

# A FINITE ELEMENT APPROACH TO OPTIMIZING DAMAGE TOLERANCE OF AIRFRAME PANELS

**S. Arsene**, Alcan, Alcan - Pechiney Centre de Recherches de Voreppe (CRV), France

**S. Van der Veen**, Alcan Aerospace – Pechiney Rhenalu Issoire, France

**R. Muzzolini**, Alcan Aerospace, Seattle USA (corresponding author: [Raphael.Muzzolini@alcan.com](mailto:Raphael.Muzzolini@alcan.com))

## ABSTRACT

Aluminum alloys have been used for decades in the aerospace industry because of their high strength to density ratio and their relatively low price. One of the critical fields in aircraft design is damage tolerance (DT), which represents the ability of a material to sustain loads when damaged. The FAA requires aircraft manufacturers to show that the residual strength of the stiffened panels that constitute most of current aerospace structures is high enough. Though the final validation is done by testing, cheaper ways to evaluate the damage tolerance of panels are necessary during the numerous iterations of the design phase. Finite Element Modeling (FEM) has proven reliable for Fracture Mechanics calculations and is therefore more and more widely used. However, despite a continuous reduction in computation times, a complete analysis still involves the generation of many models for which meshing needs to be done very carefully. If they are not automated, these operations are considerably time-consuming and lead to overall analysis times that are not compatible with design optimization procedures.

A software package has therefore been developed in Alcan's research center to help speed up the DT evaluation through FEA. This tool called PAnel is based on commercially available software MSC.Marc Mentat and MSC.Marc and acts as a large macro launching meshing, calculations and post-treatment operations. The user only needs to provide a text file containing some parameters describing the desired configuration. He will then obtain as a result from PAnel the Stress Intensity Factor vs crack length curve needed for the DT analysis.

Features include curvature, pressurization and bi-directional stiffening for modeling of both wing and fuselage panels. Bonded, integrally stiffened as well as conventional riveted structures can be modeled, and in the built-up configuration, an elastic-plastic material law can be given to stiffeners. Both force and displacement controlled options are available within PAnel. Validation of the tool has been done through comparison to experiments and aircraft manufacturers' models.

## INTRODUCTION

The aerospace industry is setting more and more aggressive targets for weight and cost of aircrafts along with a new focus on reduced development time. Achieving those targets requires optimized design and processes but also improved material properties. Alcan, one of the largest aluminum suppliers for the aerospace industry, is aimed at developing new alloys that will help engineer lighter airplanes. In order to do so as efficiently as possible, design capabilities have been developed internally that greatly enhance the collaboration with aircraft manufacturers. By knowing the key design features, we can not only understand why the requirements for material properties evolve during the aircraft development cycle but anticipate these

changes and even ultimately propose new concepts. Even if results are ideally directly obtained with the OEMs' design tools, it is important for Alcan to have its own design capability. Innovative ideas can therefore be evaluated independently and benefit the entire aerospace community. However, Alcan does not have access to the huge amount of experimental and in-flight data that support the semi-empirical design tools used by the aircraft manufacturers. It would require too much time and resources to build such models. Besides, our focus is clearly on the development of new advanced aluminum alloys. Our goal is therefore not to design an entire plane but only to gain better understanding of material requirements and to evaluate the weight savings provided by these new advanced aluminum alloys.

The fast iterations on design loops needed by OEMs, which constitute one of the reasons why such models are still widely used, are not required for our less specific studies. It was accordingly decided to build a design tool based on Finite Element Modeling, which was named PAnel (for former Pechiney Aerospace stiffened panel simulator). It is based on the commercially available software MSC.Marc, which has proven to give reliable results in the Linear Elastic Fracture Mechanics field. It acts like a large macro, using information stored in a simple text file to automatically create the required meshes, launch the corresponding computations, extract the results from FE output files and store them in the desired format.

## **DESIGN METHODS**

### **Introduction**

The three main loads an aircraft structure has to carry are its own weight, the lift and the pressurization of the cabin [1]. Schematically, the wing box and the fuselage can therefore be represented as thin-walled beams working in bending. Aircraft structures are consequently roughly divided into areas mainly driven by compression and areas mainly driven by tension. Stiffened panels provide one of the highest resistance-to-weight ratios in both configurations. Fuselage and wing panels are thus usually made of a thin skin on which are assembled stringers in the bending direction and frames or ribs perpendicularly (see Figure 4). Our focus in this paper will be on tension driven panels, since they require more intensive studies that led us to the development of PAnel.

A cyclic tensile load on a panel will eventually lead to fatigue cracks initiating in zones where the stress concentration is high (mostly rivet holes). Under continued loading, those cracks will propagate until one reaches a critical size, for which the panel will not be able to sustain the loading anymore and will brutally fail. The commercial aviation currently uses design approaches that take into account the presence of damage in the structure. The damage tolerant approach relies on slow and predictable growth of damage and designs the structure to the required residual strength. It is used together with the fail-safe approach, which allows for large damage by providing several load paths through the structure. Two main requirements regarding the resistance to damage have been defined and are administered by the Federal Aviation Administration (FAA). In simple terms, it consists in (i) preventing the damage to grow too fast and thus allowing at least two

inspections before a crack can reach a pre-defined critical size; and (ii) ensuring that the structure has enough residual strength to sustain maximum loading with such a critical crack.

The aim of design studies is therefore to determine the minimum weight configuration that meets these requirements [2]. Given the loads the studied structure has to carry and the material employed, iterations are done on the geometrical parameters. For each configuration, the two design criteria – crack propagation time and residual strength - need to be determined. Due to the high number of iterations required, fast semi-empirical models are used by aircraft manufacturers. Testing is performed only as validation on the final configuration to keep the cost as low as possible.

For small comparative studies, FEA is an interesting alternative to compute residual strength and crack propagation. The Linear Elastic Fracture Mechanics (LEFM) methods can be used quite simply with FEM and have proven to give reliable results. Besides, the constant and exponential improvement in processor speed and memory allows for always faster and more accurate FE calculations. However, the preparation of FE models is still extremely time-consuming with meshing operations that can take up to several weeks.

### **Stress Intensity Factors**

The crack propagation requirement is set to allow two inspections before a crack becomes critical. OEMs are closely working with the FAA to define what crack lengths should be considered as initial and critical. Common practices are to use the detectability limit as the initial crack length and a 2-bay-crack as the maximum allowable crack length. Once these 2 sizes are defined, designers compute the number of load cycles that occur between those two crack lengths and compare it to the planned inspection interval time. This is done according to the LEFM by determining the evolution of the Stress Intensity Factors (SIF) with the crack length and integrating for instance the Forman law [3]. The second requirement, the residual strength, is directly obtained by comparing the same SIF vs crack length curve to the R-curve of the material [4]. Both crack propagation and residual strength can therefore be evaluated from the computation of SIF, which can be performed in many different ways with FEM [3,5,6].

