Experimental and Numerical Investigation on Fatigue Failure of Composite Helicopter Main Rotor Hub

M. Giglio[1,2], A. Manes[1], F. Viganò[1]
[1] *Politecnico di Milano, Dipartimento di Meccanica, Italy*
[2] *Corresponding author, E-mail: marco.giglio@polimi.it*

The Main Rotor Hub is a very critical component of the helicopter considering that its failure may determine catastrophic consequences for the whole rotorcraft. A complete experimental and numerical analysis is so necessary to certificate the fail safe and eventually the damage tolerant behavior of the component. In this paper the fail safe behavior of a composite titanium-graphite Rotor Hub is analyzed. The component, with artificial technological defects, was tested with complex contingent fatigue load in order to cause the failure of the titanium section. The failure started in proximity of a high stressed area and it propagated quickly in the whole titanium section but without involve the surrounding filament winding graphite, that is a fail safe device. The behavior of the hub during the whole test is simulated, with good accuracy, by means of a complete FE model that reproduces also the 3D propagation of the crack in the titanium section.

1. INTRODUCTION

Main Rotor Hub (MR/H) is one of the most critical component in a helicopter; its failure may determine the loss of the rotorcraft. For this reason, MR/H, like others critical structural components, is subjected by JAR 29.571 rules to obtain the certification. JAR 29.571 imposes a complete fatigue tolerance evaluation that is comprehensive of safe-life, fail-safe, and flaw tolerant evaluation.
The main aim of this work is just a complete experimental and numerical analysis on the crack propagation behavior of a real MR/H (with artificial manufacturing defects) when subjected to fatigue loads. In particular a numerical analysis has been carried on in order to reproduce in full detail the behavior of the component during the application of the fatigue loads and subsequently for how concern the 3D crack growth (nucleated in the most stressed zone) until the critical value for the complete failure of the titanium section. The numerical 3D crack growth evaluation has been made using a main finite element model with a dedicated FRANC3D submodel.
A similar work has been described in [1] which analyzes the 3D crack growth of a main rotor damper attachment lug with the support of experimental test data. In this work, a finite elements lug model with FRANC3D submodel was carried out; the result was a crack propagation with a path very similar to the experimental evidence.
For geometrically simple rotorcraft components, a simplified numerical crack evaluation can be done using NASGRO and AFGROW codes; acceptable results

are obtained in [2]. For complex geometries and loads, like spiral bevel pinion gear, a complete arbitrary 3D crack growth analysis was made in other works [3-4] using a integration of FRANC3D with a finite element code; results are supported by agreeing experimental data. For shell helicopter structures [5], like fuselage, a good agreement with the experimental data was obtained using a complete automatic 3D shell crack propagation code. The code developed by the authors [6] isn't applicable for general solid 3D crack propagation.

1.1 Description of the MR/H

The MR/H considered in this work belongs to a medium lift, five blades helicopter. It is composed by a main titanium (Ti-6Al-4V) hub (blue in Figure 1) winded by a graphite filament (yellow in Figure 1). The composite filament winding represents a fail-safe device and it is jointed to the titanium part by a structural adhesive film. The blade links are situated on the hub's handles. The internal surface is connected with a shaft (mast) which transmits the motion from engine through rotor transmission. Moreover conical surfaces constrain movements in axial direction, respect the mast.

