30th Congress of the International Council of the Aeronautical Sciences
DCC, Daejeon, Korea ; September 25-30, 2016

XFEM-BASED FRACTURE MECHANICS ANALYSIS OF AERONAUTICAL STRUCTURES AFFECTED BY RESIDUAL STRESSES

Ismael Rivero Arévalo*, Javier Gómez-Escalonilla Martín*, Víctor Ramírez González, Álvaro García Martínez-Pardo**, Diego Garijo Hernández*****
***Airbus Defence and Space (Spain), **Safran Engineering Services (Spain), ***IMDEA Materials Institute (Spain)**

Abstract

Machining processes, overloads, dents or impacts induce residual stresses on metallic structures that can have a very important influence over the fatigue and crack growth behavior of the structure. Compressive residual stresses can extend the fatigue and crack propagation lives of the structure (this is the objective of manufacturing processes such as cold-working or shot-peening), but residual stresses of different nature can reduce the integrity and durability of the structure. This fact makes necessary to accurately predict the influence of these residual stresses, to both take benefit from them to improve the fatigue strength of a component and to define an appropriate maintenance program to avoid unsafe conditions.

The accurate evaluation of the influence of a residual stress field over the behavior of a crack propagating in a structure requires the use of numerical approaches. In this paper, the eXtended Finite Element Method (XFEM), introduced in 1999 by Belytschko et al [1] and Moës et al [2] is used to simulate the problem.

XFEM combines the benefits of the Finite Element Method (FEM) to analyze complex configurations with the capability of dealing with through-element discontinuities by enriching the degrees of freedom of the model with special displacement functions. The use of these functions avoids the necessity of re-meshing after each step of the crack growth analysis to adapt the mesh to the crack geometry, and relaxes the requirement of generating a very detailed mesh around the crack tip to properly capture the asymptotic stress field at this area. Thanks to this technology, the fracture mechanics analysis of a complex problem can be performed with acceptable computational cost.

The analyses included in this paper have been performed using the XFEM solutions implemented in DS Abaqus [3].

The authors of this paper performed a detailed evaluation and validation of XFEM capabilities for Linear Elastic Fracture Mechanics analyses in aeronautical structures [4]. In this new paper authors go further by applying the methodology to more complex problems affected by residual stresses.

This application will need to overcome some of the limitations of the current XFEM technology implemented in DS Abaqus, which are also discussed in this paper.

1. Introduction

The development of lighter and more efficient structures is a primary target in the aeronautical industry. Weight is one of the most critical factors in this kind of structures, since it is directly related with the payload an airplane is able to carry, and therefore with the operational profits.

Maintenance has also a very important contribution to the operational costs. To guarantee flight safety regular inspections and tests are carried out to check the structural integrity.

Several efforts have been made in order to develop more accurate methods to calculate and predict the behavior of the structure, in both analytical and computational fields. Especially in fatigue and damage tolerance area, the success of these efforts allows predicting with higher accuracy the process of initiation and development of cracks in the aircraft structure that leads to an improvement of the periodicity of the inspections as uncertainties are more controlled, reducing the associated costs.

In fatigue and damage tolerance field, the use of computational methods such as the finite element method is a standard practice for the calculation of Stress Concentration Factors for fatigue life. However for crack growth analysis, standard methods based on analytical calculations under conservative hypothesis are still mainly used, due to the limitations of finite elements method to deal with discontinuities.

To avoid these limitations new alternatives have been developed. One of them is the XFEM (eXtended Finite Element Method) what allows the simulation of the crack, solving the problem regarding through-element discontinuities.

In this paper, Abaqus 6.14 XFEM has been used to continue with the work started at [4]. The flexibility of Abaqus as a FEM solver combined with the XFEM potential has allowed the authors to develop new algorithms and methods to maximize the control over the crack growth and the stress concentration factor in a crack propagation analysis and overcome the limitations inherent to the XFEM methodology implemented in Abaqus.

2. XFEM method description

XFEM is included in the so called meshless methods (MM). These methods are devised to solve some of the problems related to the presence of a mesh for the approximation of the field variable in boundary value problems. XFEM is based on the enrichment of solution-type functions to the MM basis, allowing discontinuities through the elements of the mesh by adding additional degrees of freedom to the element nodes affected by this discontinuity.

