EXPERIMENTAL/NUMERICAL TECHNIQUES FOR AIRCRAFT FUSELAGE STRUCTURES CONTAINING DAMAGE

Padraic E. O'Donoghue, Jinsan Ju
Dept. Civil Engineering
National University of Ireland
Galway, Ireland
padraic.odonoghue@nuigalway.ie, jinsan.ju@nuigalway.ie

Abstract

Fracture mechanics has always been a major problem for the aircraft industry as designers must ensure that catastrophic events such as fuselage failure will never occur. An on going EU project, TANGO, is focused on developing and designing the next generation aircraft. Clearly damage tolerance assessments play a very important role in the design of these new structures. Thus, the work programme includes an extensive series of testing and analysis of aircraft fuselage panels containing pre-existing cracks. While briefly mentioning some of the curved panel testing work, this paper primarily describes the advanced computational analysis techniques that are used to determine stress intensity factors for the damaged panels. A hierarchical analysis strategy has been developed that uses an automated global-local approach. Results are presented for a number of the test panel configurations.

Introduction

An ongoing EU project, TANGO, is focused on developing and designing the next generation of aircraft structures. The goals of this project are to introduce significant cost and weight savings over existing designs. This requires new, high potential materials, design methods, manufacturing techniques and assembly processes to be employed on the key primary airframe elements. The geometry of an aircraft fuselage panel is extremely complex with various arrangements of stiffeners, frames, shear clips etc and the added problems associated with the analysis of the rivets. It is also necessary to allow for some of these components to be broken.

An extensive series of full scale curved panel tests is being conducted as part of the TANGO project. The purpose of these tests is to examine different configurations of frames, tear straps, materials etc. In addition a number of different materials have been considered including aluminium and the composite material, GLARE. A wide variety of fracture scenarios are included in the panels to investigate different cracked panel configurations. These include circumferential and axial cracks in various locations on the fuselage. The findings of these curved panel tests will then feed into a pressurised barrel test in which different panel configurations will be examined.

The curved panel testing is complemented by finite element assessments of the structures. This is highly significant analysis as it allows an initial investigation of a curved panel configuration prior to the test being carried out. This paper focuses on the analysis aspect of the project. The computational methodology is briefly described along with validation from some previous work. Fracture simulations of the curved panel tests are then carried out and assessments of the results are provided.

Technical Approach

The geometry of an aircraft fuselage panel is extremely complex with various arrangements of stiffeners, frames, shear clips etc and the added problems associated with the analysis of the rivets. It is also necessary to allow for some of these components to be broken. The final level of complexity is provided by the cracks that have been placed in the skin. Thus, the direct analysis of such a structure is almost impossible, particularly when it is necessary to obtain a relatively detailed stress distribution in the vicinity of the crack tip. The approach that is used here is an extension of the three-step hierarchical approach that was developed by Kawai et. al. [1]. In the current approach , a two stage global-local technique is used [2].

The global analysis model considers the entire curved panel and is simply concerned with the overall flow of stress through the structure. The crack is present but no attempt is made to obtain detailed stress evaluations in the crack tip vicinity. However, simplifications are used with respect to the frames and stringers and these are modelled using beam elements. The local model then focuses on the bay in the vicinity of the crack of interest. Here, the frames and stringers are modelled more precisely using shell elements as is the fuselage skin, while beam elements are used for the rivets. The crack tip region is now carefully modelled to allow accurate calculation of the stress intensity factors. The key point about this approach is that the local model here consists of curved shell elements while a 2-D local model was used previously [1]. This allows for a more realistic simulation of the bulging effects in the vicinity of the crack.