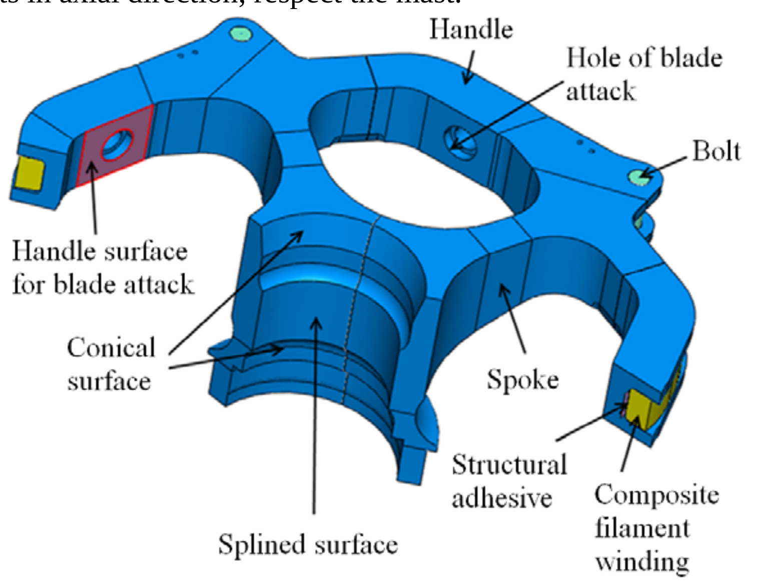


Figure 1: Main Rotor Hub and its nomenclature

The Ti-6Al-4V hub was obtained by a forging and machining process followed by a shot peening treatment on the whole surfaces. The composite filament winding is composed by carbon fibers into an epoxy matrix. Artificial defects were created in the composite filament winding in order to reproduce manufacturing defects that may occur during the filament winding.

1.2 Applied loads on the MR/H

During the operational condition, the MR/H is subjected by four kinds of forces: centrifugal (CF), traction beam (TB), traction chord (TC) and damper forces (DF).

CF derives from inertial effects due to the rotation, TB is due to the lift and TC is due to the drag of blades. Damper forces born due to damping effect of blades was neglected in this evaluation. Figure 2 shows positions, directions, numerations and positive versus of each force.

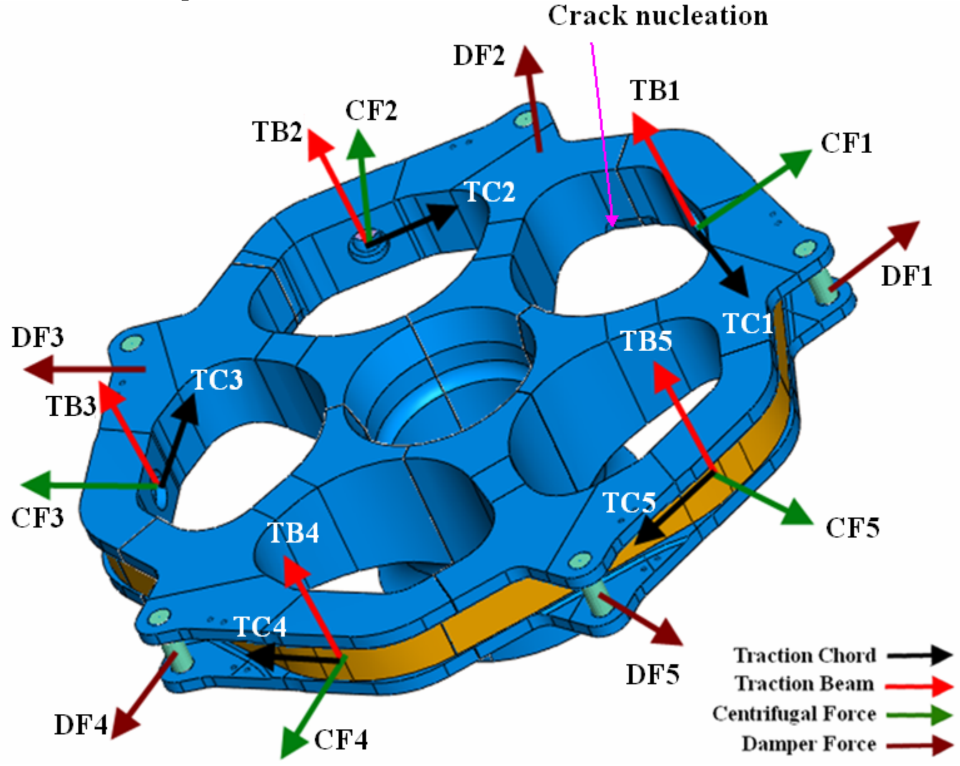
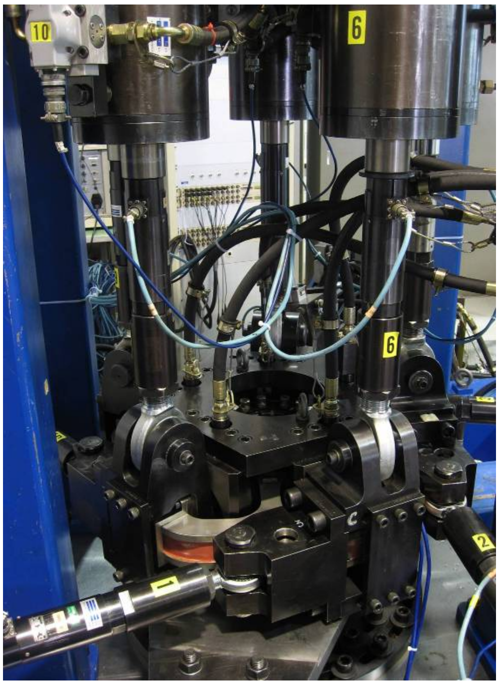