The equation representing this is shown next:

$$\mathbf{u}^h(\mathbf{x}) = I + J + K \tag{1}$$

where:

$$I = \sum_{i \epsilon I} \phi_i(\mathbf{x})\,\mathbf{u}_i \tag{2}$$

Being u_i the nodal parameters of the entire set of nodes (Set I) and ϕ (x) the corresponding shape functions.

$$J = \sum_{j \epsilon J} \phi_j(\mathbf{x})\,\mathbf{b}_j H(\mathbf{x}) \tag{3}$$

Being b_j the nodal enriched degrees of freedom for the nodes of the elements that are fully cut by the crack (Set J), and H(x) the jump-function.

$$K = \sum_{h \epsilon K} \phi_k(\mathbf{x}) \left(\sum_{l=1}^{4} \mathbf{c}_k^l F_l(\mathbf{x}) \right) \tag{4}$$

Being c_k^l the nodal enriched degrees of freedom of the nodes of the element where the crack tip is located, and $F_l(x)$ adequate asymptotic functions for the displacement field near the crack tip (Set K).

Usually XFEM is combined with the use of level-sets for monitoring and postprocessing the crack without requiring geometric or mesh data. The crack geometry is defined by two almost-orthogonal signed distance functions. The first one, describes the crack surface (level-set Φ), while the second is used to construct an orthogonal surface so that the intersection of the two surfaces gives the crack front (level-set Ψ). No explicit representation of the boundaries or interfaces is needed because they are entirely described by the nodal data.

3. XFEM methodology implemented in Abaqus

Abaqus offers different approaches for XFEM calculations, depending on the desired output. For Fatigue and Damage Tolerance analyses, which are focused on calculating Stress Intensity Factors (or J-integrals) and crack propagations under cyclic loading, two approaches are available:

1) XFEM Stationary Crack

XFEM Stationary Crack approach implemented in Abaqus considers full element enrichment (all the terms included in Equation 1 are taken into account).

This methodology can be used to calculate Stress Intensity Factors (SIFs) or J-integrals 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). Calculation is based on contour integration around crack front (a ring submodel is automatically generated at crack front to allow contour integration).

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

This approach allows performing crack propagation analysis under cyclic loading spectra without pre-defining the crack path (solution-based crack growth).

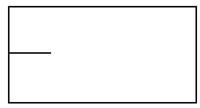
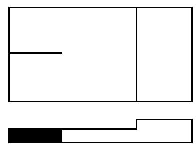
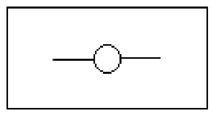
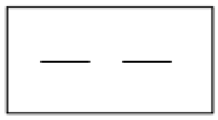
The analysis is performed using Abaqus Direct-Cyclic approach what significantly reduces the computational costs of the analysis, as the damage growth can be extrapolated through several load cycles avoiding the necessity of evaluating cycle-by-cycle the full model. Calculation of the crack length increment in each loading cycle is based on the Linear Elastic Fracture Mechanics principles.

4. XFEM validation

The authors performed a detailed validation of the Abaqus XFEM methodology by comparing the obtained results with consolidated analytic or numerical methodologies [4].

Next table shows some of the benchmark cases used for this validation (typical fracture mechanics analysis in aircraft structures).

Table 1: Validation cases

Problem	Description	
EDGE CRACK	Edge crack in a finite width plate under uniform axial stress	
THICKNESS CHANGE	Edge crack growing towards a thickness change.	
HOLE CRACK	Double crack in a loaded hole with by-pass stress	
EMBEDDED CRACKS	Two embedded cracks in a finite width plate.	

Very accurate results were obtained both for Stress Intensity Factor (SIF) calculation and crack propagation analysis, confirming the validity of XFEM methodology to perform Fracture Mechanics calculations.