The two main methods are the COD (for Crack Opening Displacements) and the Strain Energy Release Rate (SERR). Both are implemented in PAnel. The COD is the simplest because it only requires the computation of the displacements along the crack edges. A Finite Element Model of the studied configuration where actual loads are applied to the desired geometry provides the displacement results and furthermore the SIF value in mode I ( $K_I$ ), for example in plane stress:

$$K_I = \lim_{r \rightarrow 0} \left[ E u_2(r) \sqrt{\frac{\pi}{8r}} \right]$$

where  $E$  is the material Young's modulus,  $u_2$  the C.O.D. (displacements perpendicular to the crack) and  $r$  the distance from the crack tip. In practice no limit is computed, the  $K$  value is taken at a small distance  $r_0$  from the crack tip. That method was proven to give good results. The SERR method uses a different approach, calculating the energy needed by the crack to grow. Two FE models are required in that case for slightly different crack lengths  $l_1$  and  $l_2$ . The total elastic energy of both configurations is computed, and the difference  $\Delta W$  is the energy that was required for the crack to grow from  $l_1$  to  $l_2$ . Since SIF and energy are two different ways of determining the onset of failure, they are linked by a formula, given below in the plane stress case for a crack of length  $2a$ :

$$K = \sqrt{\frac{E \cdot \Delta W}{t \cdot 2 \Delta a}}$$

This method provides a mean value for  $K$  between the 2 crack lengths, and takes into account all 3 modes of crack opening. PAnel computes the value for a user-defined crack length  $l_0$  by generating 2 models for which total elastic energies are extracted. The main model contains a crack of length  $l_0$  and the second a slightly extended crack whose length is  $l_0 + \Delta l$ . Such a method is obviously more intensive than the COD method, since it requires 2 different FE analysis to compute  $K$  for a given crack length. However, using two methods with different theoretical bases procures a higher level of confidence in the results.

For a given configuration, the determination of the two DT requirements therefore goes through the calculation of SIFs for several crack lengths between the initial and critical lengths. For most of the applications, the variations of the SIF with crack length are such that the computation of 10 values is a very minimum requirement to achieve a good level of precision (some

OEMS are asking for 30 values). That implies that at least 20 Finite Element Models need to be built and more than 20 calculations launched and post-treated. The interest of automation is obvious.

## DESCRIPTION OF PANEL

PAnel consists in 2 different executables written in Fortran, PAnel\_pre and PAnel\_post. The first is dealing with the preparation of FE models and the computation whereas PAnel\_post is compiling the results and putting them in a standardized format. A complete DT study on a given panel requires only a small number of parameters, mostly describing the geometry (skin thickness, stiffeners shape) and the loading conditions. All these parameters are stored in a text file, which is read at the very beginning of the procedure. Alcan's research center Pechiney CRV has developed powerful Fortran-written tools in the past 10 years to handle text files. Any text files can be thoroughly searched and information extracted without the necessity for a rigid format. Besides, these tools allow for the generation of text files from templates, which is a convenient way to automate the writing of scripts that are further used in multiple manners: Unix executables, MSC.Marc Mentat or MSC.Marc procedures, input file for other software...

### Procedure

The complete procedure used in PAnel is schematically described in Figure 2. Once data is analyzed from the input text file, a loop begins in which the mesh is generated and calculations launched for each crack length specified. The loop contains several operations. First, a MSC.Marc Mentat procedure file is generated. MSC.Marc Mentat indeed allows the user to sequentially execute a list of operations written in its own operating language [7]. Any existing operation can be performed, such as creation of points, nodes, elements but even complex meshing operations as copy and paste elements, rotations, translations or scaling of solids; or administrative operations such as opening and closing files. PAnel generates a file containing the complete sequence of meshing operations. The input file is analyzed to see what operations need to be performed and to extract numerical values needed for the meshing. A specific UNIX command is then executed which launches the execution of the procedure within a newly opened MSC.Mentat window. This window automatically closes just after having created two input files for MSC.Marc.

PANel\_pre then takes back the hand and launches both calculations with MSC.Marc. As described in the above section, the use of SERR method to compute SIF requires 2 models with slightly different crack lengths. In practice, a first model is built with the current crack length and the corresponding input file is saved. Then, only one or 2 nodes are released at both right and left crack tips slightly extending the crack and thus creating the required second model and input file.

Once all the calculations have been completed, the user simply executes PANel\_post to post-treat MSC.Marc output files and compute SIF values for the specified crack lengths. The way PANel\_post works is very similar to PANel\_pre. A MSC.Marc Mentat procedure is generated whose action is to open all the output files generated by MSC.Marc and perform the following post-treatment operations. Node paths are defined along the crack edges and the displacement vs arc length (distance to the first node in the path) curves are plotted and saved in a table. The value of total elastic energy of the whole model is also stored in another file gathering these values for all crack lengths. Once MSC.Marc Mentat has executed the procedure and closed, the newly generated files are analyzed by a Fortran program that computes the SIF values obtained from COD and SERR methods. Finally, the user can launch a VB Macro within MS Excel, which sums the results in a table and plot curves (see Figure 3).

### Geometry:

Our goal when developing PANel was to be able to calculate design allowables for wing and fuselage panels. These aerospace structures have been historically made of a skin on which are attached stiffeners in the longitudinal direction (stringers) and in the case of the fuselage also in the transverse direction (frames). A general schematic view of aircraft structures is given in Figure 4. Since the function of the stringers is mostly to increase the out-of-plane stiffness of the panels, they need to have a high inertia and therefore are usually made of a thin web with top and bottom flanges preventing the web from locally buckling. In PANel, ten parameters are used to describe the stiffener geometry, which covers a wide range of shapes (see Figure 1). Six parameters are driving the meshing, while the four remaining parameters correspond to the thickness of the different members of the stiffener (flanges and web), which are used as numerical parameters in the FEA (see section *Modeling* below).

