CRACK PROPAGATION STARTING AT HOLE'S EDGE GROWING TO PLATE'S MAJOR DIMENSION USING FEM

Bruna Luiza Nolli*, Ana Thais Almeida de Melo*, Carlos Alberto Cimini Júnior*
***Universidade Federal de Minas Gerais**

Keywords: *geometry factor, stress intensity factor, crack propagation, NASGRO, finite element analysis.*

Abstract

Holes edges are the preferential way for fatigue crack propagation. NASGRO® software doesn't predict the crack propagation if the dimension between the edge and the center of hole is bigger than half of total width for a crack on full plate thickness that starts on hole. This paper's objective was to develop a FEM to evaluate the stress intensity factor (K) for this scenario.

1. Introduction

The wide use of damage tolerant structures in aircraft components made the study of fatigue crack propagation important for airplanes safety. To prevent fracture it is necessary to calculate how the crack size affects components strength, in order to determine the critical crack size, and the time this crack will take to grow from initial to critical size to calculate safe operation life [1].

Selecting the place where the flaw will appear is the first step of a damage tolerance analysis. Stress concentration regions, like holes or notches, are the preferential way for fatigue crack nucleation [2].

The theory of linear elastic fracture mechanics (LEFM) stands fatigue crack growth rate is a function of stress intensity factor (K). Consequently, in order to predict behavior of fatigue crack propagation it is necessary to determine this parameter as a function of applied load and crack geometry [3].

Literature presents theoretical solutions for the most common types of cracks and loading. For the more complexes geometries it can be used, for example, the finite element alternating method (FEAM), the boundary element method, the three-dimensional virtual crack closure technique [4]. It is also possible to use fracture mechanics and fatigue crack growth software, like NASGRO® [5], which are based in theoretical and experimental data.

NASGRO® performs different types of fatigue and fracture mechanics analysis, like fatigue crack growth lifetimes, stress intensity factors from a library of solutions, critical crack size at failure, threshold crack size for no growth, etc. It has an extensive scenario library with different types of loading and geometry, as a crack starting at a plate edge, or at the center of a plate with tension/compression and bending loading.

However there are some geometric limitations at its scenarios. Fig. 1 shows one example of NASGRO® scenarios, a through the thickness crack initiating from a hole's edge.

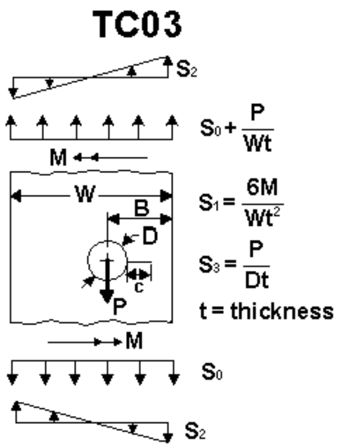

Fig. 1. Example of NASGRO® scenario.

For loading it can be input tension or compression (S_0), out-of-plane bending (S_1), in-plane bending (S_2), and bearing (S_3). The distance between hole's edge and center (B) must be less than or equal to half of total width (W). But if it is necessary to evaluate the crack propagation with B being greater than W/2 (crack propagating to the larger side of plate), NASGRO® cannot perform the calculation.

This project was developed with the objective to build a finite element model to evaluate the stress intensity factor (K) for this above-mentioned scenario, not covered by NASGRO®. The model was used to obtain the geometry factor (β) and the gross nominal stress. It was used the software Hypermesh® [6,7] to preprocess the finite element model, software Hyperview® to post process, and software NASTRAN® [8,9] as solver.

2. Development

2.1. Background

LEFM studies the stress field around crack tip using theory of elasticity. It assumes that crack grows as the stress near its tip exceeds material fracture toughness. This theory requires the plastic zone near crack tip to be smaller than any crack length dimension, therefore the stress intensity factor (K) controls the plastic deformation near the crack tip.