Table 2: Validation cases – SIF Results

Problem	Crack length (mm)	$SIF_{XFEM}/SIF_{analytic}$
EDGE CRACK	10.0	1.07
	20.0	1.06
THICKNESS CHANGE	10.0	1.02
	20.0	0.96
HOLE CRACK	1.27	1.00
	4.8	1.05
EMBEDDED CRACKS	TipA=10	1.07
	TipB=10	1.09

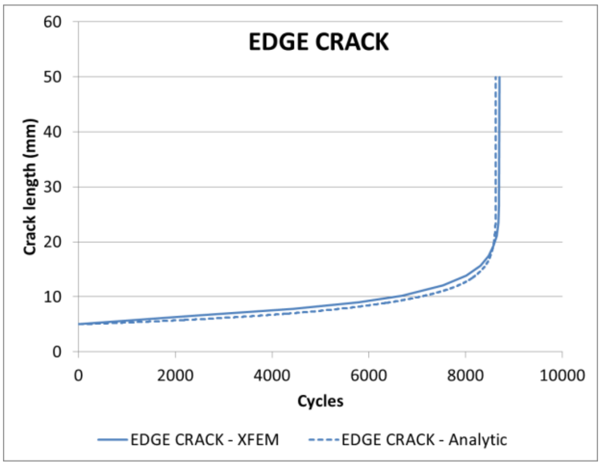


Figure 1: Edge Crack – Crack propagation results

5. Overcoming Abaqus XFEM limitations with iCracx

Abaqus offers a robust XFEM methodology to perform SIF and J-integral calculations (Stationary Crack approach described in section 3), however some major limitations affect the crack propagation simulations (Crack Propagation approach described in section 3). The most relevant limitations are the following:

- 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 (term K not included in Equation 1). As a consequence crack tip cannot be located inside an element (the crack has to propagate across an entire element at each crack increment).

- Linear elastic material behavior as the analysis is based on Linear Elastic Fracture Mechanics (elastoplastic behavior cannot be simulated).

- Abaqus only allows a linear modelization of the dadN curve (Paris' law)

- 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.

These limitations reduce significantly the capability of the software to perform these simulations for actual industrial cases. The authors of this paper have developed a Python software tool able to overcome all these limitations by taking benefit from the robustness of the XFEM stationary crack solution implemented in Abaqus combined with an alternative methodology for propagating cracks.

This software tool, which has been named **iCracx** (**i**mproved **Cr**ack **a**baqus **c**omputation with **x**fem) allows performing crack propagation simulations without a pre-defined crack path (solution-based propagation) calculating simultaneously Stress Intensity Factor and J-integral calculations without the limitations in terms of contacts, material model, loading spectra complexity… inherent to the approach implemented in Abaqus. Next figure shows the basic scheme of iCracx.

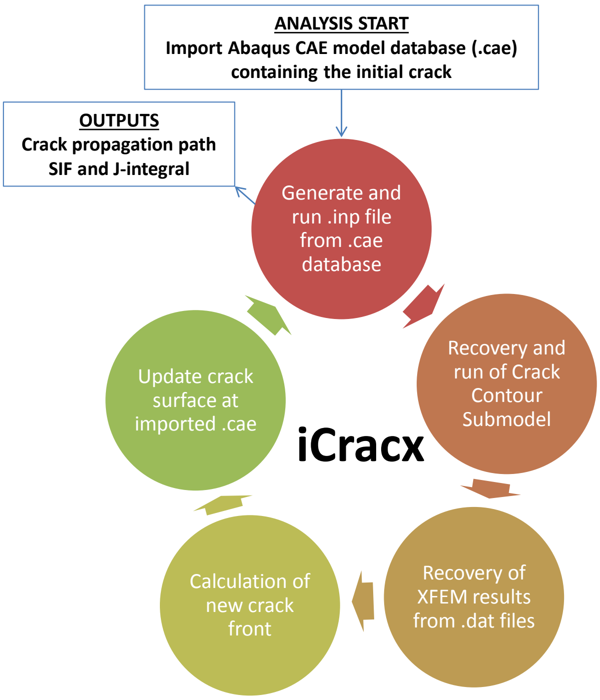


Figure 2: iCracx execution flow

6. Practical application of iCracx

Benchmark test

As an initial validation, iCracx has been tested against the EDGE CRACK validation case (not the same specimen geometry) described in section 4.

