U.P.B. Sci. Bull., Series D, Vol. 71, Iss. 3, 2009 ISSN 1454-2358

FATIGUE LIFE PREDICTION OF A CRACKED LAP SPLICE SPECIMEN USING FRACTURE MECHANICS PARAMETERS

Pedro M.G.P. Moreira[1], Paulo F.P. de Matos[2],
Ştefan D. Pastramă[3], Pedro P. Camanho[4], Paulo M.S.T. de Castro[5]

Una dintre problemele importante ale structurilor din industria aeronautică este inițierea şi creşterea fisurilor sub acțiunea sarcinilor variabile. Acest fenomen este accentuat în zonele cu concentratoare de tensiune, ca de exemplu în zonele în care panourile de fuselaj sunt asamblate cu nituri. Lucrarea prezintă o analiză numerică tridimensională utilizând metoda elementelor finite, pentru studiul unei epruvete tip "lap-splice"asamblată cu ajutorul a trei nituri. Scopul analizei este calibrarea factorului de intensitate a tensiunii la vârful fisurilor ce se dezvoltă la marginea găurilor de nit, în zona slăbită a secțiunii transversale a epruvetei. Valorile acestui parametru s-au calculat pentru cazurile unei singure fisuri sau a două fisuri simetrice si au fost utilizate pentru determinarea ulterioară a dimensiunii inițiale echivalente a fisurii (EIFS) şi pentru predicția duratei de viață a epruvetei sub acțiunea sarcinilor variabile.

One of the main problems of aeronautical structures is the onset and growth of damage due to fatigue loading. This phenomenon is accentuated in areas of stress concentration, for example in the connection of components, such as the riveted joints of fuselage panels. This paper is focused on the study of a single-lap splice with three rivet rows and one rivet column. A three dimensional stress analysis using the Finite Element Method was carried out to determine the stress intensity factors for a single crack and symmetric cracks starting from the edge of the hole located at the critical cross section. The obtained values were used together with experimental data in order to obtain the Equivalent Initial Flaw Size (EIFS) and then to predict the fatigue life of the specimen.

Keywords: Stress intensity factor, Lap splice, Equivalent Initial Flaw Size.

[1] Dr. Eng, INEGI - Institute of Mechanical Engineering and Industrial Management, Porto, Portugal
[2] Dr. Eng, IDMEC- Institute of Mechanical Engineering, Porto, Portugal
[3] Professor, Department of Strength of Materials, University POLITEHNICA of Bucharest, Romania, e-mail: spastrama@resist.pub.ro
[4] Associate Professor, Department of Mechanical Engineering and Industrial Management, University of Porto, Portugal.
[5] Professor, Department of Mechanical Engineering and Industrial Management, University of Porto, Portugal.

1. Introduction

Riveted thin sheets are often encountered in the aeronautical industry. The rivet holes are the most likely places for crack initiation and fatigue growth, due to the high stress concentration that occurs in this area. For certain loading conditions, crack propagation may occur, leading to the final failure of the structure. In order to avoid such accidents, it is necessary to know the stress distribution and stress intensity factor (SIF). For a fracture mechanics study of riveted joints, several geometric configurations are currently used, [1].

In this paper, a lap splice with three rivet rows and one rivet column, subjected to tensile load is studied in order to determine fracture mechanics parameters necessary for subsequent fatigue calculations, by using a three dimensional finite element analysis.

A detailed analysis involving crack propagation requires a stress intensity factor calibration. Due to the complexity and the three dimensional nature of the problem, this calibration is not commonly available.

Two types of cracks that appear at the critical cross section, containing the first rivet are studied: a single crack and two symmetric cracks. The specimen bending, due to the eccentricity of the joint, leading to a non-uniform stress distribution along the specimen thickness, is taken into account in the developed models.

With the results of the finite element analysis, the equivalent initial flaw size (EIFS), for 45 specimens tested at $R = \sigma_{min}/\sigma_{max} = 0.05$ and $\sigma_{max} = 160$ MPa is calculated, and used to predict the fatigue behavior of the studied joints.