One of the important requirements in the development of the analysis methodology was the automation of the entire process. This is necessary as the time associated with mesh generation for problems of this nature can be extremely time consuming. The input was parameterised to allow for variations in stringer/frame size, shape and spacing. Once a number of key parameters are input, then the meshes (both global and local) are automatically generated. The other area where automation is critical is in the transfer of boundary conditions. Essentially the output of the global model is used as the input for the local model. It is also possible to vary the size of the local model, a factor that is very important as different crack lengths are considered. The ABAQUS finite element package [3] is used for the basic analysis while special purpose routines have been developed to parameterise the input and to transfer the boundary conditions. The analysis procedure is now highly efficient and the automation feature also allow for different geometry, material and fracture configurations to be easily considered.

Validation of the approach was provided by comparison with an earlier set of curved panel tests that was carried out by Deutsche Aerospace [4]. A crack was introduced into the panel and this was allowed to grow under a simulated fatigue pressure loading and the crack length versus number of loading cycles was recorded. Based on these results, it was possible to back calculate the stress intensity factor as a function of crack length since the relevant fatigue crack growth law was known. Fig. 1 shows the back calculated stress intensity factor compared with the computational analyses using the hierarchical approach. As can be seen, the agreement between the results is very good. This clearly establishes that the numerical approach is capable of producing the stress intensity factors in a fast and accurate manner.

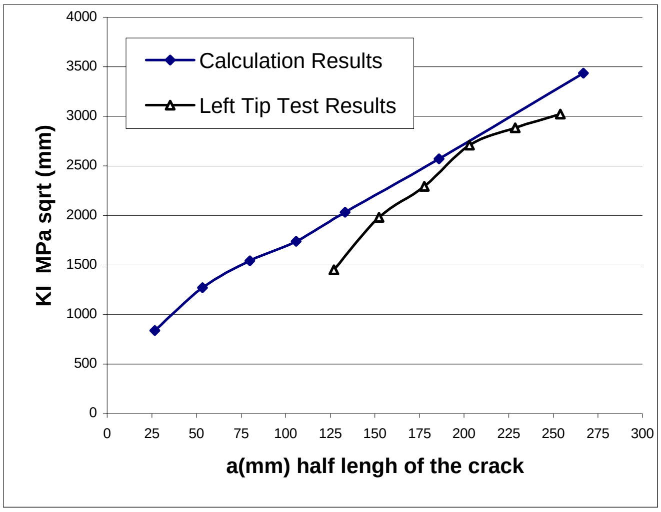


FIGURE 1. Comparison Between Current Analysis and Results Back Calculated from Experiment (Left Crack Tip).

Curved Panel Tests

Three different test panel configurations are considered in the analyses that were conducted as part of the present study – referred to as TXB1, TXB2, TXB3. All three have the same geometrical layout of stringers and frames but have different material configurations. There are also differences in the tear straps that are used to inhibit the crack growth. The key dimensions of the panels are provided in Table 1. The configurations of TXB1-3 are shown in Figs. 2-4. There are 7 frames from C1-C7 and 10 stringers from P1-P10 in each test. There are two longitudinal and two circumferential cracks locations in each model, which locations are shown in figures as well. The internal pressure is 0.0593 MPa and axial and circumferential loads are applied to the edges of the panels to ensure consistency with a pressurised cylinder loading.

The skin of TXB1 is 2524 aluminium and, in addition to the (aluminium) stringers and frames, there are also tear straps present. The strap material is GLARE2A-3/2,which consists of three layers of 0.3mm thick 2024 aluminium alloy and two layers of 0.25mm thick pre-preg containing 0 degree fibres – the longitudinal direction.

The skin of TXB2 is GLARE3. This GLARE3 material consists of three 0.3mm thick layers of 2024 aluminium alloy and two 0.4mm thick layers pre-preg 0/90 degree fibres. There is internal splice under frame four located at C4 also.

TABLE 1. Geometrical and Material properties.