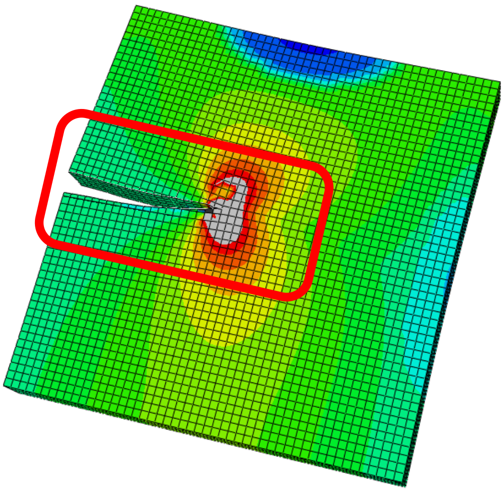


Figure 3: iCracx simulation for EDGE CRACK validation case

The comparison in terms of Stress Intensity Factors shows the accuracy of iCracx calculation.

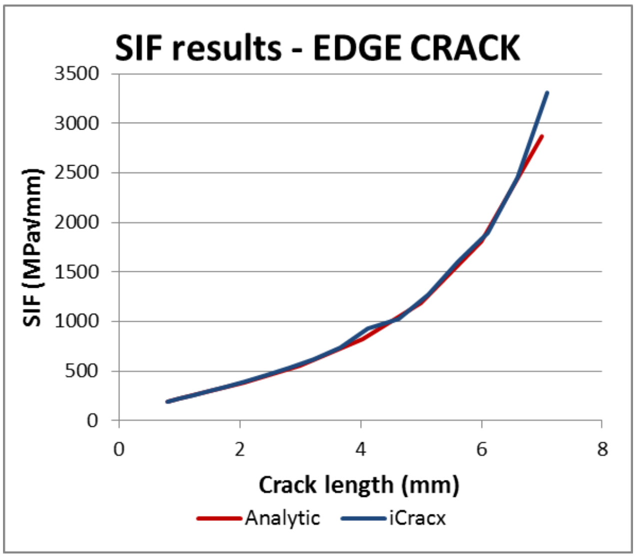


Figure 4: iCracx vs Analytic approach for EDGE CRACK validation case

The same EDGE CRACK case has been analyzed under Shear Loading to check the capabilities of iCracx to deal with mixed-mode crack growths. It can be observed in the next figure that the 45deg crack path orientation is clearly obtained in the iCracx simulation.

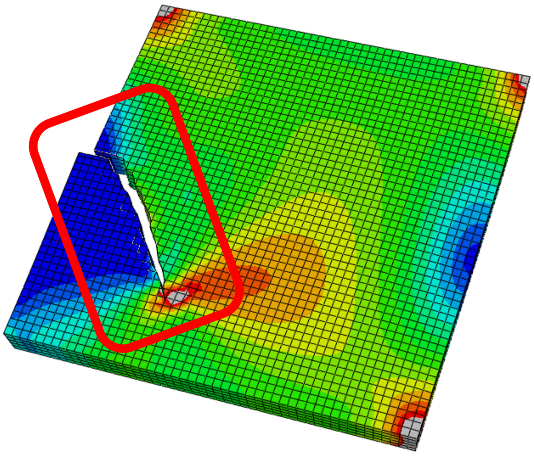


Figure 5: iCracx simulation for EDGE CRACK case under Shear Loading

A test case has also been performed to check the capability of iCracx to deal with non-planar geometries (valid also for configurations with important changes in thickness). This test case consists on an edge crack growing at a L-section profile.

The next figure shows the crack growing towards the profile fillet radius.

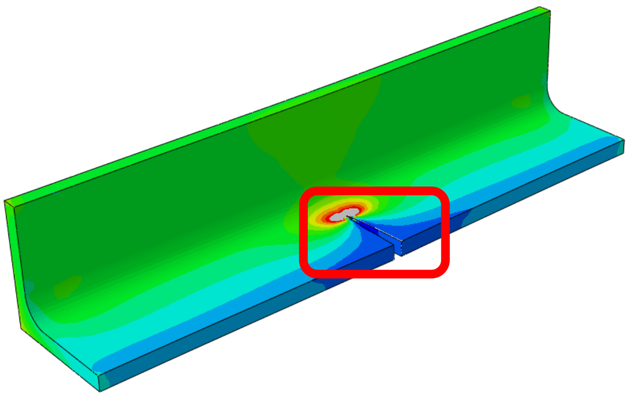


Figure 6: iCracx simulation for L-section profile. Crack growing towards fillet radius

In the next figure the crack can be seen growing through the fillet radius to the upper flange of the profile that confirms the capability of iCracx to deal with non-planar sections and thickness changes.