2. Description of the specimen

The structure under investigation is shown in Figure 1. The characteristics of the lap joint are:

Specimen type: Lap splice with three rivets rows and one rivet column;
Material: Aluminum 2024-T3 Alclad sheets;
Specimen length: 260 mm, 60 mm overlap;
Sheet thickness: 1.2 mm;
Specimen width: 20 mm (corresponding to a normal rivet pitch of a fuselage);
Rivets: NAS 1097 AD4, 3.2 x 7 mm (no countersunk);
Rivet material: Aluminum 2117-T4;
Cracks: symmetric cracks and single crack (Fig. 2).

In order to determine the mechanical characteristics of the material, tensile tests were performed, leading to the following results: Young's modulus E = 70.61 GPa and Poisson's ratio ν = 0.33.

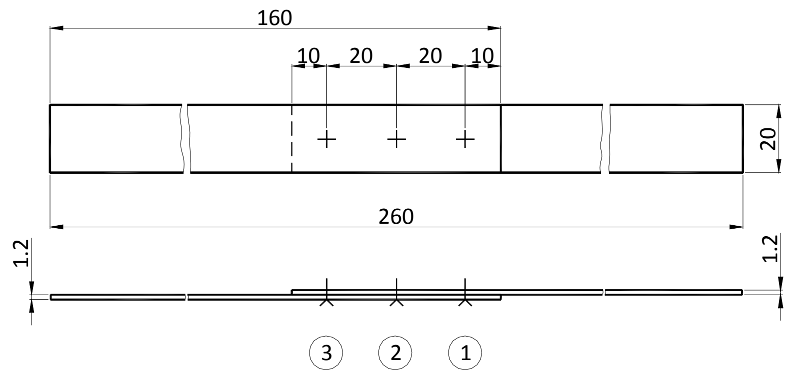

Fig. 1. The lap splice specimen

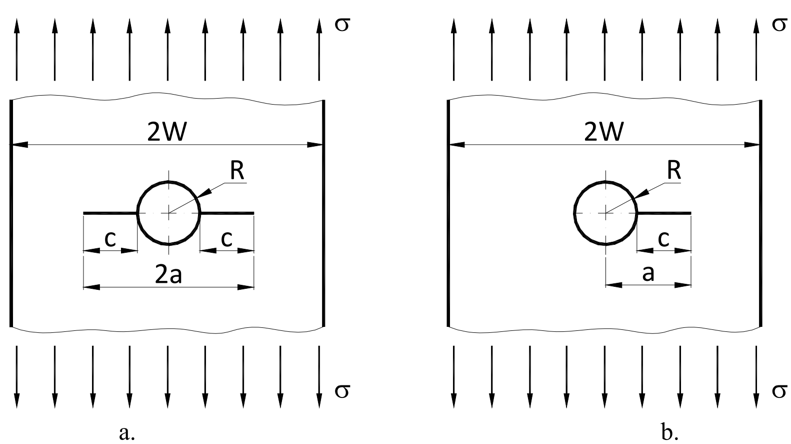

Fig. 2. Crack configurations: a. Symmetric cracks; b. Single crack

3. The finite element analyses

The complexity of the specimen geometry and the presence of bending due to the eccentricity of the joint, require a three dimensional stress analysis. For that purpose, the finite element code ABAQUS [2] was used.

The plate was modeled with 20-node brick isoparametric elements. Each plate contains 3648 elements and has two elements through the thickness. The rivets were modeled with 8-node brick elements (C3D8). Each rivet contains 2448 elements. The joint was modeled with no interference (perfect fit joint). Friction between the lap joint components was not taken into account. A detail of the mesh in the contact area is shown in Fig. 3.

