STRESS INTENSITY FACTOR CALIBRATION FOR A LONGITUDINAL CRACK IN A FUSELAGE BARREL

S.M.O. Tavares[1] , P.M.S.T. de Castro[1]

[1] Faculdade de Engenharia da Universidade do Porto and IDMEC-Porto
Rua Dr. Roberto Frias, 4200-465 Porto, Portugal
E-mail: {sergio.tavares;ptcastro}@fe.up.pt

ABSTRACT

Aircraft structures require minimum weight configurations with high strength in order to support all operation stresses with high reliability. Framework construction is the base of these airframes where cross sectional shapes are bolted, welded, bonded, pinned, riveted or machined into a rigid assembly. The vertical and horizontal cross-members are arranged to withstand all structural loads and the skin to support the pressure gradient. This type of fuselage has been in use for about 80 years; it is very strong and of relatively light weight when used with high specific strength materials. Due to the impossibility of producing defect free structures and to avoid damages during all life cycle, these structures require to be damage tolerant in order to be trustworthy. Damage tolerance is a concept predominantly applied in the primary structural parts of civil airframes in order to tolerate a defect that can be detected and repaired in the next maintenance check.

The two most frequently types of structural damages in a fuselage are the longitudinal cracks due the pressurization cycles and the circumferential cracks due the bending and torsion of the fuselage.

In this article, the stress intensity factor, quantifying the intensity of the stress field around a crack tip for a longitudinal crack under the pressurization load, is studied.

For this purpose, a barrel composed by two frames was chosen, with the longitudinal stiffeners and with the geometry usually found in civil airframes. A central crack, between the two frames, was simulated in a finite element model composed by solid elements. The stress intensity factor for different crack lengths, until the crack tips reach the frame were calculated using linear elastic fracture mechanics assumptions and the modified virtual crack closure technique. In addition, the stress intensity factors along the skin thickness were determined. The variation of the SIFs values along the thickness is non symmetric due the bulging effect, which it is illustrated in this article.

KEY WORDS: Airframes, damage tolerant, linear fracture mechanics, modified VCCT, stress intensity factor.

1. INTRODUCTION

The fuselage is the main structure in the aircraft that holds crew, passengers and cargo. An aircraft fuselage structure must be capable of withstanding many types of loads and stresses, and at the same time with low weight.

Truss, monocoque, and the semi-monocoque solutions are found for the design of this structure. Truss or framework types of construction have wood, steel or aluminum tube, or other cross sectional shapes which may be bolted, welded, bonded, pinned, riveted or machined into a rigid assembly[1]. The vertical and diagonal cross-members are arranged to withstand both tension and compression loads. This type of fuselage has been in use for about 80 years. It is very strong and of relatively light weight. The truss assembly is usually covered with a fabric skin. The fabric skin is usually doped and painted which makes it taught and airtight, and adds to its strength. Although cloth fabric is not considered a primary structural member, some aircraft are covered with a glass cloth or mat consisting of impregnated glass fiber reinforced with epoxy or other resins, which is sometimes part of the primary structure. Both the monocoque and semi-monocoque fuselage structures use their skin as an integral structural or load carrying member. Monocoque (single shell) structure is a thin walled tube or shell which may have rings, bulk-heads or formers installed within. It can carry loads effectively, particularly when the tubes are of small diameter. The stresses in the monocoque fuselage are transmitted primarily by the strength of the skin. As its diameter increases to form the internal cavity necessary for a fuselage, the weight-to- strength ratio becomes more efficient, and longitudinal stiffeners or stringers are added to it. This progression leads to a semi-monocoque fuselage, which depends primarily on bulkheads, frames and formers for vertical strength, and

[1] This structural design was also used in automotive engineering, as in the iconic Maserati Birdcage of the sixties.

longerons and stringers for longitudinal strength. Semi-monocoque is the most popular type of structure used in aircraft design today, [1]. It is composed of a long tube shape with different reinforcements in order to sustain and reinforce the structure.