Figure 2: Forces on the main rotor hub

2. EXPERIMENTAL TEST

The set up of the test was designed in order to apply all the principal forces that involve the component in their exact magnitude and direction: a picture of the test device, with actuators and test specimen, is shown in Figure 3. The test rig was composed by a horizontal plate with five 100 KN maximum force actuators for the Traction Chord (TC) load and a vertical structure with five 200 KN maximum force hanged actuators for the Traction Beam (TB) load. Each actuator was monitored by a 100 KN load cell and applies the load in the blade attachment section. To avoid the complexity of a real blade attachment with elastomeric bearing, dedicated grips were designed for the actuators in order the reproduce on the hub a realistic load transfer. Moreover other five dedicated actuators were used inside the arms of the Hub to apply a radial load that simulates the centrifugal force (CF). The load application of these actuators was monitored by a single pressure transducers. Digital command and control instrumentation were used. The M/R Hub was constrained to a dummy must. Six strain gauges were positioned on the hub, also in the most critical zone. More details about position and results from strain gauges for the validation of the finite element model are reported in 3.4 paragraph.

The test was executed in C4 Laboratory of Politecnico di Milano, Dipartimento di Meccanica. An amplified Start Stop load spectrum, described by nine load steps, was applied to the specimen, Figure 3.



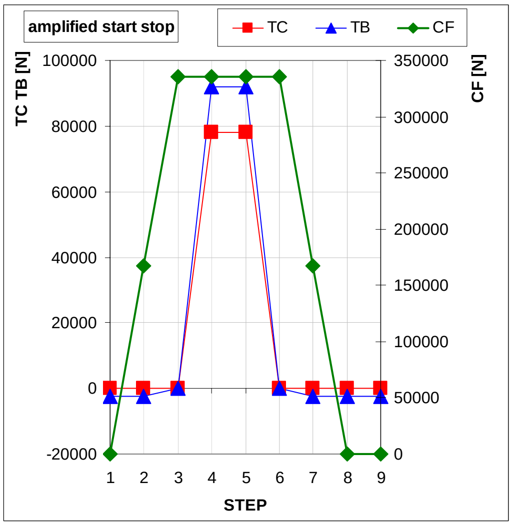


Figure 3 –The test rig with the specimen and the actuators (on the left)) and the amplified Start Stop load spectrum applied

At 59032 cycles a visual inspection showed no damages. At 62305 cycles the test was stopped by the TC1 inferior peak alarm. A visual inspection showed a large crack involving the whole titanium section between TC1 and TC2 actuators hinges, Figure 4. Non Destructive Inspection was carried out with the result that no breaking of the circular strap was happened. The failure involved only but completely the titanium section. Thus a crack nucleated (from the threshold of barely visible) and propagated, to an instable length value, in less than 3273 cycles.