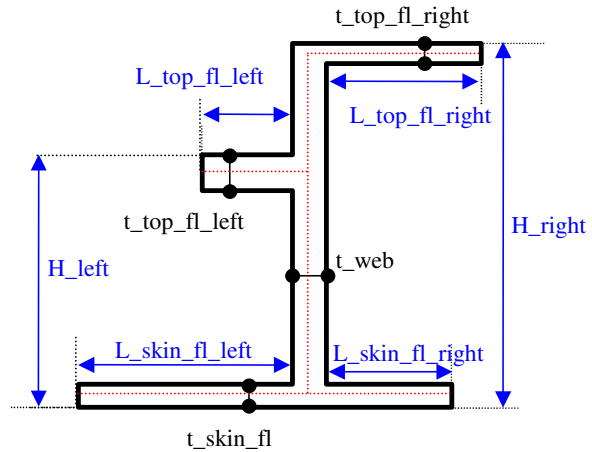


Figure 1 : Parameters used to describe the stiffener geometry

To keep the meshing simple, only the stiffeners perpendicular to the crack are accurately represented with shell elements (see Figure 5). Ribs are usually not represented in the wing modeling, which is subsequently a simple skin-stringer panel. Only transverse cracks (perpendicular to the stringers) are analyzed in that case. In the fuselage, both transverse and longitudinal cracks need to be studied. Consequently, depending on the user choice in the input file, the stringers (respectively frames) will be represented with shells whereas the frames (respectively stringers) are represented by beam elements. Bending and twisting inertias are required in the input file for adequate beam modeling along with the cross-section area of the stiffener. Symmetry is used with respect to the crack plane, which is restrictive in the fuselage configuration. The crack indeed has to be located in the middle of a bay (zone between 2 stiffeners - see Figure 5). A further development currently under way is to represent cracks running along a stiffener, which requires many changes such as not using symmetry and representing stiffeners with shells in both transverse and longitudinal directions.

The following section describes the procedure used to build a transverse crack in a fuselage panel. Other configurations are built similarly. The width of the panel is defined by the number of stringers to be modeled and the stringer pitch (distance between stringers). A flat stringer bay is first generated, corresponding to a stringer on top and in the middle of a piece of flat skin as wide as the stringer pitch. This bay is then duplicated as many times as the specified number of stringers.

Similarly, the user defines the number of frames he wants, and the length of the panel is calculated accordingly by PAnel. Beams representing frames are built so that the right number of bays is modeled. Finally, the panel can be curved according to a radius defined in the input file.

## Modeling

### *Mesh and elements*

Models are built with 4-nodes 3D linear shell elements (type 75 in MSC.Marc). Shells are located at the neutral fiber - or mid-thickness - of the 3D solid (represented as red dotted lines on Figure 1) to ensure the proper calculation of the stiffness of the real structure. The thickness of each element has to be provided: it is used by the software when compiling the stiffness matrix [6]. For simulating thin-walled aluminium structures, these elements were proven to be more reliable than the 8-node full 3D linear elements in many studies carried out in the research center in the past, especially when loaded in bending. An intensive study on Stress Intensity Factor calculation also validated the results obtained with shell elements. It gave some guidelines that are carefully respected in the models generated by PAnel.

One very important thing is the mesh size at the crack tip. Since the Fracture Mechanics methods used are based on the elasticity theory, yielding is not permitted in the skin. Theory then shows that the stresses are infinite at the crack tip, but rapidly decrease when going away from it. A very fine mesh is thus required to capture the stress gradient as well as possible. It is usually even recommended to use 8-node quadratic shells at the crack tip and place the mid-side node at  $\frac{1}{4}$  of the edge, closer to the crack tip. However, we found that subdividing the elements another time instead of switching to quadratic elements provided the same level of precision. To take into account that need for small element sizes and not generate too large models, automated meshing is performed in two steps. First, the entire panel is meshed with a global mesh size defined by the user relatively to some geometrical parameters (rivet pitch or stiffener pitch). Then, the mesh is locally refined on the crack plane by successive waves, each dividing the mesh size by two (see Figure 5). Usually, mesh is locally 8 or 16 times smaller than in the rest of the panel.

### *Skin-stiffener junction*

Three types of assemblies can be modeled in PAnel (see Figure 5). **Riveted structures** are modeled by putting springs between nodes of the stiffener flange and nodes of the skin at the rivet locations. A user-defined MSC.Marc sub-routine has been included in PAnel that allows the user to further describe the rivets by specifying a *Force / Deflection* curve. Modeling of **bonded structures** also uses links, with the difference that *all* nodes of the stiffener flange are this time *rigidly* linked to nodes of the skin. In **integral structures**, the bottom flange of the stiffener and the skin are merged, being represented by only one layer of shell elements whose thickness is the sum of the flange and the skin thicknesses. In built-up and bonded structures, the crack is only growing in the skin whereas in the integral structures, there is no discontinuity preventing the crack from growing in the stiffener. However, little is known about the way the crack will behave when reaching the stiffener. In the absence of relevant data on this subject, it was decided to include an additional user-defined ratio in the input file: crack growth rate in the stiffener divided by crack growth rate in the skin.

### *Boundary conditions*

To simplify the modeling, symmetry is used in the longitudinal direction. The crack lies in the symmetry plane, as can be seen on Figure 5. Nodes belonging to the symmetry plane are restricted from displacement in the longitudinal direction, except nodes within the crack location, thus simply defining the crack. At the opposite end of the rectangular panel, tension conditions are applied being either displacement or stress controlled. The user simply specifies which displacement or stress he wants in the input file. On the remaining longitudinal edges, the user chooses between free edges or symmetrical boundary conditions. Finally pressure can be applied to simulate fuselage panels: in that case, the pressure indicated in the input file is applied to the skin only.

### *Material models*

No yielding of the skin is included in our modeling, since it would violate the LEFM theory. However, it is very important to enable the yielding of the stiffeners in order to accurately derive the residual strength of the panel. In the case of riveted or bonded structures, the final failure of the structure indeed most often occurs after the crack has crossed the adjacent stiffeners (cracks longer than 2 bays).

The intact stiffeners then provide some additional resistance by crack bridging. If the stiffeners were to remain elastic, that resistance would be much over-estimated. The user can subsequently choose to include yielding in the stiffeners. A simple Von Mises criterion is then applied in conjunction with a *stress / plastic strain* curve given by the user in a separate file. Another material feature included in PAnel is the possibility to define different Young's modulus for the skin and the stiffeners, thereby allowing for the modeling of structures including Aluminum-Lithium alloys with higher Young's modulus. A single Poisson's ratio is defined for the whole model.

### **EXAMPLE: CRACK RETARDATION**

In addition to the evaluation of weight savings achievable with new advanced aluminum alloys, PAnel also enables us to test some innovative ideas. Complete cost and weight studies can thus be carried out and shown to airframe manufacturers. New concepts bringing cost and/or weight savings can thus be further investigated more specifically by the OEMs. One example is the work that was performed at the Research Center CRV on integral structures.