The principal source of the stresses in this structure is the internal pressure in high altitude caused by difference of cabin pressurization and reduction of the outside pressure with increase in altitude, but the structure is subjected to other loads, as bending, torsion, thermal loads, etc.. In this article, the effect of internal pressure when the fuselage presents a crack was analyzed. The traditional aircraft fuselage is composed of the skin consisting of a cylindrical shell typically 1-3 mm thick, circular frames and axial longerons (or stringers), and normally these components are manufactured with an aluminum alloy and are connected by rivets. As an example, a fuselage configuration of the Fokker 100 is presented in Figure 1. Figure 2 shows the cross-sectional properties of the substructure reported in [3], representative of a generic frame and longerons design. It was intended to carry out a SIF calculation of a representative section of a cracked fuselage. Since the intention was to test a methodology and numerical procedure, and not to calculate solutions for a given aircraft model, an equivalent geometry based in the data in the report [3] was used as modeled a representative case.

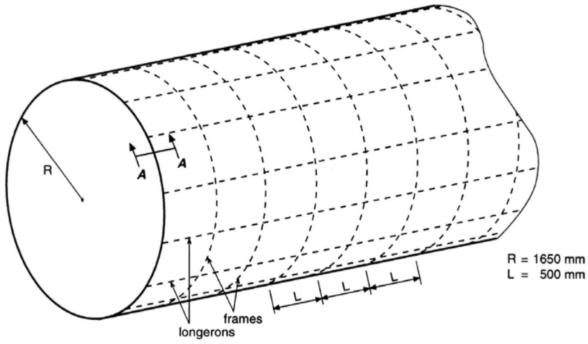


Figure 1. Aircraft fuselage configuration of Fokker 100, [2].

Detailed dimension of the frame and longeron used in the finite element mesh are shown in Figure 3.

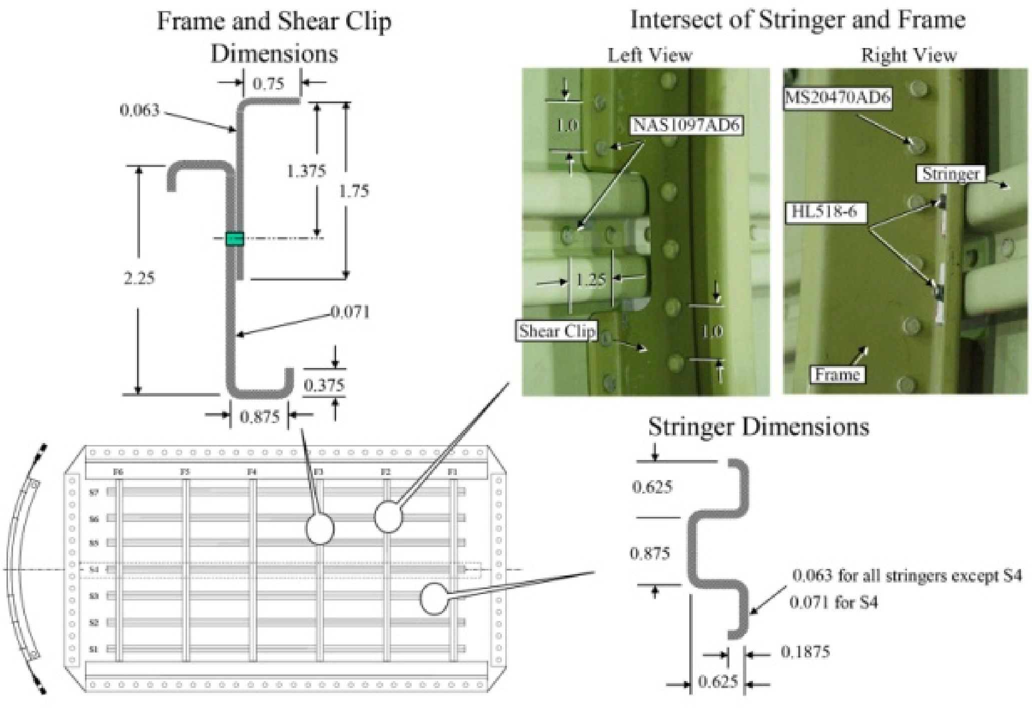


Figure 2. Dimensions of frame, stringer and intersect of stringer and frame, [3].

