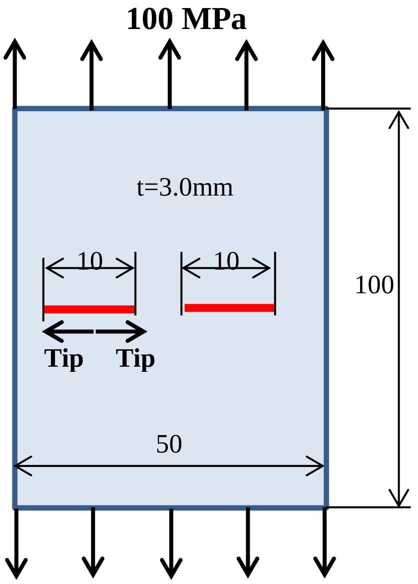

Figure 11: Multicrack scenario

The next figure shows the comparison of the crack growth results for XFEM and analytic Linear Elastic Fracture Mechanics calculation (superposition of Rooke and Cartwright solutions 1.1.5 and 1.2.3, (Rooke and Cartwright, 1974)).

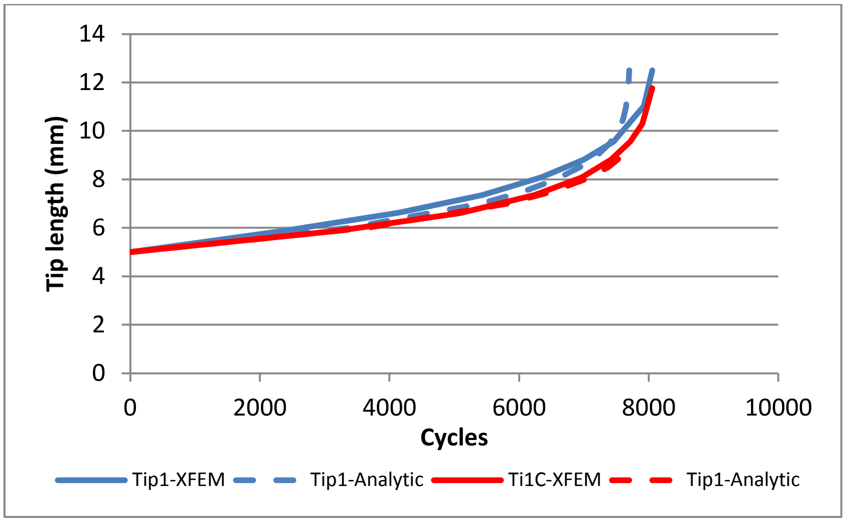

Figure 12: Multicrack scenario. Crack growth results comparison

A very good correlation between analytic and XFEM crack growth results is observed.

4. Conclusions

In this paper a set of practical cases from simple configurations to complex applications have been evaluated using the XFEM capabilities implemented in Abaqus 6.14.
This methodology has demonstrated to be robust and reliable when obtained results obtained for Stress Intensity Factor calculation and Crack Growth analysis have been compared with alternative approaches. These results allows being confident about the potential of XFEM to become in short period of time an alternative industrial standard approach in the Fracture Mechanics field.

It would report clear benefits to the aeronautical industry as the improvement of the accuracy of crack growth analysis directly reduce aircraft weight, as lighter structures can be designed by reducing conservative approaches, and maintenance costs, as more accurate crack growth analysis lead to lower structural inspection requirements.

However XFEM methodology currently available in Abaqus still requires development and improvement of its capabilities. Limitations associated to the methodology when dealing with contacts between components, material plasticity or complex loading spectra must be overcome to provide an analysis tool able to deal with every aspect of a complex Fracture Mechanics analysis.

5. References

1. Du, Z. eXtended Finite Element Method (XFEM) in Abaqus. ©Dassault Systèmes.
2. JSC-22267. Fatigue and Crack Growth Computer Program NASA/FLAGRO. 1986
3. Park, Jun. Failure Prediction without Prescribing Crack Paths by using XFEM in Abaqus. ©Dassault Systèmes. 2012
4. Rooke, D.P., Cartwright, D.J. Compendium of Stress Intensity Factors. 1974

6. Acknowledgments

This paper is a result of the fructiferous collaboration between the fatigue departments of Airbus Defence and Space and Safran Engineering Services. Going on with this collaboration will surely come out in great results.
The authors also want to acknowledge the support provided by Ginés Boluda, Eduardo Oslé, Ruben Tejerina and Fernando Sánchez from Airbus Defence and Space. Their contribution has made possible to match the objectives proposed for this work.
The authors want to thank Principia for all their support with the use of XFEM capabilities implemented in Abaqus.