Based on reference stress, which is a stress out of concentration influence zone (σ_{ref}), the geometry factor (β) and the initial crack size (a), the stress intensity factor (K) is calculated, as shown in Equation 1.

$$K = \beta\sigma_{ref}\sqrt{\pi a} \qquad (1)$$

As a increasing loads acts on a plate with thickness t and modulus of elasticity E, it storages strain energy (potential energy U) as a spring damper. If a crack starts to grow in this plate (Δa) it will occur a decrease in member stiffness and a strain energy release will follow (ΔU), as shown on Equation 2. [10]

$$\Delta U = \frac{K^2}{E*}\, t\, \Delta a \qquad (2)$$

With E* = E/(1-ν) for plane strain and E* = E for plane stress. ν is the poisson`s ratio.

The rate between potential energy decreases with the increase of crack area is known as strain energy rate ($\mathcal{G}$), and it is given by Equation 3. [11]

$$\mathcal{G} = -\frac{1}{t}\frac{\Delta U}{\Delta a} \qquad (3)$$

Rearranging Equation 1 to 3, it is possible to reach an expression for the geometry factor as can be seen in Equation 4.

$$\beta = \frac{\sqrt{\frac{E}{t}\frac{\Delta U}{\Delta a}}}{\sigma_{ref}\sqrt{\pi a}} \qquad (4)$$

2.2. Methodology

The first step of the present study consists of getting the curve for the geometry factor (β) and the stress intensity factor (K) from the software NASGRO®. In order to perform the task it was chosen to be used the module call NASSIF which calculates the parameters β_0, β_1, β_2 and β_3 for each crack size (a) based on the crack type and the problem geometry. With this parameters and the crack size it is possible to calculate K using Equation 1 and β using Equation 5.

$$\beta^* = \left[S_0.\beta_0 + S_1.\beta_1 + S_2.\beta_2 + S_3.\beta_3\right] \qquad (5)$$

The second step involves building a finite element model to reproduces the geometry and the crack growth simulated in NASGRO®. It was modeled a plate with a hole, having dimensions B and W that respect the software restriction (B<=W/2). Fig. 2 shows this model.

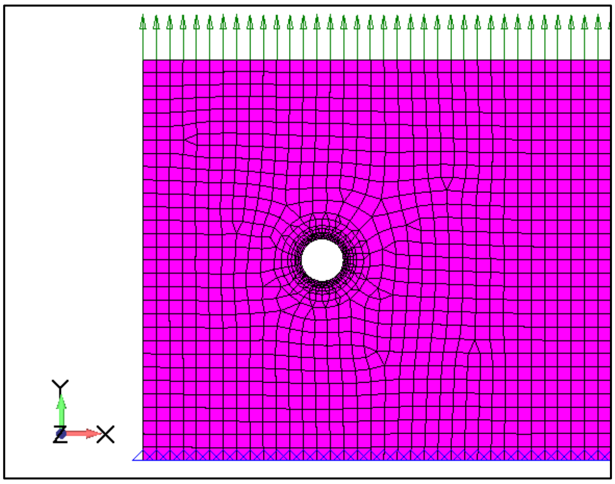

Fig. 2. Finite element model.

The crack and it's propagation were represented by the nodes separation on the model. The nodes were constrained through Multi-Point Constrain (MPC's) and each pair were released in one increment of the analysis, representing the crack propagation of MPC's type. Fig. 3 shows the simulation of crack growth.

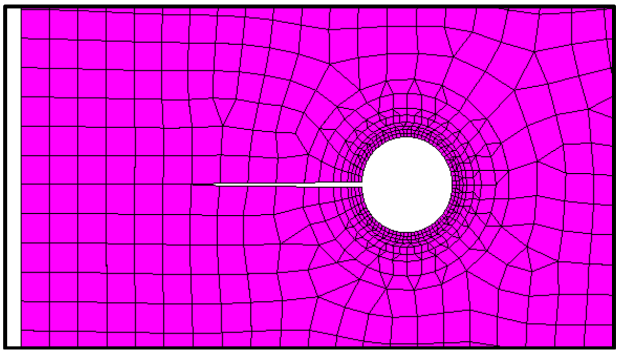

Fig. 3. Simulation of crack growth in finite element model.

In order to simulate a plate in tension, it was applied load in Y+ direction on the upper nodes of the plate, and the lower nodes were restricted in Y direction. Only one node far from the crack growth was fixed (to avoid rigid body movements at the model). The load and constrains can be seen in Fig. 4.