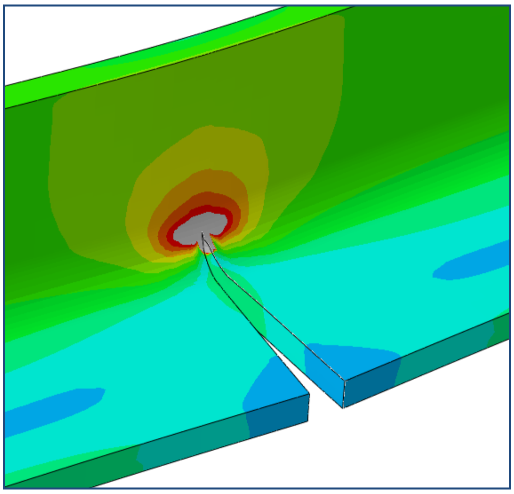


Figure 7: iCracx simulation for L-section profile. Crack growing through the fillet radius

After this initial validation for simple cases, the capabilities of iCracx have been tested in a practical problem for which XFEM methodology for cyclic crack propagation implemented in Abaqus is not able to provide a solution.

Crack propagation in contact-driven scenario

The selected case is a very typical configuration in aircraft structures. It consists on a lug-bolt attachment where contact condition between lug and bolt makes not possible to simulate crack propagation directly with the Abaqus XFEM approach, which is based on the direct-cyclic method as described in section 3.

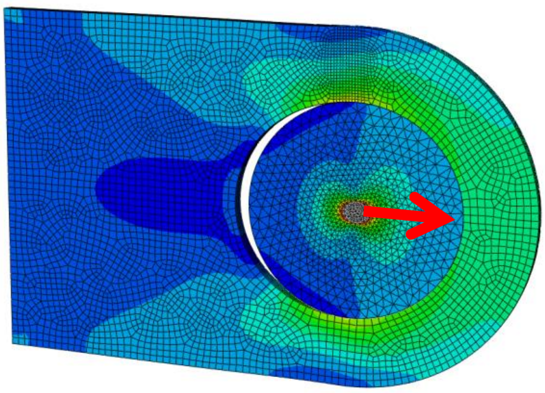


Figure 8: Lug-bolt model – uncracked condition

The simulation of the propagation of an initial crack located at the point of max stress concentration will be performed for two loading conditions: axial load spectrum (0deg) and transversal load spectrum (90 deg)