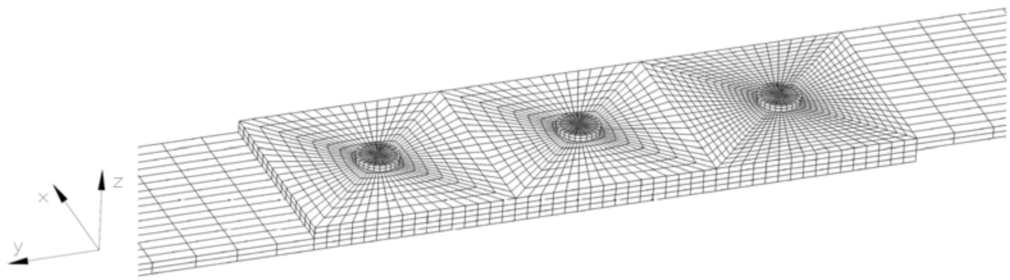

Fig. 3. Mesh detail in the contact area

Several contact surfaces were modeled using 27-node brick elements (C3D27): contact between the two plates to avoid the interpenetration due to bending; contact between the rivet and plates and contact between rivet head and each plate. A total of 13 contact pair surfaces were created.

The boundary conditions used in the model are shown in Figure 4. The displacement in the z direction on three lines of elements along the Ox axis were constrained in both plates, approximately on a length of 25.69 mm, equal to the grips used in the fatigue tests carried out for similar specimens.

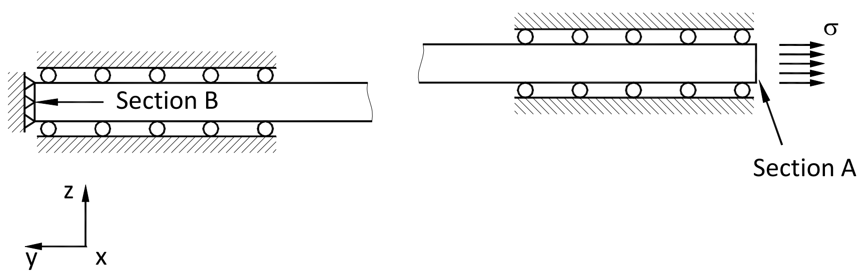

Fig. 4. The boundary conditions.

The remote load was applied in section A of the top plate. In section B of the bottom plate, nodes were constrained on all the three directions. The deformed model for the case of full load is shown in Fig. 5. The load eccentricity effect is evident in this figure, where displacements were enlarged in post-processing of the FEM analysis, to highlight this effect. A detail of the deformed model in the separation area is shown in Fig. 6.

Fig. 5. The deformed model

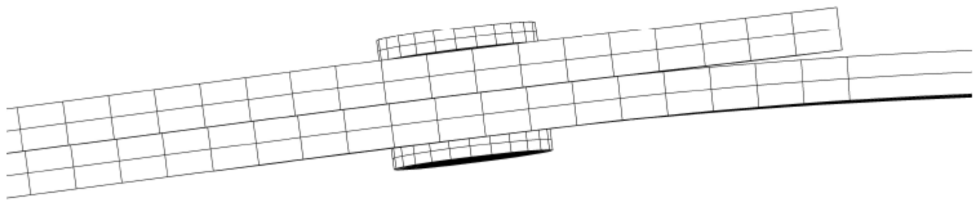

Fig. 6. Detail of the deformed model, showing the separation of the plates

4. Evaluation of the stress intensity factor

In a finite element analysis for a cracked structure, care should be taken for a proper modeling of the crack. In an elastic analysis, the stresses near the crack tip have a $1/\sqrt{r}$ singularity, which can be modeled using collapsed (singular) elements. This means one side of the element is collapsed to the crack tip, and all three nodes on this side have the same geometric location. Also, the mid-side nodes on the sides connected to the crack tip are moved to a point at a distance from the crack tip which is a quarter of the side length (the so called quarter node point technique). [citation required]

In this analysis, the elements near the crack tip were changed according to the above-mentioned technique, as shown in Fig. 7. The crack tip is represented by the white dots and the quarter nodes are represented by the light grey dots.