3. NUMERICAL SIMULATION

Numerical evaluation executed to support the experimental tests can be shifted in two phases. The first phase consisted of a main finite element (FE) model of MR/HUB that was used to estimate the stress behavior and in particular the most stressed zone. In the second phase an intensive use of a FRANC3D/BES submodel together with the main FE model was accomplished in order to simulate the 3D crack propagation in the hub handle.

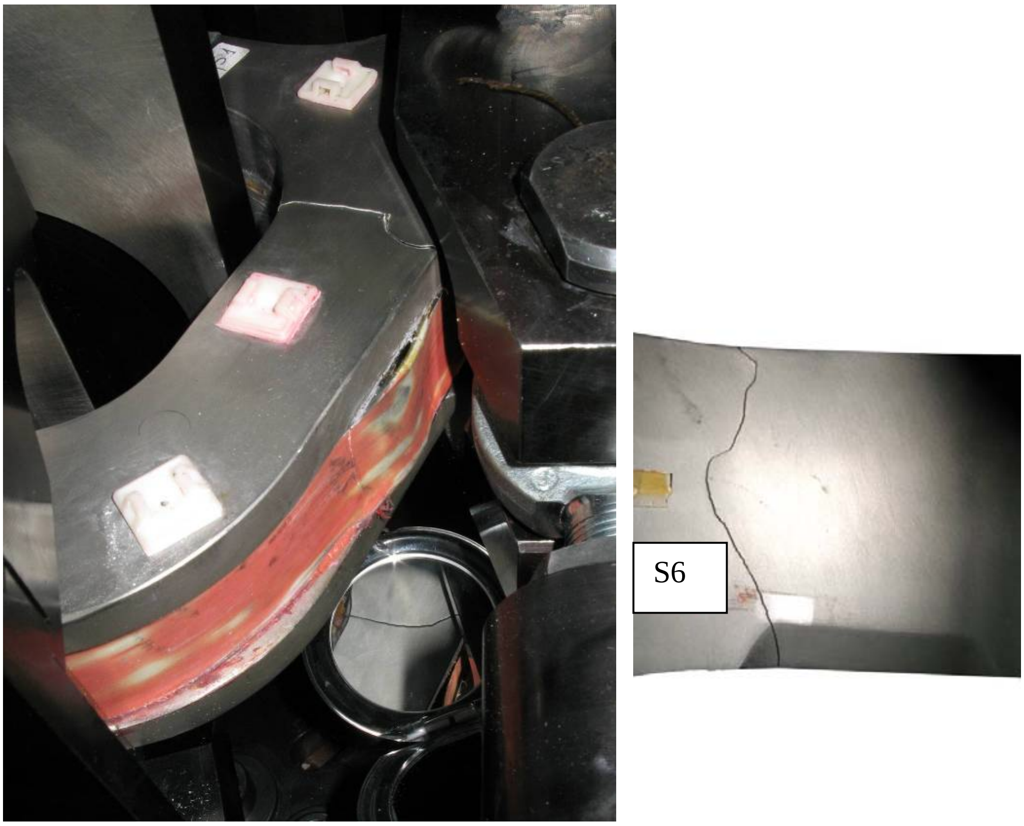


Figure 4 – The crack on the whole titanium section (on the left) and a detail of the internal side of the handle, on the right (S6 label indicates the position of a strain gauges): this is the most stressed zone

3.1 Description of the main finite element model

The main FE model of MR/H was built using Abaqus®/Standard v6.7 finite element commercial software. The model was comprehensive of a titanium hub, composite filament winding and all the features under test, exception for the technological defects introduced in the real component, to avoid a much more complex model. However, as it is possible to see in the next paragraph, the strain gauges validation shows a very good agreement.

All materials of the components were modeled with linear elastic behavior. The titanium hub was modeled with 3D modified quadratic tetrahedral elements (C3D10M); the mesh was refined near the geometrical notches. The composite filament winding was modeled as a solid part with orthotropic transversally isotropic material properties along the winding direction; 3D modified quadratic hexahedral elements (C3D20) was used to model it. Also the 0.19mm thickness film of structural adhesive between titanium hub and composite filament winding was considered; for this aim some 3D cohesive finite elements with linear elastic material proprieties provided by the manufacturer were used. Figure 5 shows the mesh on one fifth geometry of each component modeled on the main FE model.