Axial loading

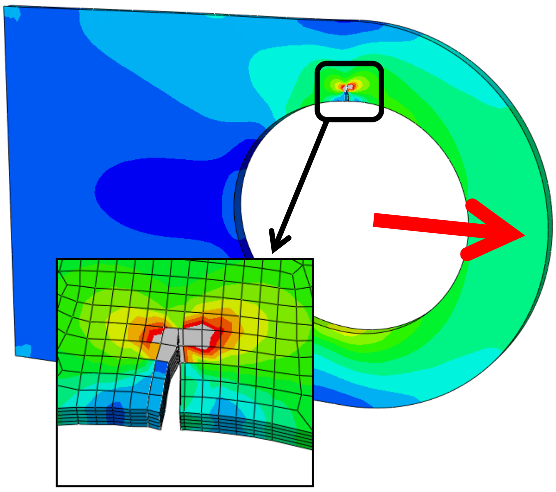


Figure 9: Initial crack – Axial loading

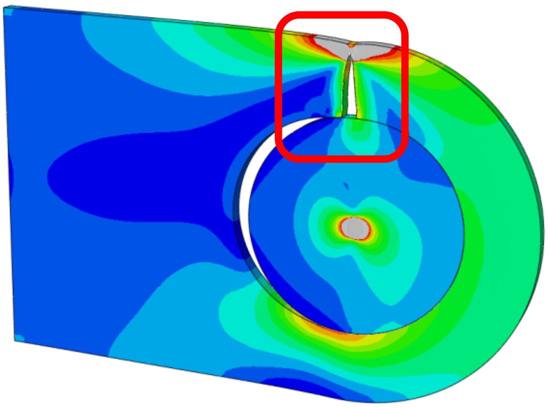


Figure 10: Propagation with iCracx – Axial loading

Transversal loading

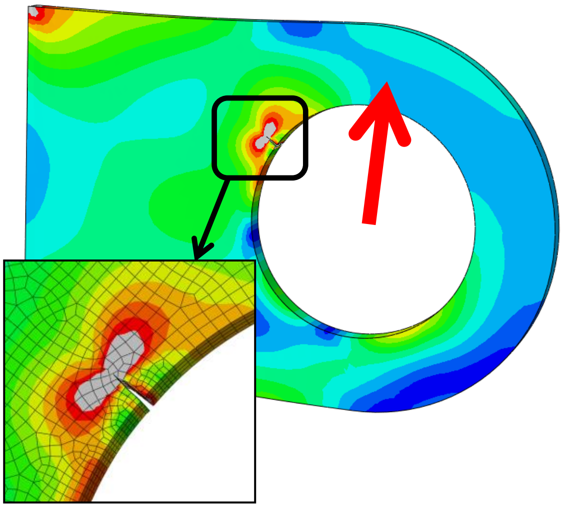


Figure 11: Initial crack – Transversal loading

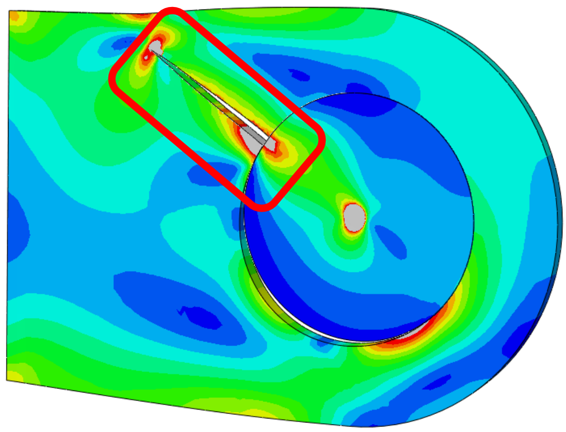


Figure 12: Propagation with iCracx – Transversal loading

Stress Intensity Factors calculated by XFEM are compared with the values obtained using the standard methodology at Airbus DS for axial loading condition to confirm the validity of the results obtained with iCracx.

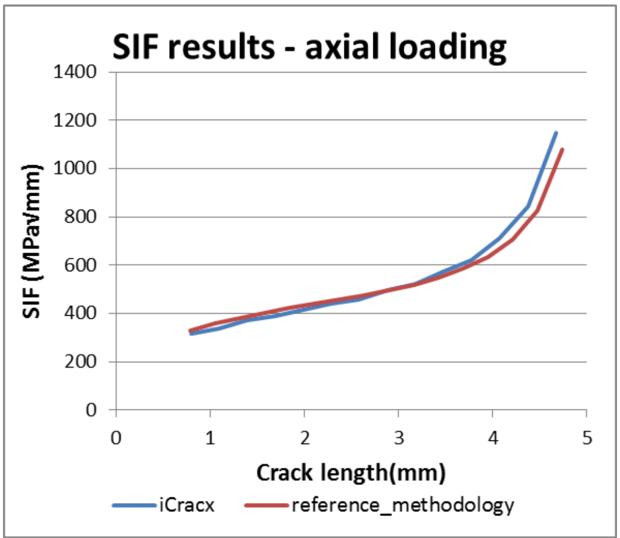


Figure 13: SIF comparison for axial loading

Crack propagation under residual stresses

Abaqus XFEM methodology for cyclic crack propagation is not able to properly deal with residual stress fields, specially compression stresses as the contact between the crack faces due to this compression cannot be simulated. This limitation is also overcome by iCracx.

For the previous lug-bolt model, a simulation of a simplified “cold-work” process has been performed to create a residual stress field at the lug and after this process the crack propagation simulation has been performed.

This process has been performed for two levels of cold-working to simulate two different residual stress conditions.