A part of this fuselage with two frames with 500 mm spacing and 28 longerons, representing half of this fuselage, will be used for SIF calculations. Figure 4 shows a schematic representation of the modeled geometry. The SIFs are calculated for the situation consisting of a single crack in the center of the two frames and parallel to the longerons. An infinite number of crack locations and configurations could be modeled; however the chosen case for the present study represents a typical one and is one of the worst cases. Only half of the fuselage (180°) was modeled. This option makes it possible to use this model for future calculations considering other loading scenarios and simulates the real case with better precision than a smaller part as in Figure 2.

Bending and torsional moments, vertical shear and pressure loads are examples of loads on the fuselage during its service. Internally, the passenger floor, the cargo floor and structure weight promote bending, torsional and vertical shear stresses in the structure. In addition, the wings create significant stresses in fuselage, [4]. The internal pressure is carried primarily by the skin, rather than by the internal framework, hence it is considered the principal load and cause of loss of fuselage integrity, in the presence of some crack or other damage, [5]. The SIF calculation presented in the following section takes into account the effect of the internal pressure only. Other effects are not easily quantified due the dependency to the location in the overall fuselage.

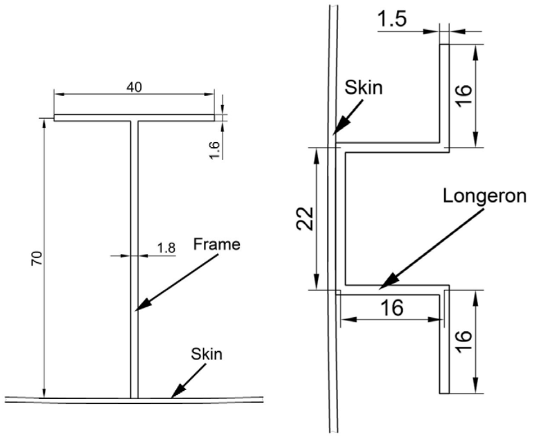


Figure 3. Longeron and frame geometries and dimensions used in the finite element model.

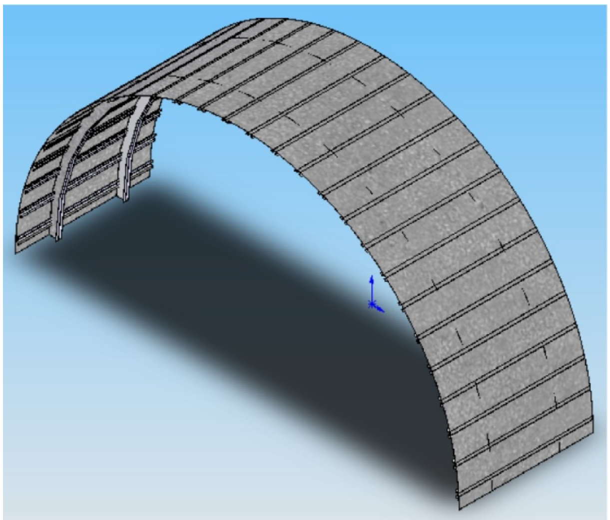


Figure 4. Fuselage frame, representation of modeled geometry.