### **Concept and models**

Integral structures offer significant cost savings whether they are machined out of a plate or result from the welding of several pieces [8]. The gain in assembly time along with the reduction in the number of parts lead to decreased costs. Riveting is indeed labor intensive, and the management of sometimes several hundreds of small parts for a single sub-component is costly. However, integral structures are intrinsically less fail-safe: cracks can propagate from stiffeners to skin and vice versa, providing only one load path. The final weight of an integrally stiffened panel driven by damage tolerance can therefore be higher. Apart from improving the properties of the material, there are also some ideas to modify the design of integral panels in order to increase the damage tolerance. One of the advantages of integrally machined structures is that for a similar cost, the panel geometry can be locally tailored. To exploit this fact, a study was launched on the influence of the stringer shape and stringer pitch. PAnel was used to calculate SIF in three different configurations (see Figure 6). The first is the reference, close to what is currently done in the aerospace industry, with a stringer pitch of 150 mm and a T-shaped stringer. In the second configuration, the stiffener has a very simple rectangular shape, with the same section as the

reference T-shaped stringer and the stringer pitch remains unchanged. Finally, a rectangular stiffener is also used in the third configuration, but half as wide as previously and with a stringer pitch also reduced by half. Since the skin thickness is the same in the three models, the total weight is kept constant.

### **Results**

The *SIF vs half crack length* curves as produced by PAnel are given in Figure 6. In all configurations, a central stringer is broken and the crack is symmetrical with respect to that stiffener. Schematic illustrations of the three geometries are superimposed to the graph and scaled to the half crack length abscissa. It therefore helps visualizing the location of the crack tip on the diagram: for each point of the curve, a vertical line drawn down to the corresponding diagram (same color) will give the vertical crack front location. The thick black curve gives the theoretical value of SIF in an infinitely wide cracked plate without stiffeners subject to the same tensile stress. The three configurations exhibit the same pattern in which SIF decreases when the crack comes closer to the stiffener and then starts increasing as the stiffener begins to break, reaching a peak as it exits the then broken stiffener. SIF values are alternatively below and above the black curve corresponding to the un-stiffened plate. This is a logical behavior of stiffened structures subjected to crack growth, simply highlighting the fact that the metal distribution is different in stiffened structures generating higher stresses in the skin when the crack is far from the stiffener (see for instance [9]).

What is more interesting is the fact that the SIF values oscillate around the black curve but further away from it in the cases of rectangular stiffeners. To translate that into crack propagation terms, the crack will slow down in a much more significant manner when getting closer to a rectangular stiffener than to the T-shaped stiffener. Similarly, the crack growth rate will be much higher when the rectangular stiffener will start to break than in the T-shaped configuration. However, summing these effects will not produce the same overall life for the three configurations. To derive the number of loading cycles needed for a crack to grow from detectable (50 mm) to critical (300 mm), the SIF values indeed have to be integrated or more exactly an equation similar to  $dN = k da / (SIF)^m$ , where  $dN$  is the increment in number of cycles and  $da$  the incremental crack advance.

Curves giving the evolution of the crack length with respect to the number of loading cycles have been computed in the three configurations and are given in Figure 6. A Forman law for alloy 2024 has been used.

Results show that it takes much longer for the crack to reach its critical length in the case of rectangular stiffener, and even more so when the stringer pitch is reduced. As the crack growth is reduced, a much longer time is required for the crack to go through the entire stiffener. Even if its growth is faster after exiting the stiffener, the final time or number of cycles is much higher. One way of illustrating the phenomenon is to draw an analogy with biking in the mountains. Let's consider two bikers who have the same origin and destinations but take different routes. One has a longer ride, but is riding only on small slopes whereas the other one takes the shortest route, which requires climbing a big mountain. Due to the high sensitivity of bikers' speed to the slope, it is likely that the second rider will arrive later than the first. The pad-up plays the role of the mountain on the crack path, keeping the crack speed in the low regime for a longer time.

Fatigue propagation life can then be multiplied by 2.5 when using the rectangular pad-ups with halved stringer pitch. One might want to further divide the stiffeners and reduce the stringer pitch, but results obtained for a plate without stiffeners are not as good, showing that there is an optimum geometry.

## **Validation**

The DT criterion is only one of the criteria used in aerospace design. Resistance to compression is also very important and it is therefore not possible to replace current high stiffeners by flat rectangular stiffeners, whose inertia is too low. However, it may be interesting to add such small pad-ups to the skin between the stringers to provide additional fatigue propagation life without adding weight. To test this idea and gain more confidence in this crack retardation concept, a test was performed on two plates, one including the small rectangular pad-ups and the other having a constant thickness skin (see pictures in Figure 7). Since the aim of the test was to simulate a simplified wing panel; three big rectangular stiffeners were located at the center and near the edges of the plate to represent stringers. Both panels had the same width and length but most importantly had the same weight. Numerical analyses have been performed and the excellent agreement between FEA prediction and test results can be seen on

Figure 7. For both samples, 3 curves are plotted representing the right and left half crack length (measured from the central broken stiffener) and the mean value. Due to the use of symmetry, only one curve is obtained from Numerical Modeling of both panels. They are plotted in light and dark blue on the graph.

Results obtained with our MSC.Marc-based FE models have therefore been shown to be reliable. The interest of simple rectangular crack retardation features has also been demonstrated.

## **CONCLUSION**

A tool has been developed by Alcan enabling much faster and reliable design studies on damage tolerant driven aerospace structures. PAnel acts like a large macro, launching commercial software MSC.Marc Mentat and MSC.Marc to build FE models of stiffened panels. The user simply defines the configuration he wants to study by specifying a small number of parameters in a text file, and PAnel automatically generates models, launches calculations and post-treats results. Final output files are subsequently analyzed in MS Excel. Thanks to this internal tool, which is now validated with crack propagation tests in sub-scale integrally stiffened panels, Alcan has the capability of evaluating the weight savings associated with the use of its new advanced aluminum alloys in aerospace structures. Furthermore, innovative ideas where the material aspect is important can be investigated and if proven interesting for the aircraft manufacturers, more specifically addressed by them in a second phase. An example is the crack retardation resulting from the introduction of small rectangular pad-ups between stringers. A first study conducted with PAnel showed that the overall crack propagation speed could be divided by 2.5. Tests were performed on a configuration designed with PAnel and the results were in excellent agreement with the FE predictions. A complete study on a wing panel is under way to determine the weight savings that could be achieved with this new concept. New features will also be implemented within PAnel. A new procedure for the mesh refinement at the crack tip is currently under investigation. The capability of modeling a crack running along a stiffener and not in the middle of a bay will also be added in the near future. Finally, a more global reflection on 3D modeling of fracture is currently undertaken and could lead to much more complex FE model generation.