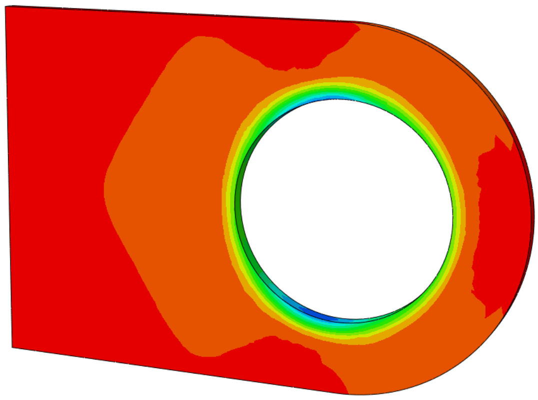


Figure 14: Residual stress field after “cold-work”

The crack growth simulation has been performed for the two loading conditions considered in the original analysis with no residual stresses (axial and transversal loading).

The compression residual stresses at the lug hole produce a “closure” effect at the crack tip that reduces the Stress Intensity Factor. This effect is clearly observed in the iCracx calculation.

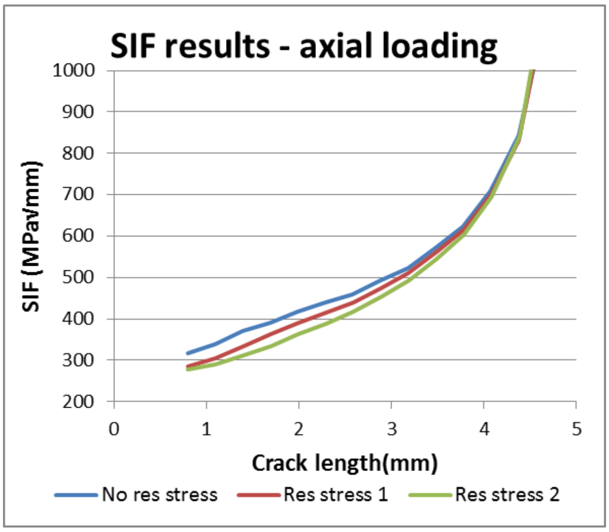


Figure 15: SIF comparison with residual stresses (axial loading)

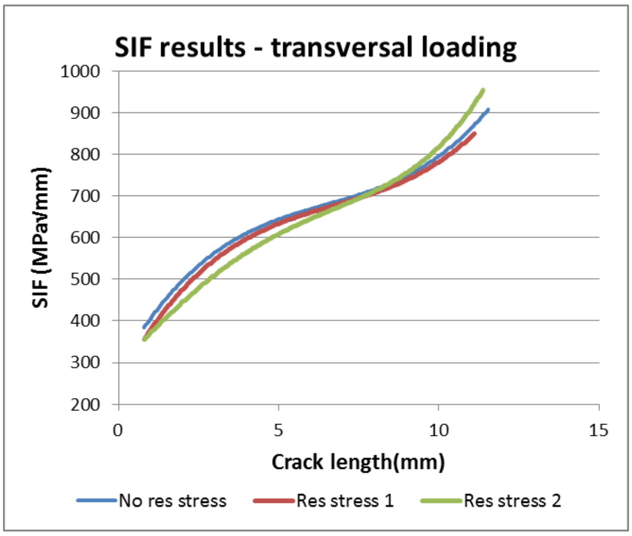


Figure 16: SIF comparison with residual stresses (transversal loading)

This SIF reduction leads to an improvement in the crack growth life of the component as it can be seen in the iCracx simulation results. This is the beneficial effect produced by the "cold-working" processes.