2. FINITE ELEMENT MODEL

With the geometry presented in Figure 4, a mesh was constructed in order to perform stress field calculation using finite element models. To model this fuselage solid instead of shell elements were used. This option aims at more flexible model, usable for thermo-mechanical analysis. Two elements along the thickness were used in the skin; with the possibility evaluate stress along the skin thickness. In the other components only one element along thickness was used.
The mesh was performed with FEMAP software because it provides powerful tools for meshing geometry, as well as for applying loads and boundary conditions, [6]. Firstly 1/28th of the complete model was modeled using this software, Figure 5. Afterwards, the complete mesh was produced with a radial copy of this part. The following step was the application of the boundary conditions. As symmetry exists in the plane xz or y = 0, the nodes in this plane were restricted in the y direction. In addition, the model was restricted in the z direction in the plane z = 0 and in the x direction in the x =0 line. These are secondary restrictions, however they allow the elimination of large displacements in the x and z directions and reduce the model size. Figure 6 shows schematically the complete mesh and the boundary conditions (in orange).

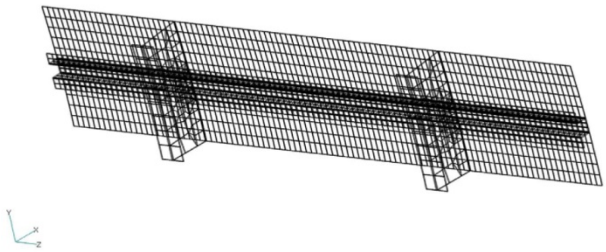


Figure 5. Mesh of 1/28th of the FE model.

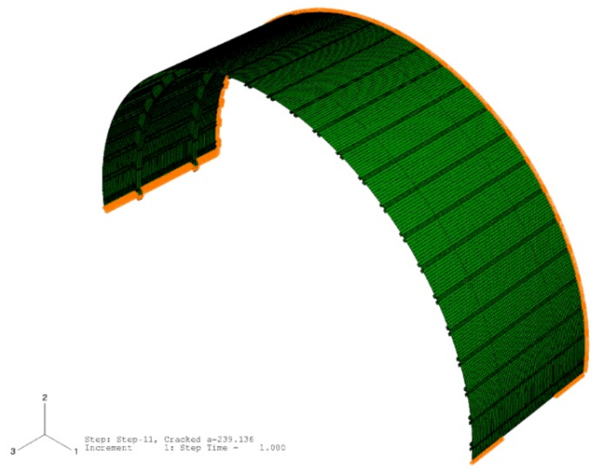


Figure 6. Complete mesh with boundary conditions.

The internal pressure is applied in all the internal faces. For this study the value 0.1 kPa is considered. This value is achieved at altitudes above 12000 m. The most common values are 0.05-0.06 kPa, but this value depends on the gradient between the internal pressure and external pressure.
In recent commercial airplanes the internal pressure is increased for a better comfort of the passengers during the flight.
Figure 7 shows the mesh with the internal pressure applied in the internal element faces of the fuselage. A central crack between the two frames was modeled.
A total of 10 different crack lengths were modeled for evaluation of stress intensity factors and calibration of this structure. The global model was composed by 11 steps and was processed in the finite element package ABAQUS, [7], with parabolic solid element C3D20 that is a 20-node quadratic brick with 27 integration points. Solid elements with 8 nodes reduce substantially the

model size however they cannot simulate the curvature of the skin and large differences in stress fields were found. The properties of this model are summarized in Table 1. The model was computed in dual Core workstation with Intel Xeon 3060 Conroe 2.4GHz processor and 4Gb of RAM memory. The model was completed after 63759 seconds (about 18 hours).