## REFERENCES

1. Niu M.C.Y., *Airframe Structural Design*, 2<sup>nd</sup> edition, Hong Kong 1999
2. Swift T., *Fracture analysis of stiffened structure*, Damage Tolerance of Metallic Structures: Analysis methods and application, ASTM STP 842, American Society for Testing and Materials, 1984, pp 69-107
3. Southwest Research Institute, *NASGRO® 4.1 User's manual*, 2003
4. Schwalbe K. H., Setz W., *R-curve and fracture toughness of thin sheet material*, J. Test and Eval., vol 9, 1981, pp 182-194
5. Broek D., *Elementary Engineering Fracture Mechanics*, Kluwer Academics Publishers Group, 1982
6. MSC Software, *MSC.Marc Theory and User guide*, 2003
7. MSC Software, *MSC.Marc Mentat User's Guide*, 2003
8. Munroe J., Wilkins K., Gruber M., *Integral Airframe Structure (IAS) – Validated Feasibility Study of Integrally Stiffened Metallic Fuselage Panels for Reducing Manufacturing Costs*, NASA/CR-2000-209337, 2000
9. Poe C. C., *The effect of broken stringers on the stress intensity factor for a uniformly stiffened sheet containing a crack*, NASA Technical memorandum, NASA TM X-71947, 1973

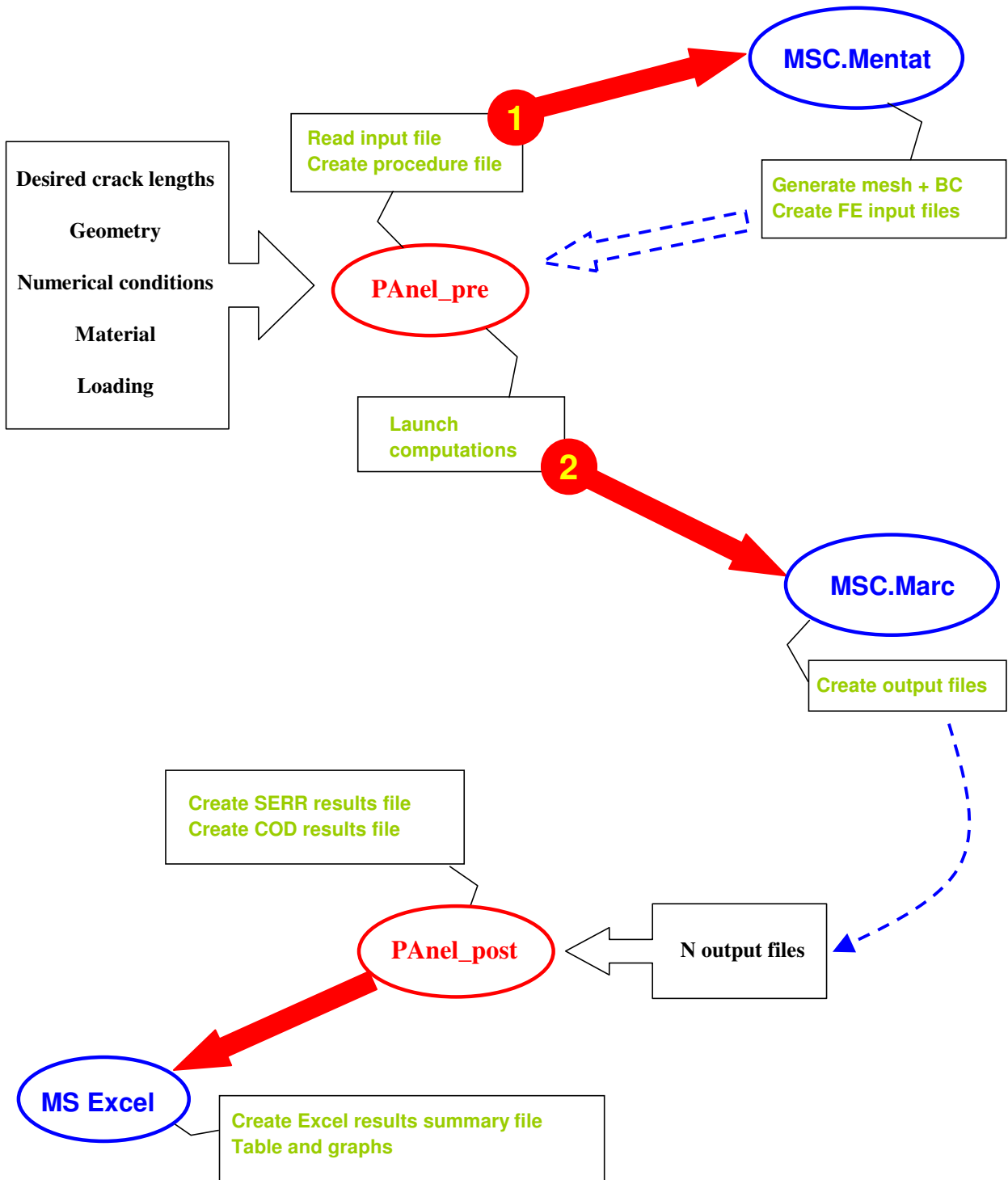


Figure 2 : Procedure used within PAnel - PAnel has 2 distinct executable files:

- 1) PAnel\_pre generates models and launches FE analyses
- 2) PAnel\_post extracts information from FE output files and stores it in a text file further transferred to MS Excel® via a macro