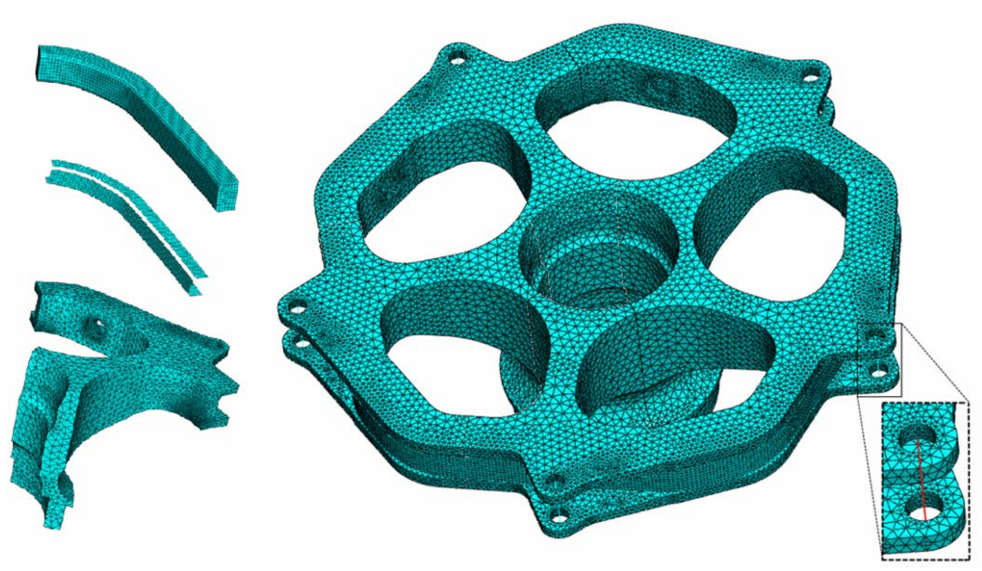


Figure 5: On the left the geometry of each component meshed (from the top to down: Filament winding, structural adhesive and hub); on the right the main FE model with the beam elements inside the damper attachment zone in evidence

The bolt was modeled with quadratic beam elements; the whole FE model has 877,772 nodes and 317,210 elements.

The rotations and the axial translation of the conical surfaces were constrained. In Figure 6 the constraint and the force application zone in the FE model are showed.

3.2 Validation of the main FE model

On the MR/H six strain gauges were placed in order to compare the strain measures with the numerical results during loads application. The position of the strain gauges is shown in Figure 7; the comparison of the results obtained from the FE model and the experimental test (Table 1) gives an evidence of a validity of the FE model.

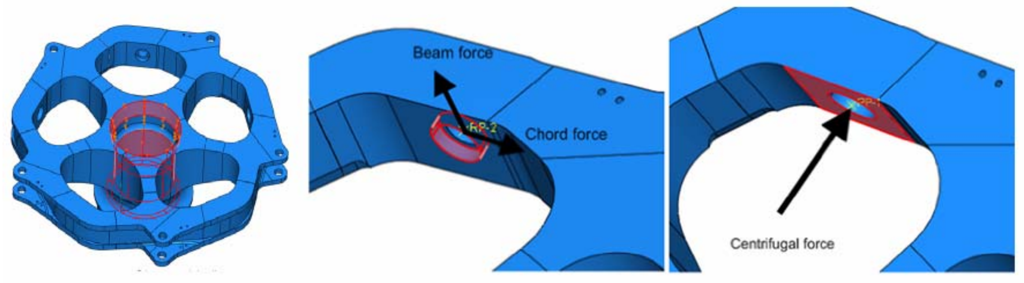


Figure 6: The first on left, the surfaces constrained, the others are the surfaces on which the forces was applied

Using the FE model is possible to highlight the most stressed zone of MR/H. In particular the most stressed zone indicated by FE model is exactly where the crack nucleated and propagated until the complete failure of the titanium section, Figure 8. Moreover the FEM shows that the stress is mainly monoaxial with a principal stress in the hub tangential direction.