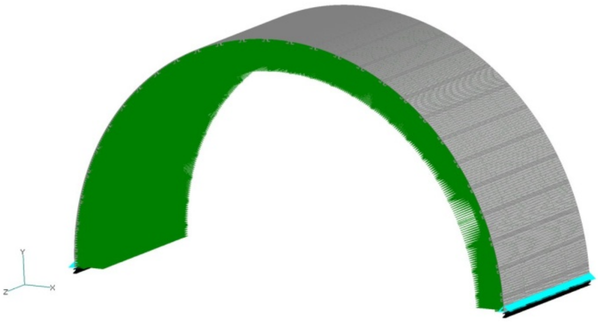


Figure 7. Complete mesh with internal pressure and symmetrical boundary conditions.

Table 1. FE model variables.

Number of Elements	136528
Number of Nodes	796251
Number of equations	2388753
Floating point operations per interactions	4.21E+12

3. RESULTS

Figure 8 shows a result of the von Mises stress field and the amplified deformed shape promoted by the pressurization of the cabin, for the complete frame without cracks. The full representation is obtained by the symmetry conditions of the plane xz or in ABAQUS coordinate system, *1-3*.

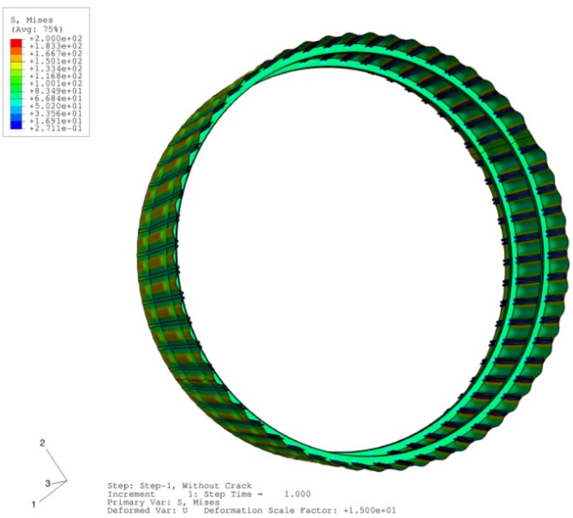


Figure 8. Von Mises stress field and deformed shape of the complete frame, displacement scale factor 15×.

Figure 9 presents a detail of the deformed shape with a scale factor of 30×. The shape of the skin between the frames and longerons is typically entitled as pillow effect.

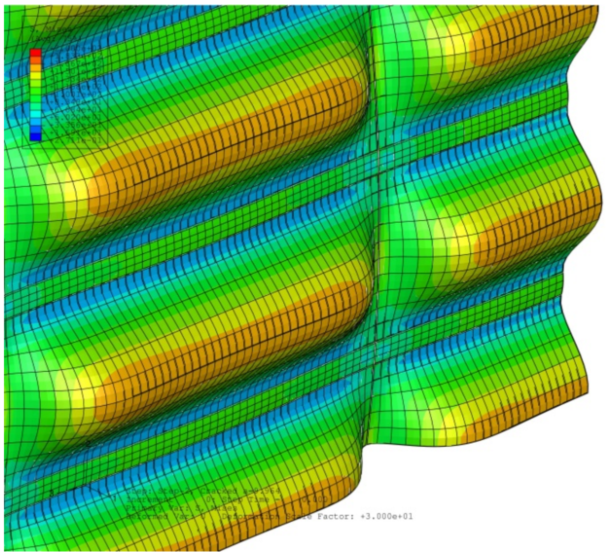


Figure 9. Von Mises stress field and deformed shape detail, displacement scale factor 30×.

3.1. Stress intensity factors

Stress intensity factors for different crack lengths were determined through the virtual crack closure technique (VCCT) in a modified version.