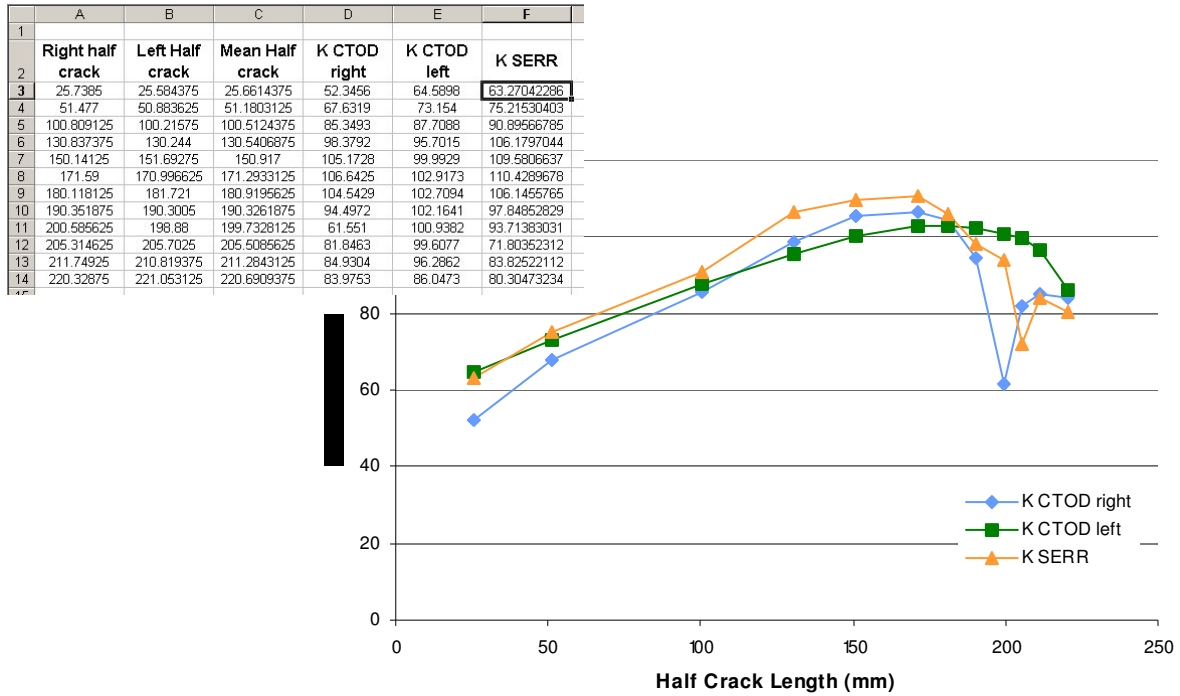


Figure 3 : Panel results in MS Excel® - SIF vs crack length curve and table of numerical results

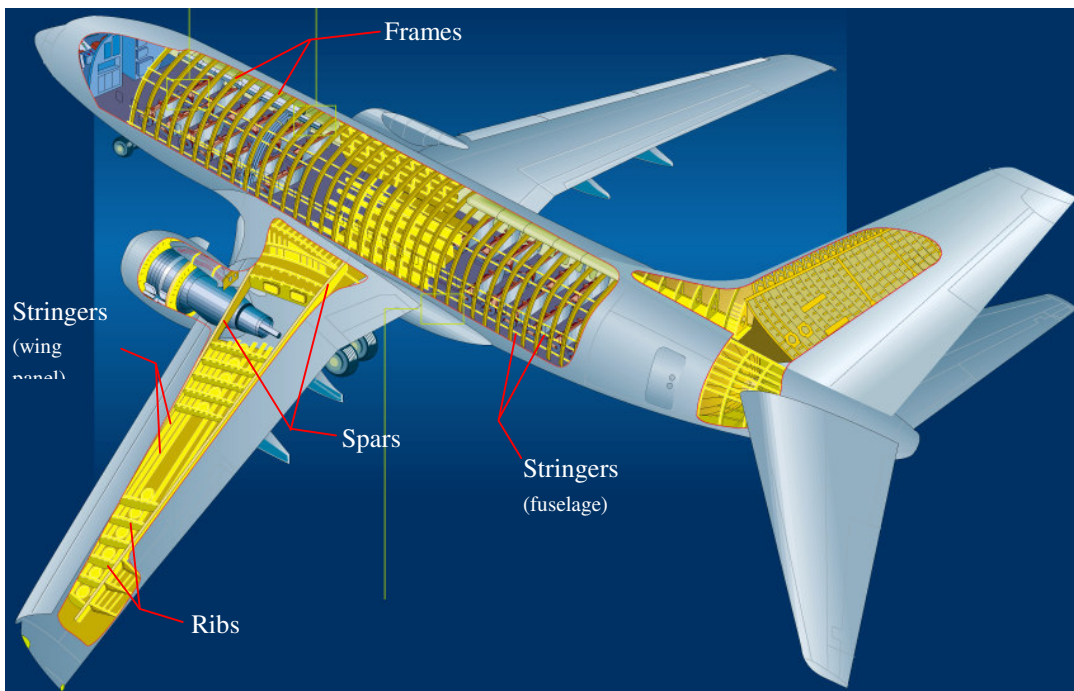


Figure 4: Typical aircraft structure - Most of an aircraft is made of thin aluminum sheets on which are assembled extruded stiffeners. The fuselage contains longitudinal stringers and circumferential frames to sustain a multiple bending /pressure loading. Wings are boxes containing 2 panels stiffened longitudinally that are linked together with ribs in the transverse direction and spars in the longitudinal direction.