The results are presented in two different ways. In the first study, stress intensity factor results for five coordinates along the thickness are calculated. Then, using the three values for each element, and according to Fig. 8, an average value of the stress intensity factor is calculated with the equation, [2]:

$$K_{average} = \frac{K_A + 4K_B + K_C}{6}. \tag{1}$$

Since this is a mixed mode situation, the effective stress intensity factor K_{eff} can be calculated using the following relationship, [3]:

$$K_{eff} = \sqrt{K_I^2 + K_{II}^2 + \frac{K_{III}^2}{1-\nu}}. \tag{2}$$

The SIFs K_I , K_{II} and K_{III} were determined using the J-integral.

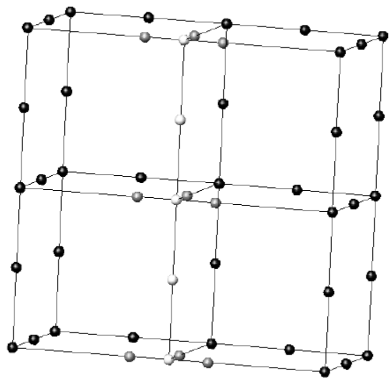

Fig. 7. Singular elements at the crack tip

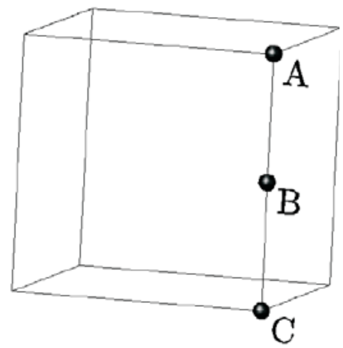

Fig. 8. Locations for determining the average SIF

The J integral is a contour integral, introduced by Eshelby [4] and Rice [5]. This parameter avoids the difficulties of the characterization of the stress field in the extremity of the crack using local parameters such as the crack opening displacement (COD), because its value is independent of the chosen contour Γ surrounding the crack tip, Fig. 9.

The analytical definition of J integral is presented in equation:

$$J = \int_{\Gamma} \left(w dy - \vec{T} \frac{\partial \vec{u}}{\partial x} ds \right), \tag{3}$$

where w is the density of deformation energy in points on the contour, $\vec{T}$ is the traction vector, $\vec{u}$ is the displacement and ds the element of the contour Γ.

For the calculation of the J integral in a finite element analysis using the ABAQUS code [2], the contour Γ is described through rings of elements around the crack tip, Fig. 10.

Different contours (domains) are created. The first contour consists of the elements linked directly to the nodes of the extremity of the crack. The following contour consists of a ring of elements in contact with the first. Each subsequent contour is defined by the next ring of elements.

The values of K_I , K_{II} and K_{III} are presented in MPa and the value of K_{eff} is calculated in the non-dimensional form as:

$$k_{eff} = \frac{K_{eff}}{\sigma\sqrt{\pi a}}, \tag{4}$$

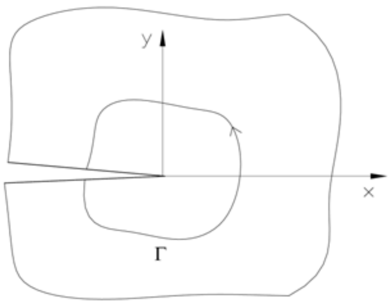

Fig. 9. Contour for determination of the J-integral

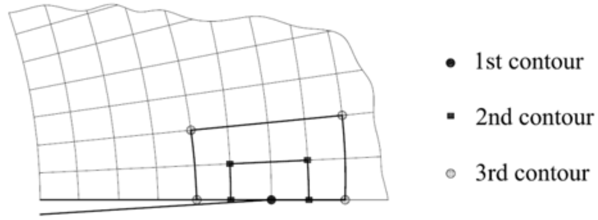

Fig. 10. Contours used in ABAQUS 2D for the calculation of J integral

For the cases of symmetric cracks and single crack, the K_I values along the thickness of the cracked plate are presented in Figs. 11 and 12 as a function of the crack length a (the crack length includes the hole radius).