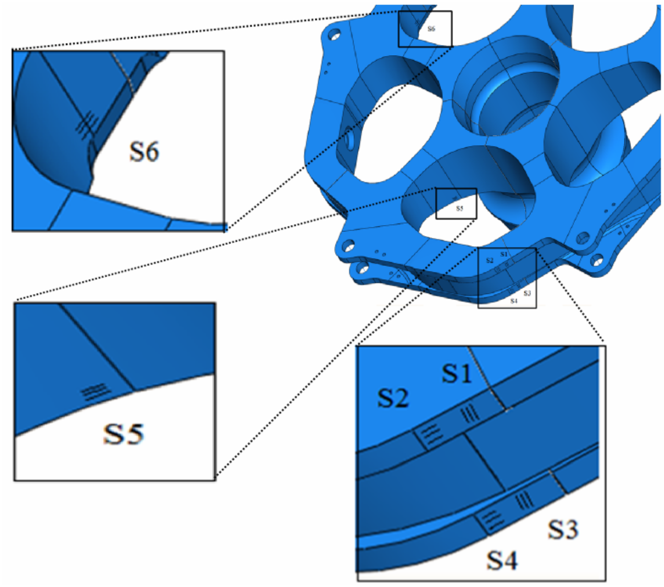


Figure 7: Surfaces on which the forces was applied

Table 1 – The strain gauges comparison

CF [N]	TC [N]	TB [N]	S1 [µm/m]		Error [%]	S3 [µm/m]		Error [%]
			Strain Gauge	**FEM**		**Strain Gage**	**FEM**	
335587	0	0	331	277	-16	305	281	-8
335587	30000	0	434	355	-18	414	359	-13
335587	0	30000	346	313	-10	289	235	-19
CF [N]	**TC [N]**	**TB [N]**	**S2 [µm/m]**		**Error [%]**	**S4 [µm/m]**		**Error [%]**
			Strain Gauge	**FEM**		**Strain Gage**	**FEM**	
335587	0	0	-974	-965	-1	-885	-957	8
335587	30000	0	-1308	-1290	-1	-1214	-1281	6
335587	0	30000	-1018	-1154	13	-841	-768	-9
CF [N]	**TC [N]**	**TB [N]**	**S5 [µm/m]**		**Error [%]**	**S6 [µm/m]**		**Error [%]**
			Strain Gauge	**FEM**		**Strain Gage**	**FEM**	
335587	78000	92000	5450	5346	-2	4860	4936	2

4. RESULTS AND CONCLUSIONS

The aim of the work was the FE simulation of the crack propagation phase in the Main Rotor Hub, starting from the crack nucleation in the most stressed zone and considering also the propagation until the final failure of the component.
The results obtained with the FE model were compared with the experimental data with the purpose to accomplish a reliable numerical model able to predict the behavior of the hub, also in the preliminary design phase.

The three dimensional crack propagation was carried out on a submodel of the MR/H realized using FRANC3D [7] code.
The geometry of the hub handle zone interested by the crack propagation, was first created in the Object Solid Modeller (OSM) and after imported in FRANC3D environment. The geometry was approximated in triangular surfaces in order to make an easy geometrical representation in FRANC3D (Figure 9).