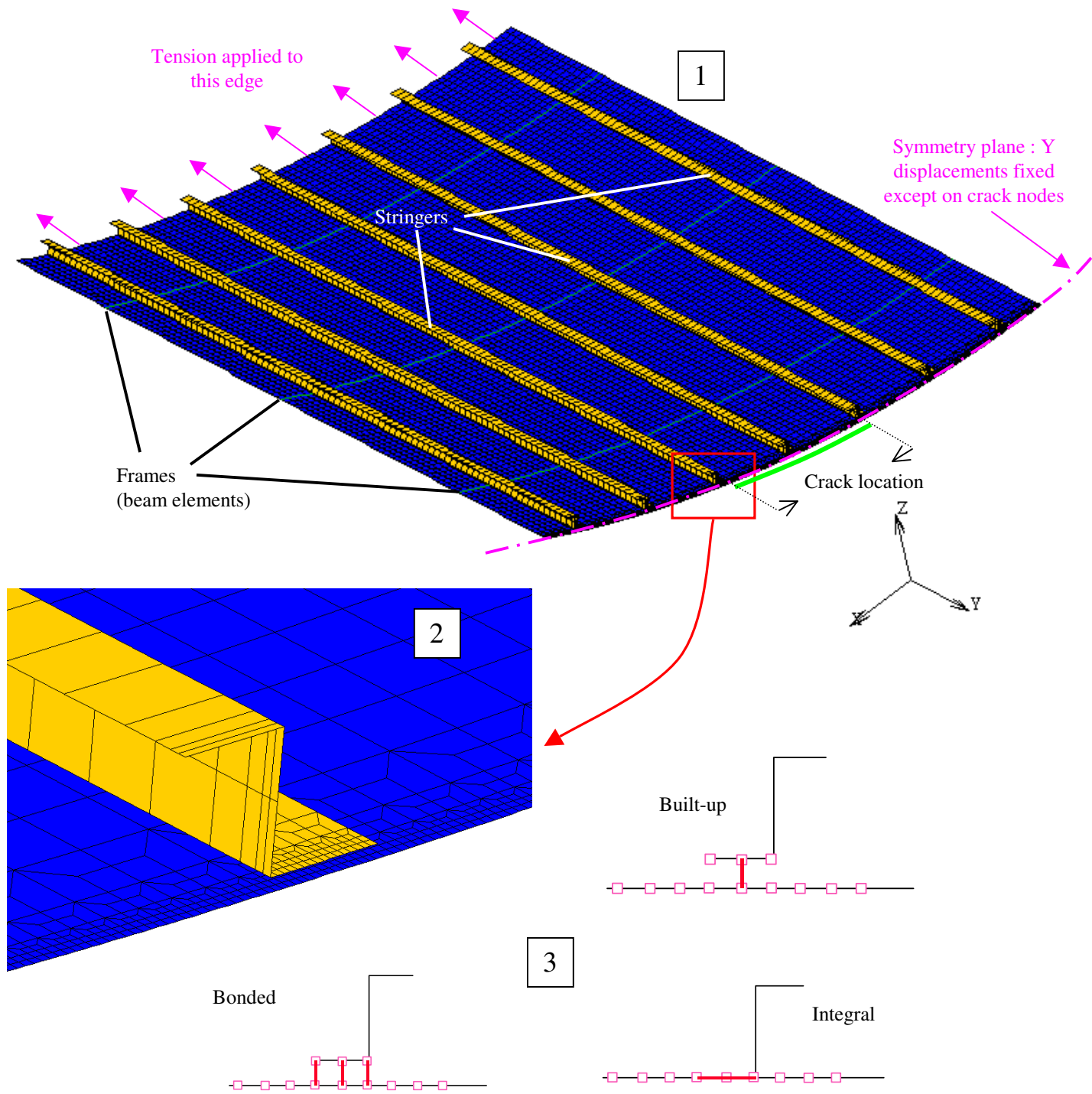


Figure 5: Finite Element Model generated by PAnel.

1) Stiffeners perpendicular to the crack are represented with shell elements whereas stiffeners parallel to the crack are represented by simple beams. The crack lies in the symmetry plane, where nodes outside the crack location are restrained in Y displacements. Tension is applied on the opposite transverse edge.

2) Zoom on the crack location: Note the mesh refinement, required condition to reliable SIF computation.

3) Schematics of sections for different assemblies. Built-up and bonded uses 2 layers of shells whereas in the modeling of integral structures, skin and stiffener flange are merged.

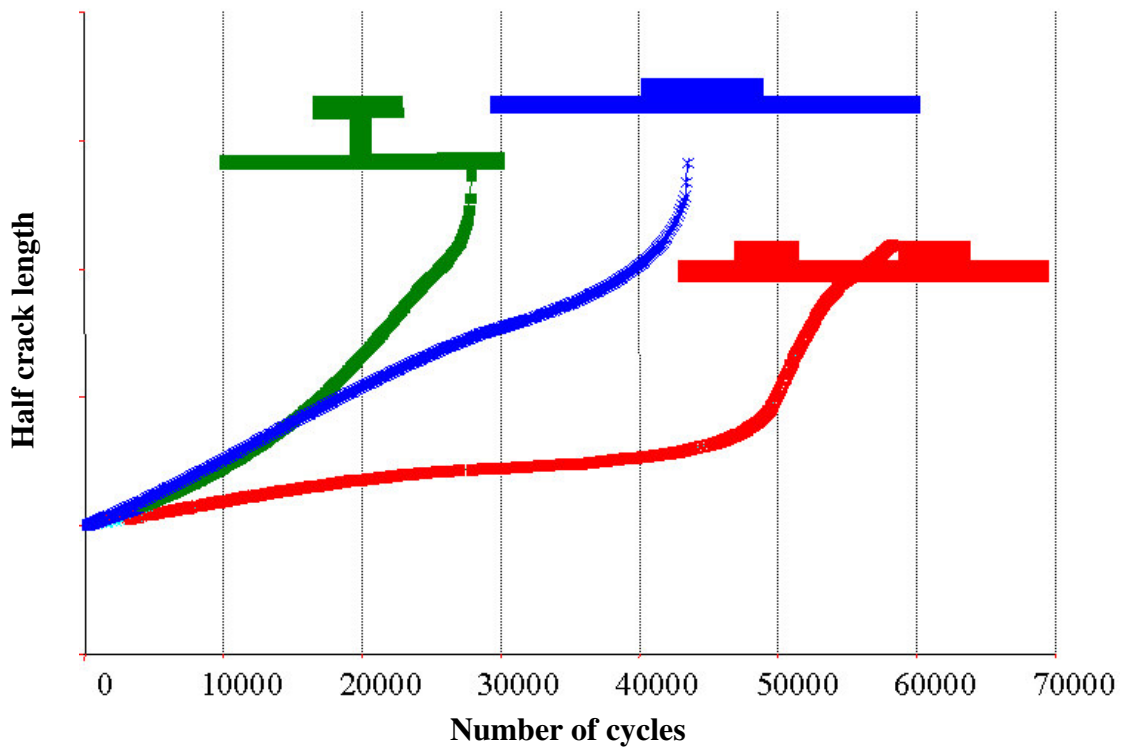
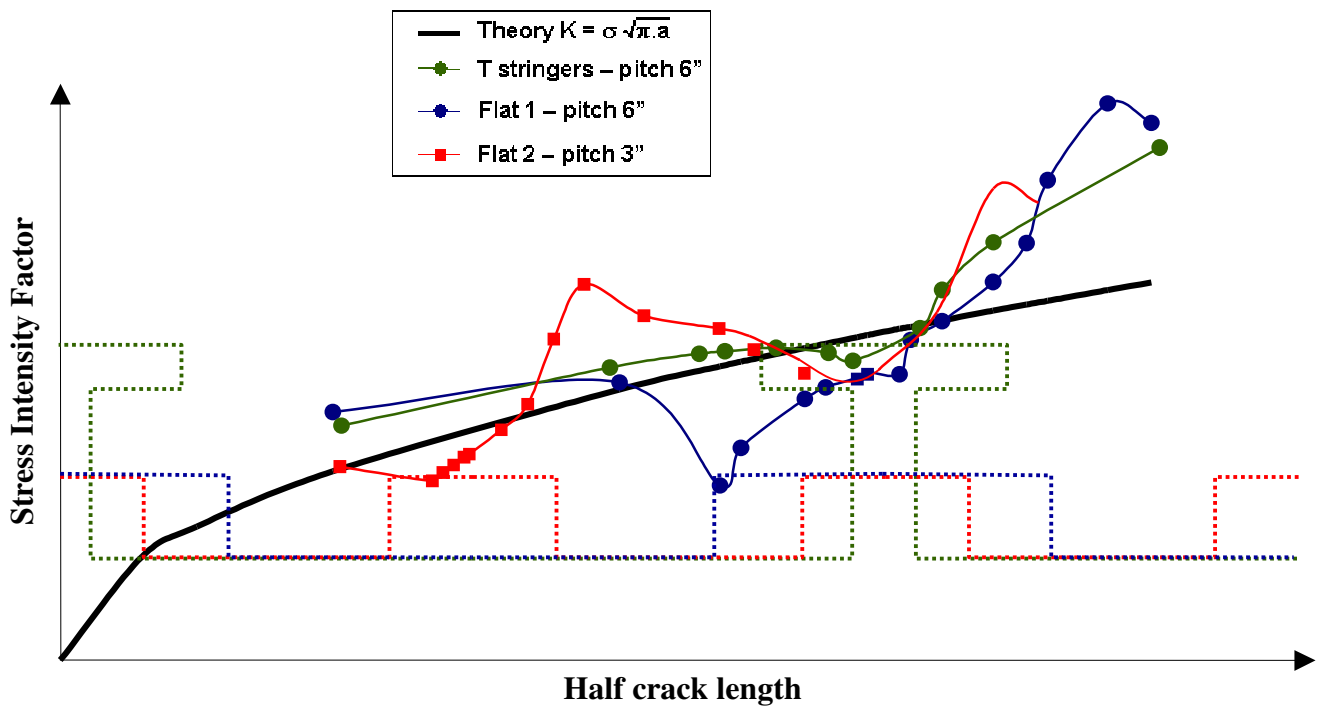