	Symbol	TXB1	TXB2	TXB3
Fuselage Radius/mm	R	2820		
Stringer Pitch/mm	b	185.9		
Frame Pitch/mm	l	533.4		
Internal Pressure/MPa	p	0.0593		
Skin:				
Skin Thickness/mm	t_s	1.4	1.7	1.4
Modulus of Elasticity/GPa	E_s	72	72	56.6
Poisson's Ratio	ν	0.33	0.33	0.28
Stringer: 				
Stringer Cross Sectional Area /mm^2	A_{str}	116	116	116
Modulus of Elasticity/GPa	E_{str}	70.9	70.9	70.9
Frame: 				
Frame Cross Sectional Area/mm^2	A_{fr}	269		
Modulus of Elasticity/GPa	E_{fr}	74.5		

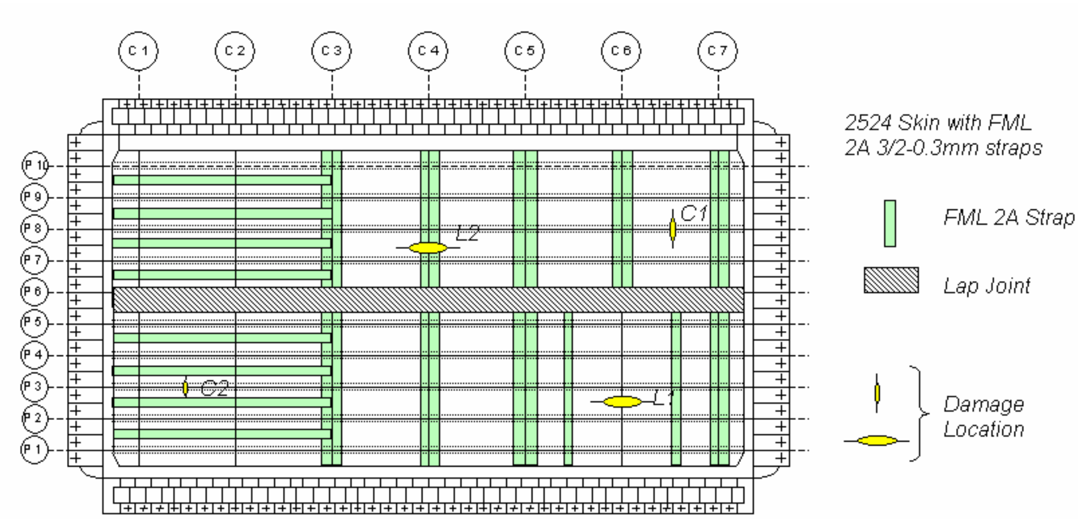


FIGURE 2. TXB1 configuration.