The virtual crack closure technique is based in the energy release rate. This energy can be calculated by variation strain energy release rate when an extension of crack length (Δa) is imposed:

$$G = \frac{\partial U}{\partial a} \approx \frac{U_{a+\Delta a}}{\Delta a} \tag{1}$$

A modified version of the VCCT proposed by Krueger in 2002, [8], presupposes that if the nodal displacements are measured, near the crack before and after grow the crack length to $a+\Delta a$ (an infinitesimal increment), for nodes equidistant to the crack tip, the nodal displacements are identical. This assumption allows computing the energy release rate (G) using only one finite element analysis for each crack length. For 3D parabolic finite elements, the determination of the energy release rate with the modified virtual crack closure technique can be determined using the nodal loads and nodal displacements; however it requires considering the different weights of the nodes in the middle and in the corner of the element. As example, for a parabolic element with 20 elements, the mode I, considering the notation presented in Figure 10, the equation used to determine the energy release rate for the node at the crack surface (node 3) is:

$$G_I = -\frac{1}{2\Delta a \cdot \Delta b}\Big[F_{z_3}\left(u_{z_{1*}} - u_{z_1}\right) + F_{z_4}\left(u_{z_{2*}} - u_{z_2}\right) + \frac{1}{2}F_{z_6}\left(u_{z_{5*}} - u_{z_5}\right)\Big] \tag{2}$$

where F_z is the nodal force in the z direction, u_z is the displacement in z direction and Δa and Δb are the element dimensions. For the nodes positioned in the middle of the element, in this case node 6 and 12, and the corner nodes 9 and 15 similar equations can be used to determine the energy release rate at the respective positions.

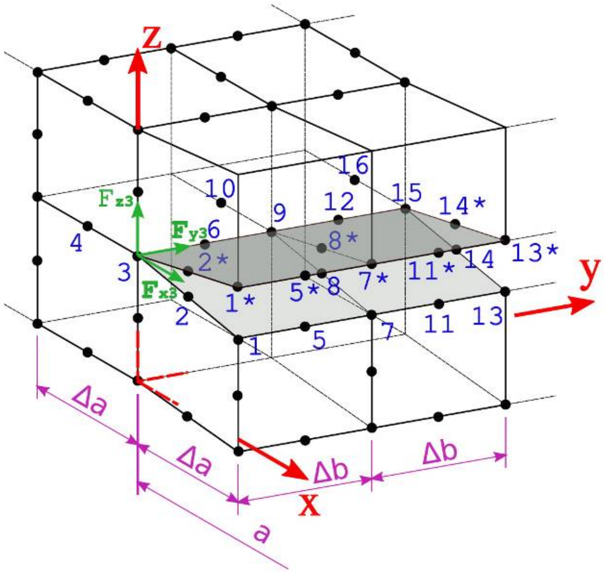
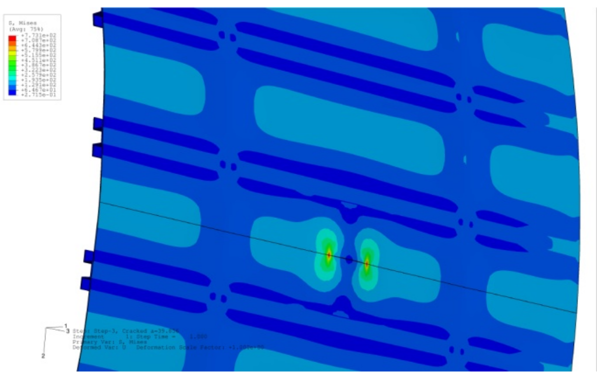
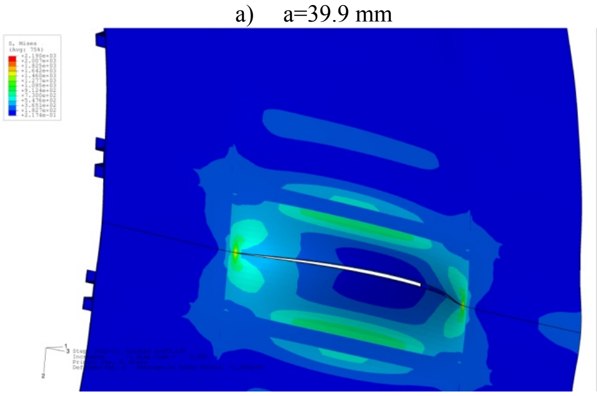


Figure 10. Modified VCCT, annotation for a generic finite element.