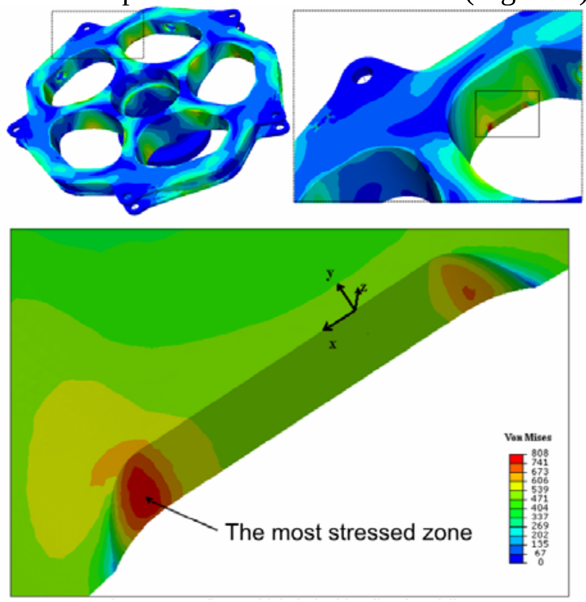


Figure 8: Most stressed zone in the MR/H

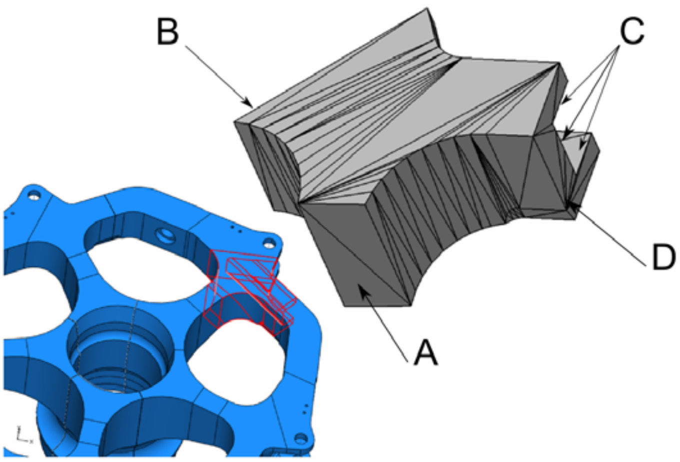


Figure 9: The FRANC3D handle submodel

On the sliced submodel surfaces (A, B, C and D, Figure 9) the stress field was transferred from the main FE model in order to reproduce the same stress condition. On the notch of the handle, an initial defect was placed. It consist of a corner crack with the characteristic dimension “c” of 0.5 mm (Figure 10), as a lower limit dimension for a barely visible flaw.

The submodel surfaces were meshed with triangular boundary elements in order to use Boundary Elements Solver (BES) that it is completely integrated with FRANC3D. Once the results was obtained, the stress intensity factors (K_I, K_{II} and K_{III}) along the crack front were estimated.
The direction and the extension of the new crack front were computed using the maximum hoop stress criteria and the power law [7].
A global remeshing was done and the new results were analyzed. These steps were repeated until the stress intensity factor K_I (mean along the crack front) reach the critical stress intensity factor (K_{IC} =1737 $MPa\sqrt{mm}$) of the material used, Ti-6Al-4V.

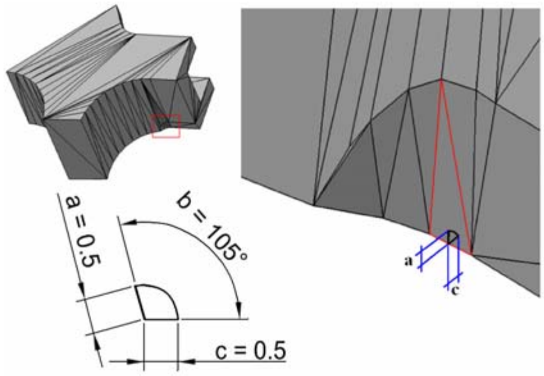


Figure 10: Initial crack in the notch of hub handle

Figure 11 shows the crack with a dimension of 3mm and 6mm: the crack propagation front remains enough regular and normal respect the maximum principal stress in the hub handle.

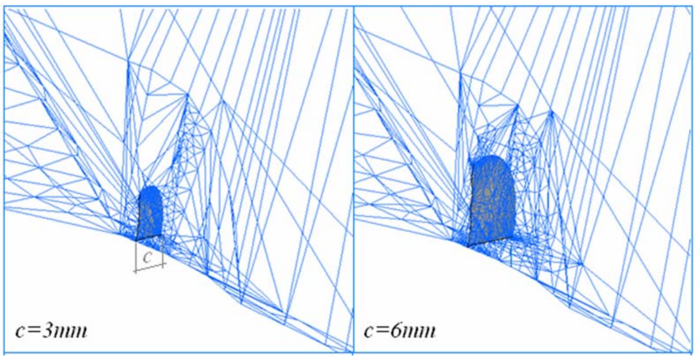


Figure 11: Progressive crack dimension during the crack propagation phase

Figure 12 shows the behavior of the K_I (mean along crack front) versus the crack length; the critical crack length ”c” is equal to 6.7 mm.
Using NASGRO 4.11 [8] equation as the propagation model, the $K_I(c)$ curve was numerically integrated; about 4170 amplified start-stop load cycles was necessary to reach the critical crack dimension.