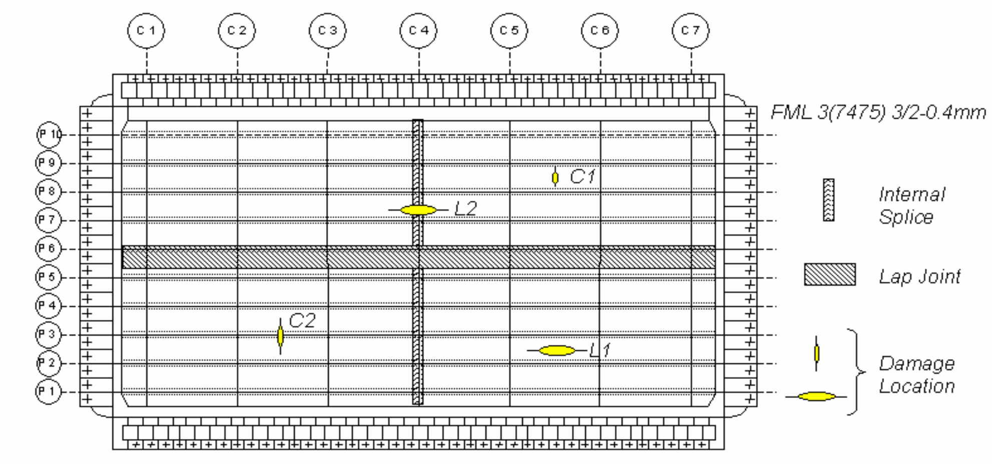


FIGURE 3. TXB2 Configuration

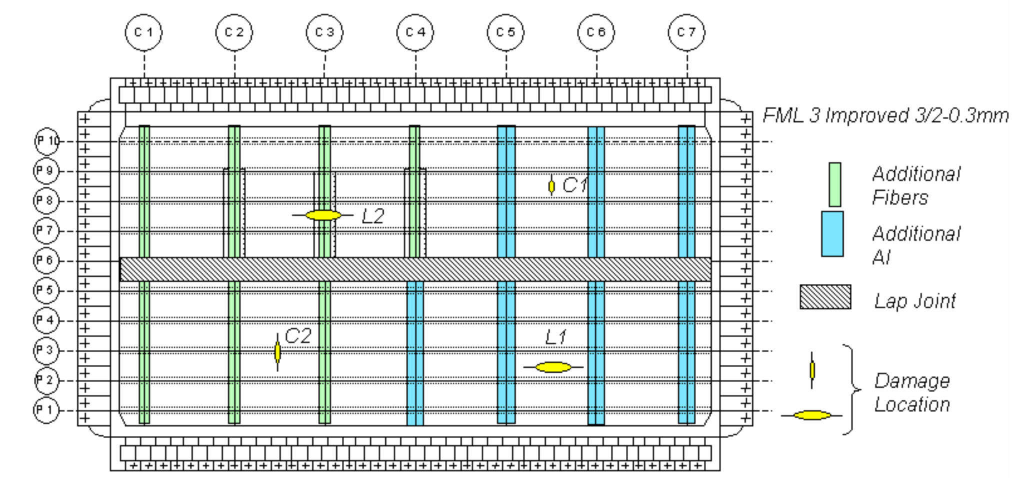


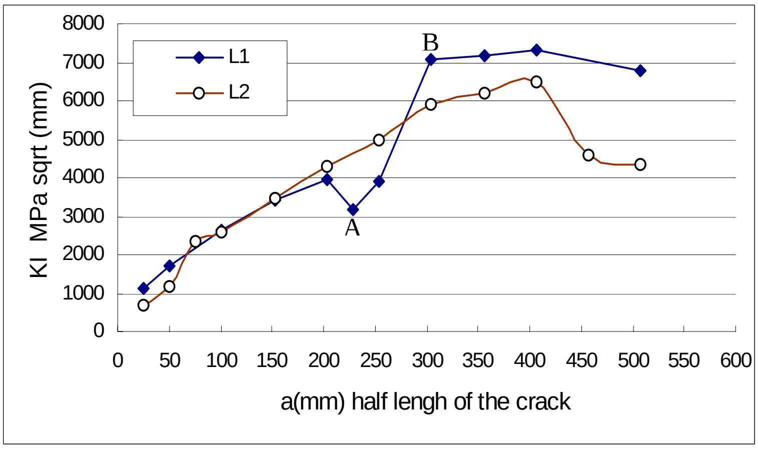
FIGURE 4. TXB3 Configuration

The skin of TXB3 is GLARE3 improved, which consists of three layers of 0.3mm thick 2024 aluminium alloy and two layers 0.25mm thick pre-preg containing three 90 degree fibre layers and one 0 degree fibre layer.

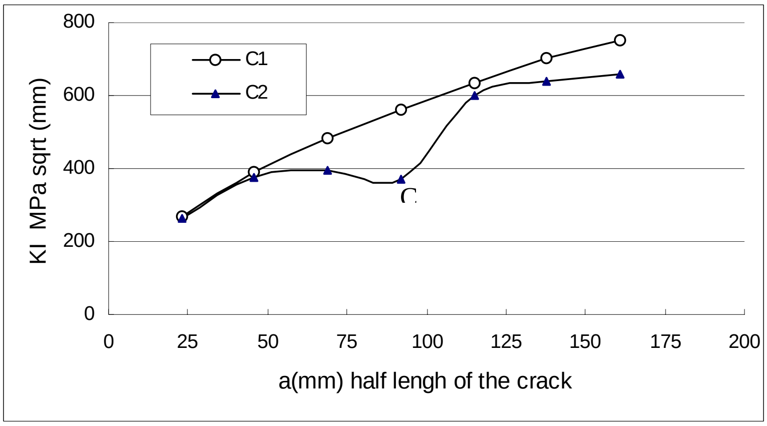
Curved Panel Analyses

TXB1 results:

From the result in Fig.5, it can be seen that the SIF values for the longitudinal cracks, L1 and L2, are relatively similar for the shorter crack lengths. This is not surprising as the cracks are well away from any straps. However, L1 encounters a strap when the half length is about 200 mm and the SIF decreases due to the added stiffness of the strap (point A in Fig. 5a). The SIF increases again after it has passed under the strap – which is now

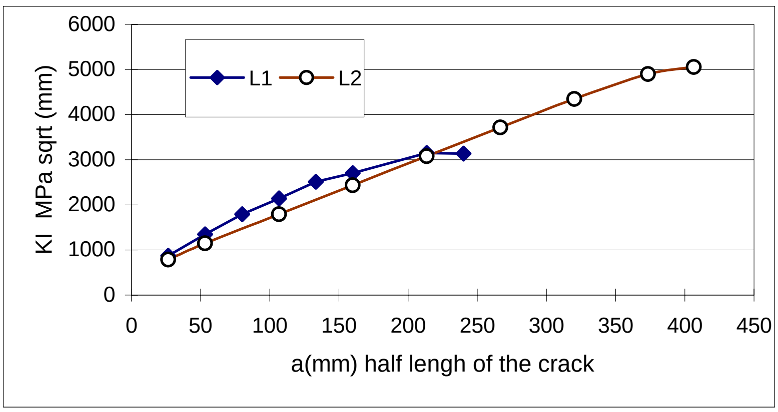


(a)

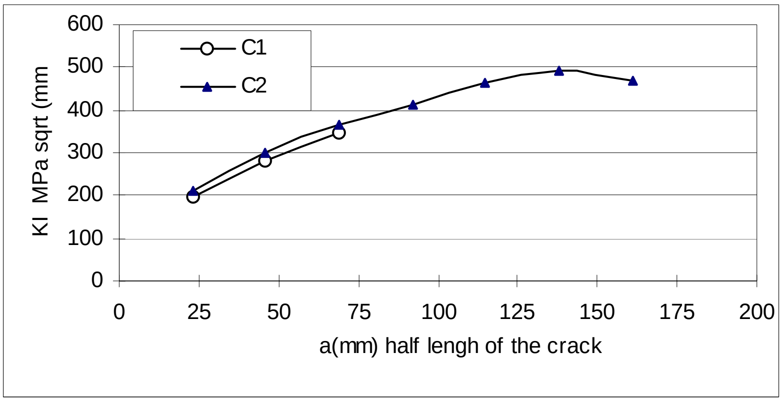


(b)

FIGURE 5. TXB1 results.

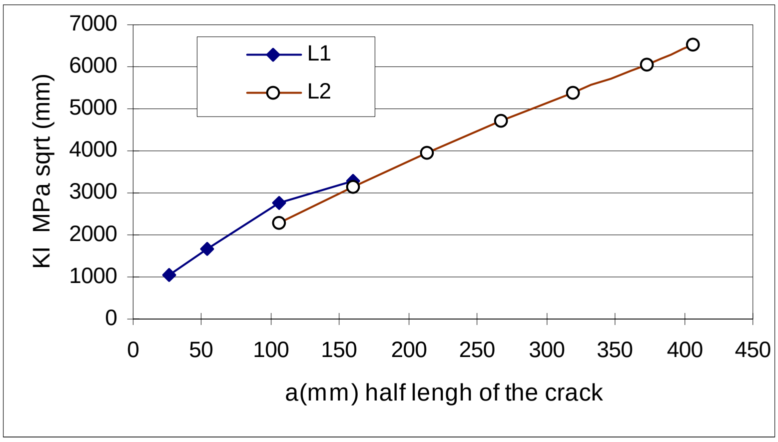


(a)