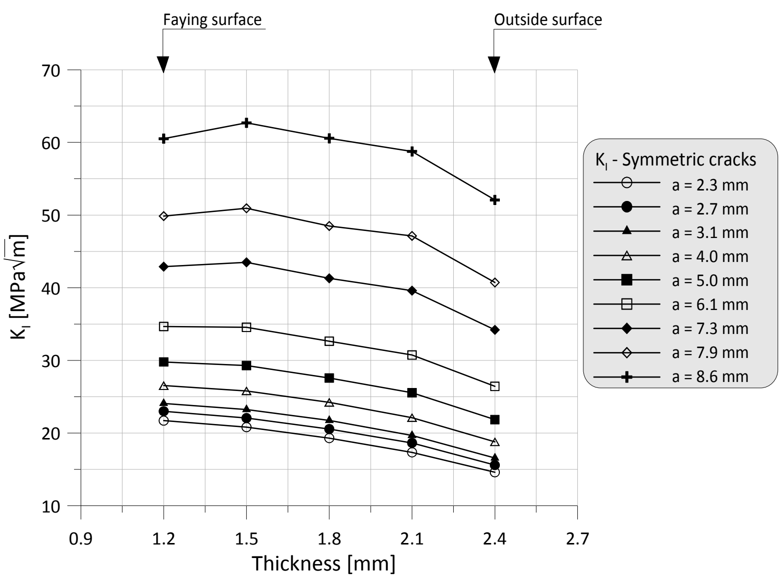

Fig. 11. K_I along the thickness for several crack lengths, symmetric crack

The results show that the stress intensity factor is maximum at or near the faying surface and minimum at the outside surface, for all crack lengths. This result is related with the bending of the plate, that tends to open more the edge of the crack located at the faying surface when compared with the edge of the crack located at outside surface. For symmetric cracks, the major difference between the stress intensity factor at the faying and outside surfaces is 17%, and occurs for a crack length of 4 mm. For a single crack, the major difference is 21%, also occurring for a crack length of 4 mm.

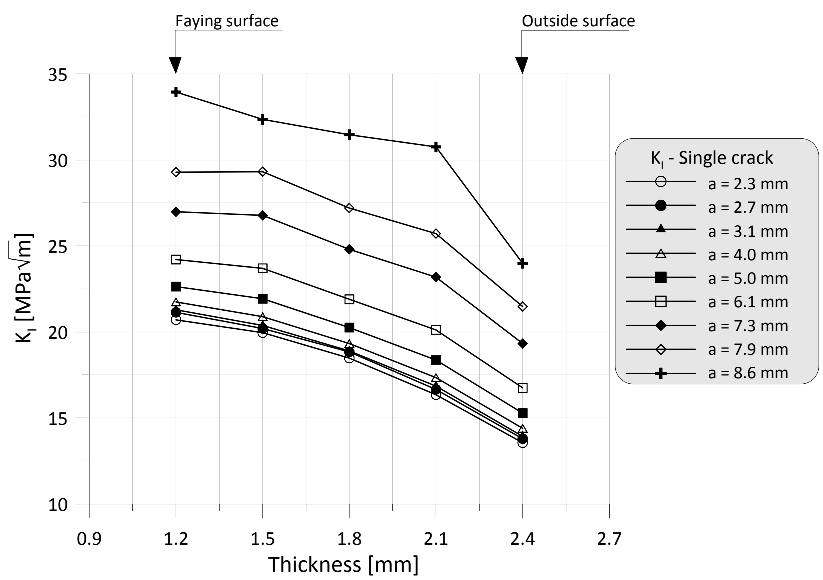

Fig. 12. K_I along the thickness for several crack lengths, single crack.

The result for the non-dimensional values k_{eff} according to equations (2) and (4) for several nondimensional crack lengths c/R, is presented in Fig. 13.