This result is in good agreement with the results obtained during the experimental test, in which a cycle interval of 3273 cycles was observed until failure.

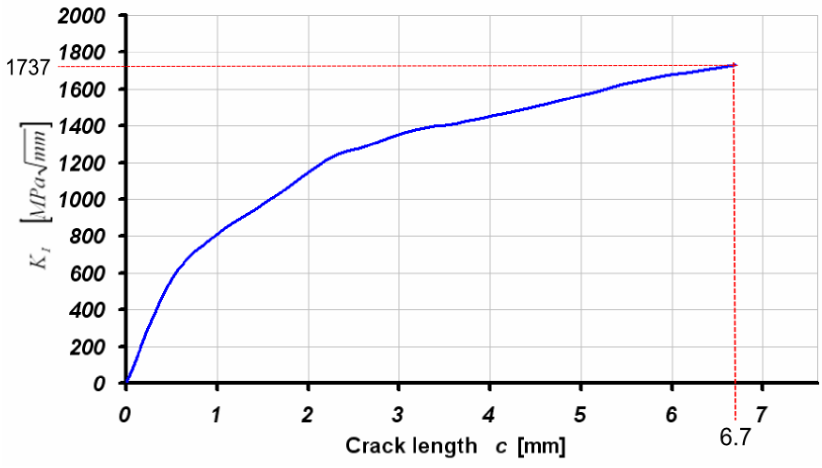


Figure 12: Stress intensity factor KI vs. crack length

In the case described above, the use of a validate numerical model can permit to reduce the full scale experimental test, mainly in the preliminary phase, obtaining a reliable indication about the most dangerous zone of the component and the crack propagation time until failure.

REFERENCES

[1] G. Scheneider, J. Wang, Crack Growth Analysis and Test of a Main Rotor Damper Attachment Lug, Annual forum proceedings - American Helicopter Society, 2004, vol 60, part 1, pp. 193-201.

[2] S. Singh, Crack Growth Life Analysis of a Main Rotor Idler Link, Annual forum proceedings - American Helicopter Society, 2005, vol 61, vol 1, pp. 659-665.

[3] L.E. Spievak, P.A. Wawrzynek, A.R. Ingraffea, D.G. Lewicki, Simulating fatigue crack growth in spiral bevel gears, Engineering Fracture Mechanics, Jan 2001, vol. 68, n. 1, pp. 53-76.

[4] A. Ural, G. Heber, P.A. Wawrzynek, A.R. Ingraffea, D.G. Lewicki, J.B. Neto, Three-dimensional, parallel, finite element simulation of fatigue crack growth in a spiral bevel pinion gear, Engineering Fracture Mechanics, May 2005, vol 72; n. 8, pp. 1148-1170.

[5] D. Colombo, M. Giglio, A. Manes, 3D fatigue crack propagation analysis of a helicopter component, International Journal of Materials and Product Technology, 2007, vol. 30, n. 1-3, pp.107-123.

[6] D. Colombo, M. Giglio, A methodology for automatic crack propagation modelling in planar and shell FE models, Engineering Fracture Mechanics, March 2006, vol. 73, n.4, pp. 490-504.

[7] Cornell Fracture Group, FRANC3D Version 2.6 Concepts and Users Guide, http://www.cfg.cornell.edu.

[8] NASGRO 4.11, "*Fracture Mechanics and Fatigue Crack Growth Analysis Software*", Reference Manual, NASA Johnson Space Center, 2004.