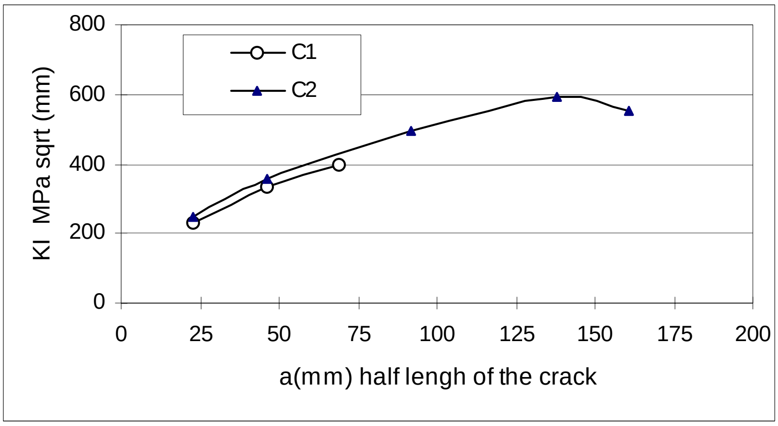


(b)

FIGURE 6. TXB2 results.



(a)



(b)

FIGURE 7. TXB3 results.

assumed to be broken. For larger crack lengths, the SIF for L1 decreases as it approaches frames C4 and C6 while the L2 SIF decreases as it approaches C3 and C5. Using the relevant crack growth law, it is then possible to predict the number of cycles for the various stages of growth. In general, it means the straps have a significant effect on the SIF and can slow the propagation of the crack when the crack tip is in strap area. On the other hand, when the crack grows beyond the area of the straps, the growth will be relatively rapid again because of the sudden increase in SIF. For similar reasons, the SIF for C2 decreases in Fig. 5b when it approaches a strap after about 90 mm (point C). Note also the significantly lower SIF values for the circumferential cracks.

TXB2 results:

From the result in Fig.6a, it can be seen that SIF for L1 is relatively similar for that of L2. However, when L1 reaches a frame after about 250 mm, the values begin to drop as expected. The SIF for C1 and C2 in Fig. 6b are relatively similar until the stringer is reached by C1. In these analyses, the GLARE skin was modeled as a continuum with the different layers being accounted for through an equivalent stiffness. However, from a fracture perspective, the layers would have to be taken into account to calculate an equivalent stress intensity factor.

TXB3 results:

The trends exhibited by TXB3 in Fig. 7 are very similar to those for TXB2 above.

Conclusions

An automated finite element analysis approach has been used here to calculate stress intensity factors for a number of curved test panel configurations. This analysis approach was validated using an earlier test series comparison. Using the methodology, it is possible to consider a wide variety of issues relating to the test panel configurations. Some of these have been illustrated here including (i) crack location, (ii) crack orientation, (iii) skin material, (iv) tear straps as crack growth inhibitors, etc. The next phase is to make predictions on the crack growth using the appropriate crack growth laws.

Acknowledgement: Funding for this work has been provided by the EC through the funding of the TANGO project. The authors appreciate the support of H. J. Schmidt, B. Schmidt Brandecker, G. Watson and N. Medwell, of DA and M. Foster of Airbus.

References

1. Kawai, H., Wang, L., Pipkins, D. S., O'Sullivan, K, O'Donoghue, P. E., Atluri, S. N., "Automation of Global-Intermediate-Local Analysis Based on Feature Based Modelling and the Finite Element Alternating Method", *Computer Modelling and Simulation in Engineering*, Vol. 4, 1999, pp. 457-464.
2. O'Donoghue, P. E., Ju, J, "Advanced Analysis Techniques for Aircraft Fuselage Structures Containing Damage", in preparation, 2004.
3. Hibbitt, Karlsson & Sorensen. Inc. ABAQUS/Standard User's Manual.1997.
4. Watson, G., personal communication to P. O'Donoghue, 2003.