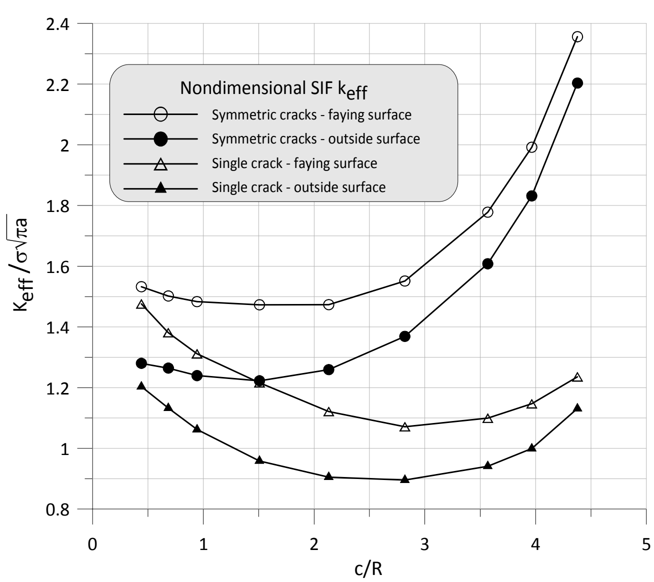

Fig. 13. k_{eff} for several crack lengths, symmetric cracks and single crack.

It can be concluded that, for a given crack length and surface, the stress intensity factor of symmetric cracks is always higher than the one of a single crack. Considering static loading, the condition for unstable propagation of symmetric cracks ($\partial K/\partial a > 0$) occurs at a crack length of 4 mm (outside surface), and 5 mm (faying surface). For a single crack, the condition of unstable propagation occurs at a crack length of 5 mm, for both the faying and outside surfaces.

The stress intensity factors K_{II} and K_{III} have a very small influence in the value of k_{eff}. This is shown in Fig. 14, where the values of k_{eff} and $k_I = K_I/\sigma\sqrt{\pi a}$ for the case of symmetric cracks are plotted together for comparison.

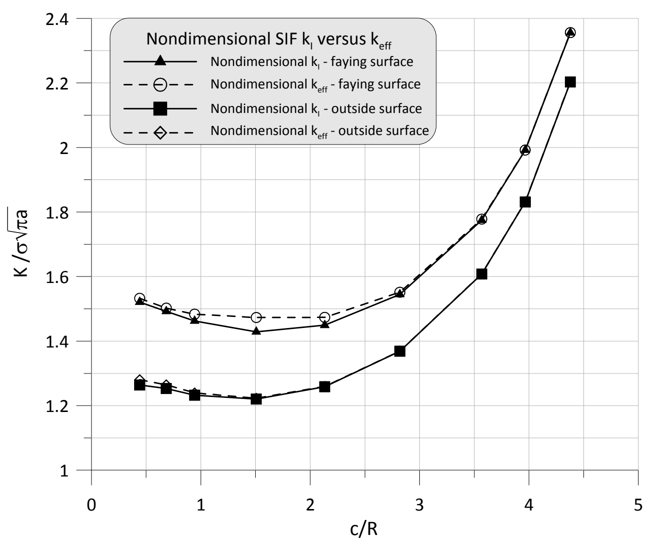

Fig. 14. k_{eff} and k_I for several crack lengths, symmetric cracks.

5. Equivalent Initial Flaw Size

The determination of the equivalent initial flaw size (EIFS) is a direct application of the SIF calibration obtained with the finite element analysis presented above. The EIFS is the size of an initial flaw that under a given cyclic loading would lead to the real number of cycles to rupture (fatigue life). In general, the EIFS depends on the test variables, fractographic results, crack size range over which the fractography has been carried out, form of crack growth model used for the back extrapolation, goodness of fit of the model to the fractographic data, and nature of the surfaces from which cracking was initiated [6]. Clearly, an EIFS should only be used with the methods and data from which it was derived. To evaluate the value of the EIFS the back extrapolation technique and the Paris fatigue crack growth rules for long-crack regime [7].

$$\frac{da}{dN} = C(\Delta K)^m, \quad (5)$$

were used. In equation (5), da/dN is the fatigue crack grow rate, $\Delta K = K_{max} - K_{min}$ is the variation of the stress intensity factor in a loading cycle and C, m are constants depending on the material, environment, frequency, temperature and stress ratio.