Figure 6 : Crack propagation in integral panels with different shapes of stringers.  
 Top: Results obtained with PAnel  
 Bottom: Predicted number of cycles required by the crack to grow from detectable to critical

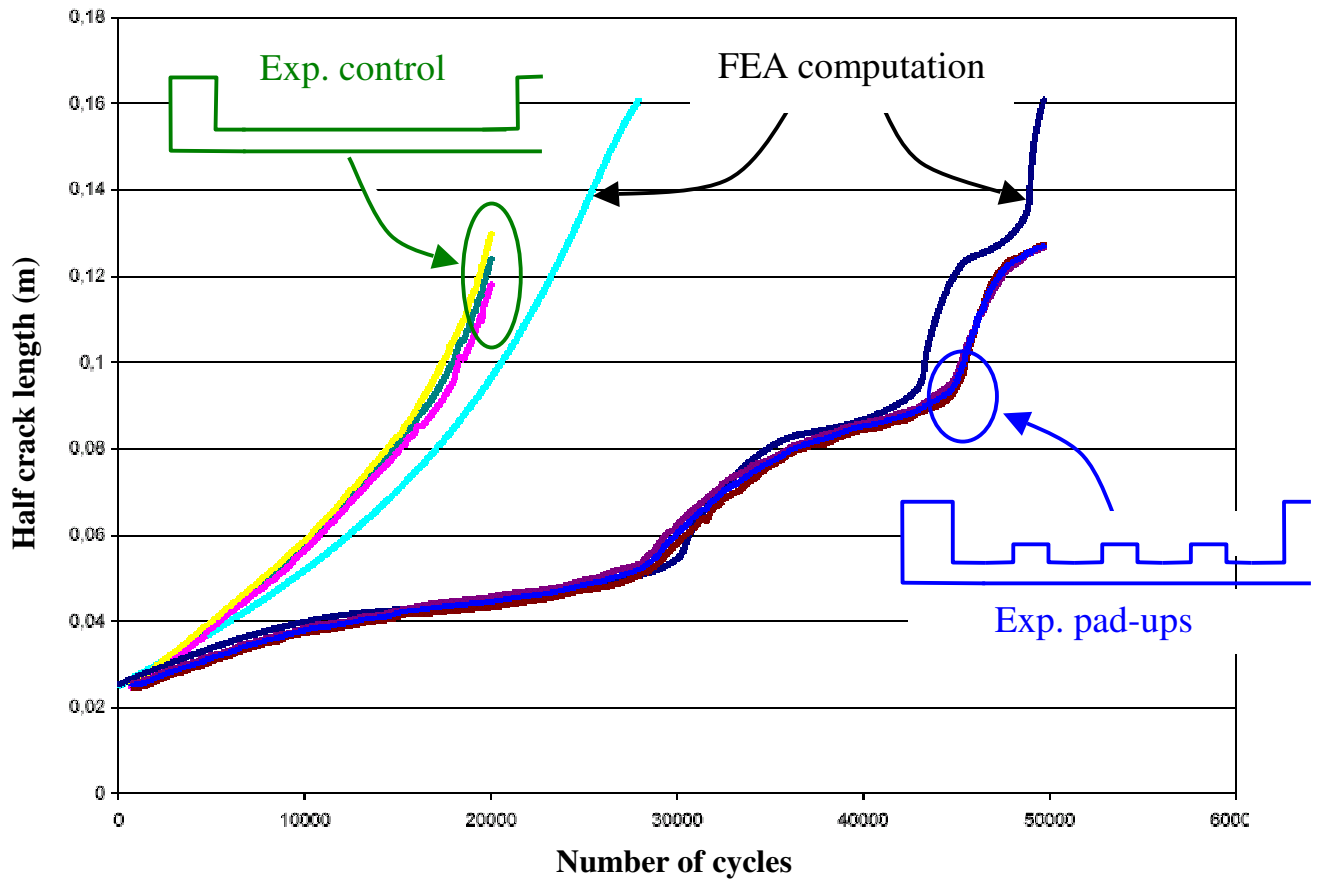
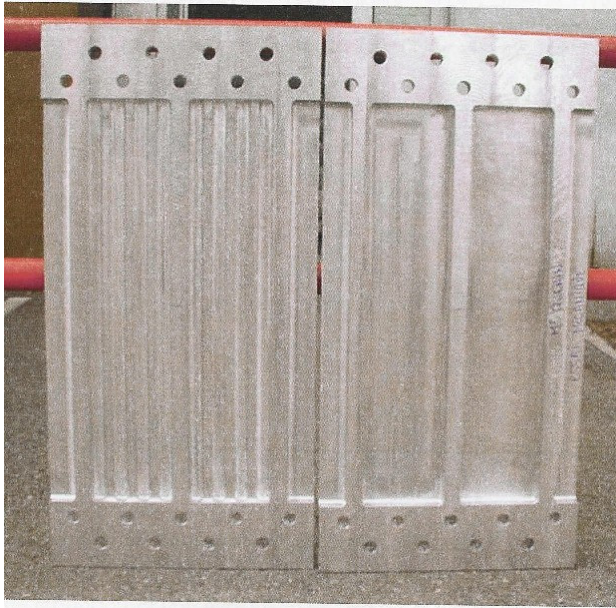


Figure 7: Experimental validation of the crack retardation concept. Top: pictures of reference plate and plate with rectangular pad-ups before testing – Testing completed on crenellated plate. Bottom: comparison between crack growth predicted by PAnel and test results