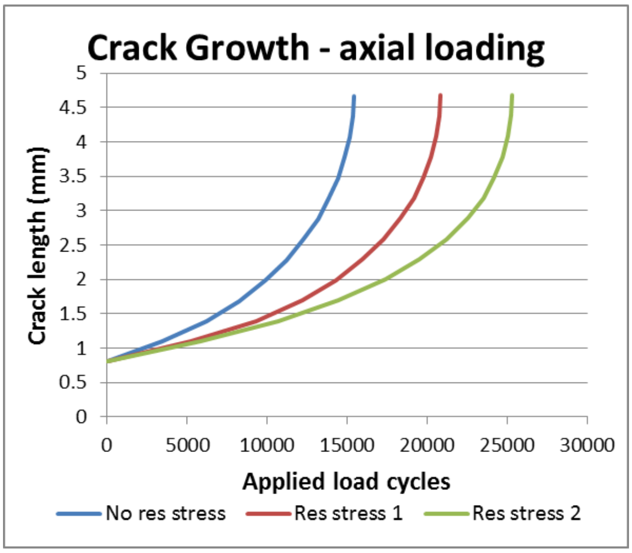


Figure 17: Crack propagation curves (axial loading)

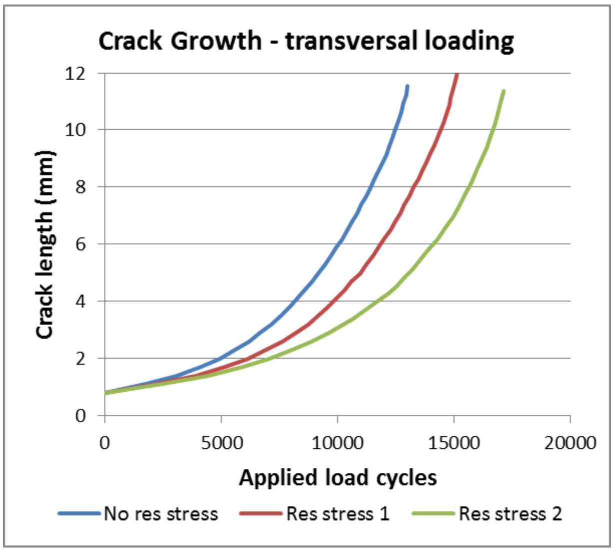


Figure 18: Crack propagation curves (transversal loading)

The results obtained with iCracx in terms of SIF and crack growth lives are in line with Airbus DS and Safran experience with residual stresses influence in Fracture Mechanics. Quantitative validation based on test evidence will be performed on a future work.

7. Conclusions

The implementation of XFEM methodology in the Fatigue and Damage Tolerance (F&DT) analysis of aircraft structures provides a powerful tool able to analyze complex cases with higher levels of accuracy than those provided by conventional approaches as conservatism can be reduced.

However XFEM approach implemented in Abaqus current versions has some important limitations that reduce significantly the potential benefits of the methodology.

The authors, with the development of iCracx, are taking benefit from all the potential of Abaqus XFEM capabilities supplementing those points at which Abaqus is not robust enough. The result is a software tool able to perform crack growth simulations at which the crack path is not pre-defined for a wide set of problems (complex geometries, residual stresses, contact conditions…) for which standard approaches are usually not able to provide accurate enough solutions.

The accurate simulations provided by iCracx allow predicting with higher confidence the crack growth behavior in an aircraft structure, making possible the optimization of weight, cost and maintenance requirements.

Future developments associated to iCracx will pursue the optimization of the software flow, capability to handle multicrack scenarios, smooth integration with Airbus DS and Safran F&DT standard tools and validation for a wider set of practical cases.

Acknowledgments

The authors want to recognize the continuous support provided by their colleagues from Airbus Defence and Space and Safran Engineering Services, especially from Juan Muñoz and Fernando Sánchez. Without their contribution it would not have been possible to reach the goal of this work.

References

[1] Belytschko, T.; Black, T. (1999). Elastic crack growth in finite elements with minimal remeshing: Int. J. Numer. Methods Eng., 45(5), 601– 620.

[2] Moës, N.; Dolbow, J.; Belytschko, T. (1999). A finite element method for crack growth without remeshing: Int. J. Numer. Methods Eng., 46, 131–150.

[3] Du, Z. eXtended Finite Element Method (XFEM) in Abaqus. ©Dassault Systèmes.

[4] Rivero, I.; Gómez-Escalonilla, J.; García, A.; Ramírez, V.; Garijo, D. (2015). XFEM Application to Crack Growth Correlation in Aeronautical Structures. NAFEMS World Congress 2015.

Copyright Statement

Contact Author Email Address

In case of interest, the authors can be contacted by sending a mail to:

ismael.rivero-arevalo@airbus.com