Values m = 3.3736 and $C = 2.7117 \cdot 10^{-11}$, obtained from [8] were used. The value of EIFS (c) for the symmetric crack was calculated. The SIF calibration for the faying surface (maximum value) was used. Fitting the k_{eff} values, normalized with the crack length c) for the faying surface, the following equation was obtained:

$$k_{eff} = \frac{K_{eff}}{\sigma\sqrt{\pi c}} = 1.6014 - 0.1261c + 0.0402c^2 - \\ - 0.0056c^3 + 0.00025c^4 + 0.000057931c^5 \quad (6)$$

The EIFS study was carried out taking in to account the experimental values obtained by IDMEC Porto in the fatigue life tests and final crack length, presented in [9]. This calculation of EIFS was done in 41 specimens.

Using the Paris law, it was possible to rebuild the fatigue crack propagation between the EIFS and the rupture moment. This representation is presented in Fig. 15.

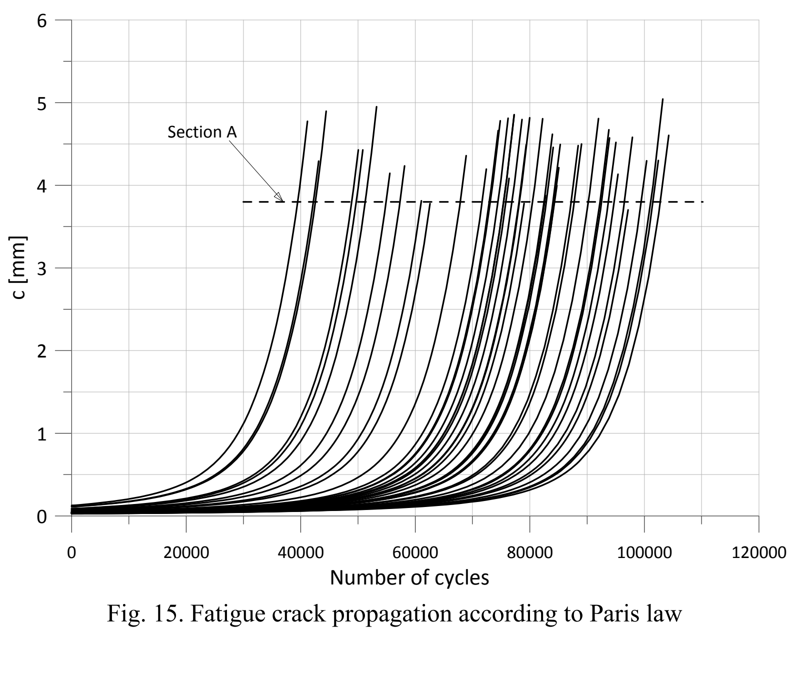

Fig. 15. Fatigue crack propagation according to Paris law

The average value of EIFS was 37.6 μm. With this EIFS, it is possible to predict fatigue life at any stress level by extrapolation. To validate the predicted fatigue life, data obtained at the Univ. of Pisa (UP), Univ. of Naples (UNAP) and IDMEC Porto, presented in [10] was used. Identical specimens at 90, 120 and 160 MPa with the same value of R were tested. The results obtained for the fatigue life are presented in Fig. 16.