The stress field around the crack is presented in Figure 11 for two distinct crack lengths. For the smaller crack length the stress field adopts the usual shape of a crack in an infinite plate. For the crack with length of 239 mm, Figure 11 b), the stress field is affected by the frame reducing the stress intensity and slowing down the crack propagation.



a) a=39.9 mm



b) a= 239 mm

Figure 11. Stress field around the crack for two crack lenghts.

The skin thickness of the barrel was simulated using two elements with three nodes per face; therefore SIFs for 5 points along the skin thickness were determined.
The values estimated for the different thickness are compiled in Figure 12, for the case of an internal pressure of 100kPa.

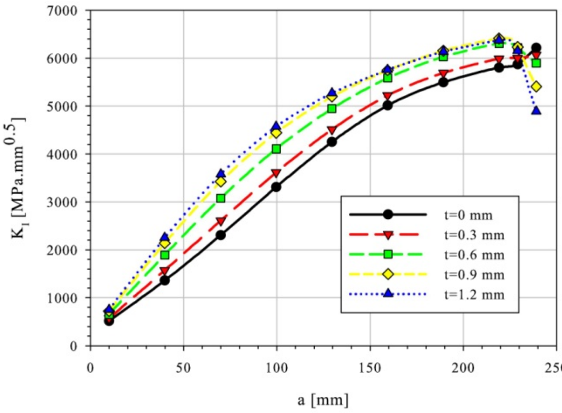


Figure 12. SIFs values for a centred crack fuselage frame.

The SIFs can be made non-dimensional using the equivalent tangential stress or hoop stress (σ_t) generated by the applied pressure and not taking into accout the reinforcement elements as the frames and longerons. This equivalent σ_t stress can be related with internal pressure in a thin-walled pressure vessel:

$$\sigma_t = \frac{pR}{t} \tag{3}$$

where R is the radius of the cylinder (1650 mm) and t is the shell thickness (1.2 mm). Applying these values the tangential stresses are 137.5 MPa.
Figure 13 presents the non dimensional stress intensity factors, with the $\sigma_t\sqrt{\pi a}$ value, in a three dimensional mode.

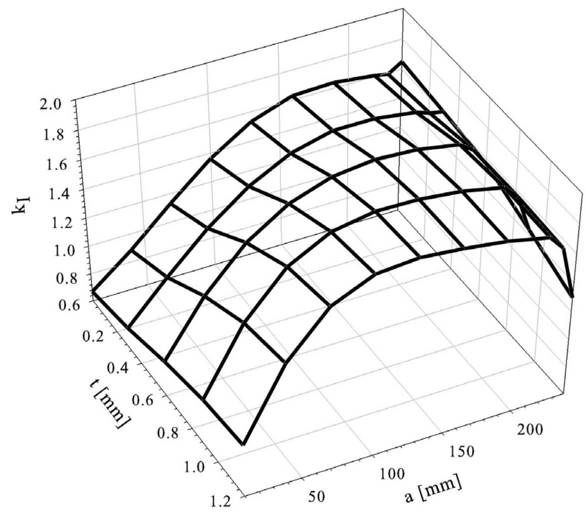


Figure 13. Non dimensional SIFs values for a centred crack fuselage frame.

Bulging is an important problem in the structure subject to internal pressure. This effect take out the symmetry of the stress intensity factors along the thickness as is noticeable in Figures 12 and 13.
Several cross sections for three different crack lengths are presented in Figure 14, where the deformation of the skin is shown at the center of the crack in order to visualize this result and for a better understand of the effects in a crack.