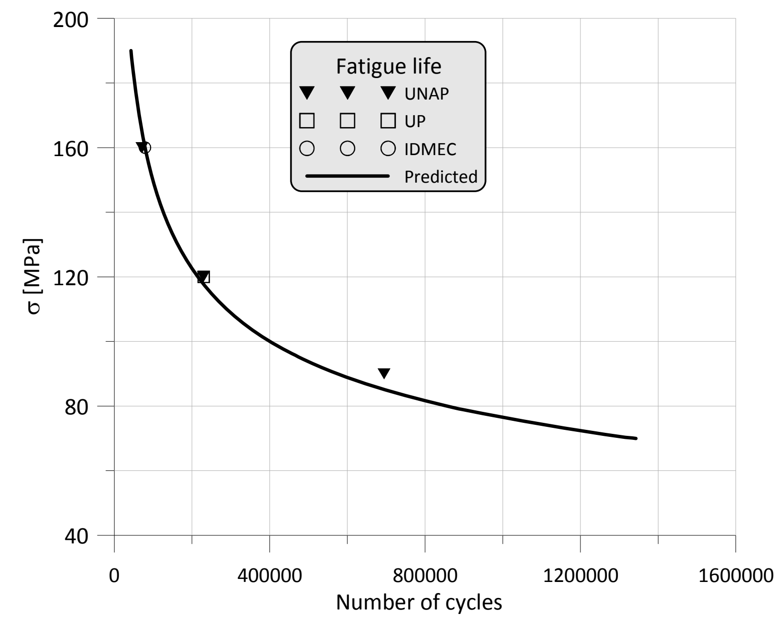

Fig. 16. Predicted fatigue life for EIFS determined with results obtained by IDMEC

This study shows the ability of the EIFS concept to predict fatigue life, for a given structural detail.

6. Conclusions

A detailed analysis of a single shear, rivet lap joint was conducted using a three dimensional elastic finite element analysis. The stress intensity factors for this geometry with symmetric and single cracks were determined. It was concluded that K_I is dominant, and K_{II} and K_{III} have values, which are one order of magnitude smaller, with the only exception of K_{III} in the vicinity of the faying surface. Using the stress intensity factor calibration, and a simple Paris law back extrapolation, the EIFS was calculated. Using the EIFS, fatigue lives were predicted and a good agreement with experimental data was obtained.

7. Acknowledgements

The present work is part of the contribution of IDMEC-Porto to the ADMIRE project (project GRD1-2000-25069; contract G4RD-CT-2000-0396) of the Commission of the European Community.

REFERENCES

[1] *J. Schijve*, Fatigue of structures and materials, Kluwer Academic, Dordrecht, Boston, MA, 2001.

[2] Karlson Hibbitt and Sorensen Inc. - Abaqus, Fracture Mechanics (course notes),1991.

[3] *D. Broek*, Elementary engineering Fracture Mechanics, Martinus Nijhoff; distributed by Kluwer, Dordrecht, The Netherlands; Boston Hingham, MA, 4th rev. edition, 1986.

[4] *J. D. Eshelby*, The continuum theory of lattice defects, in Solid State Physics, volume 3, pp. 79–141, New York, 1956.

[5] *J.R. Rice*, A path independent integral and the approximate analysis of strain concentrations by notches and cracks, Journal of Applied Mechanics, Vol. 35, pp. 379-386, 1968.

[6] *R. W. Hertzberg*, Deformation and Fracture Mechanics of engineering materials, J. Wiley and Sons, New York, 4th edition, 1996.

[7] *P. C. Paris, P Gomez, W. E. Anderson*, A rational analytic theory of fatigue, The Trend in Engineering, vol. 13 nr.1, pp. 9-14, 1961.

[8] *D. A. Skinn, J. P. Gallagher, A. P Berens, P. D., Huber J. Smith*, Damage tolerant design handbook, vol. 3, University of Dayton Research Institute, 1994.

[9] *P. M. G. P. Moreira*, A contribution to the study of fatigue of riveted lap joints, MSc Thesis, Faculdade de Engenharia, Universidade do Porto, Portugal, 2004.

[10]. *M.F.J. Koolloos, P.M.S.T., de Castro, R. Esposito, G. Cavallini*, Fatigue testing of single-rivet lap joint specimens, Technical Report ADMIRE-TR-3.1-06-3.1/NLR-CR-2003-281, 2003.