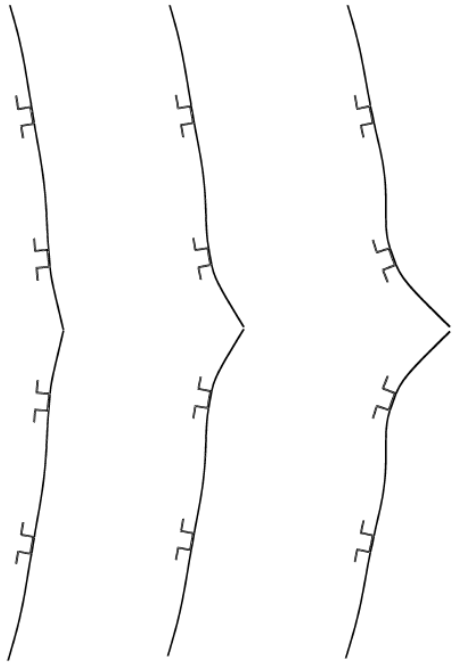


Figure 14. Bulging effect for three different crack lengths (a=59.855 mm; a=129.532 mm; a=239.136 m respectively).

4. CONCLUSIONS

Lightweight structures for air transportation exploit damage tolerant design philosophies in order to increase the service life of the structural parts with high reliability.
In this article, a 3D finite element mesh corresponding a barrel composed by two frames of one representative fuselage was done.
The internal pressure is one of the main loads that the fuselage needs to hold. The compression and decompression cycles are usually used as reference for the fatigue life of the fuselage.
A central crack growth in this finite element model was modeled and the stress intensity factors were determined using the modified virtual crack closure technique. The application modified VCCT and the 3D model, allowed to determine the variation of SIFs along the skin thickness and the influence of the bulging effect in the SIFs.
The consequence of the bulging effect may be noteworthy for a crack in the middle and parallel of two longerons. The effect of the frames in the obstruction of the crack growth was visible in the last crack lengths.
This mesh allows the application of multiple loads and crack orientations in order to study the stiffness of the structure in the presence of different damages.

ACKNOWLEDGEMENTS

DaToN Project, EU FP6 contract AST3-CT-2004-516053 and FCT fellowships SFRH /BD/ 35143/2007 are acknowledged.

REFERENCES

[1] P. Katz, *Stresses and strains on aircraft* www.navair.navy.mil/safety/documents/Stress_Strain.doc , 1997.

[2] H.J. ten Hoeve, L. Schra, A.L.P.J. Michielsen, and H. Vlieger, "Residual strength test on stiffened panels with multiple-site damage", Technical report, NLR CR 96792 L, DOT/FAA/AR-98/53, National Aerospace Laboratory NLR, The Netherlands, February 1999.

[3] John Bakuckas, "Full-scale testing and analysis of fuselage structure containing multiple cracks" Technical report, DOT/FAA/AR-01/46, Federal Aviation Administration (FAA), William J. Hughes Technical Center, Airport and Aircraft Safety, Research and Development Division, Atlantic City International Airport, 2002.

[4] L. Boni and D. Fanteria, "Finite-element-based assessment of analytical methods for the design of fuselage frames", *Proceedings of the Institution of Mechanical Engineers Part G-Journal of Aerospace Engineering*, 220(G5), pp. 387-398, 2006.

[5] P. E. O'Donoghue and J. Ju, "Experimental/numerical techniques for aircraft fuselage structures containing damage", in *15th European Conference of Fracture, Advanced Fracture Mechanics for Life and Safety Assessment*, Stockholm, Sweden, 2004. European Structural Integrity Society (ESIS).

[6] FEMAP - Finite element modeling and postprocessing - Version 9.2 - Users manual. UGS Corp., 2006.

[7] ABAQUS user's manual, Version 6.6. 2006.

[8] R. Krueger, "The virtual crack closure technique: History, approach and applications", NASA Technical Report, NASA/CR-2002-211628, ICASE Report No. 2002-10, 2002.