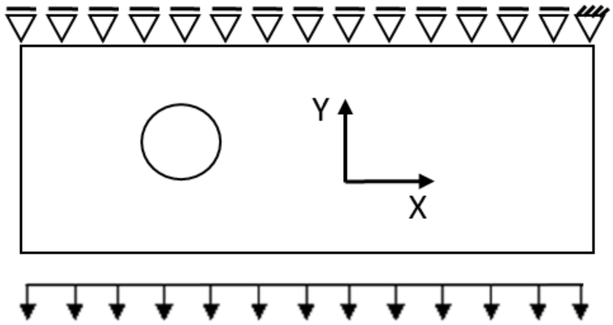

Fig. 4. Schematic of load and boundary condition for FEA.

The third step of the study was to run the model and get the total energy of deformation for each step of the analysis. The variation of this energy, the variation of crack size and material properties were used in Equation 4 to obtain the value of β. The stress intensity factor (K) was calculated using Equation 1.

The values of β and K were plotted against the crack size and a trend line was built. It was used polynomial fit with the target to reach the highest R^2.

Using the polynomial obtained, the values of geometry factor (β) and the stress intensity factor (K) were calculated using the same cracks sizes from NASGRO®'s curve. These values were compared with data from NASGRO®. If the error target was reached, it was possible to go to next step, otherwise it was necessary to refine the model and repeat step three.

Step four consisted in crack initial length to the largest dimension of the plate using the validated mesh, to build the scenario NASGRO® does not run. The same methodology of steps two and three was used in order to evaluate geometry factor (β) and the stress intensity factor (K) curves obtained in FEA model. The load was the same but there was difference in constrain: the fixed node was changed to be far from crack nucleation. Fig. 5 shows the new configuration.

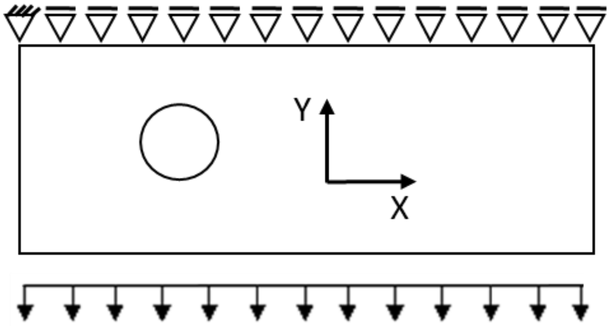

Fig. 5. Schematic of load and boundary condition for new FEA.

3. Results

The study was initiated by defining the geometry of the problem and the load applied. It can be seen the configuration in Fig. 6 and Table 1.

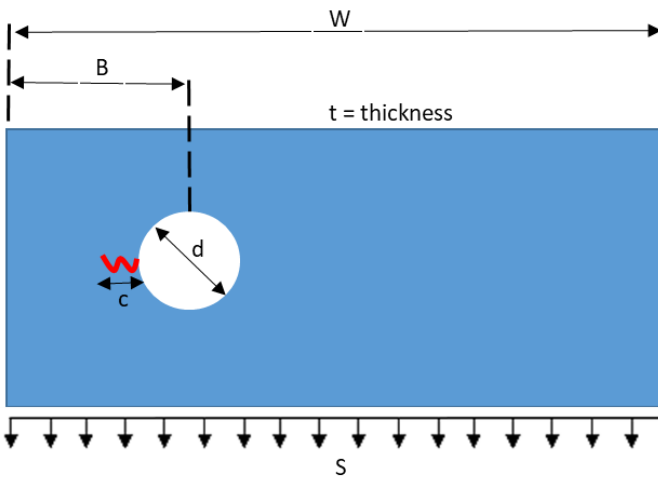

Fig. 6. Geometry and load chosen to build FEM.

Table 1. Input data for geometry and load.

W [mm]	B [mm]	d [mm]	t [mm]	S [daN/mm]
224.00	26.80	6.35	1.00	1.00

These data were used as inputs in NASGRO® in order to obtain the geometry factor (β) and the stress intensity factor (K) curves. Both curves are shown in Fig. 7 and Fig. 8.

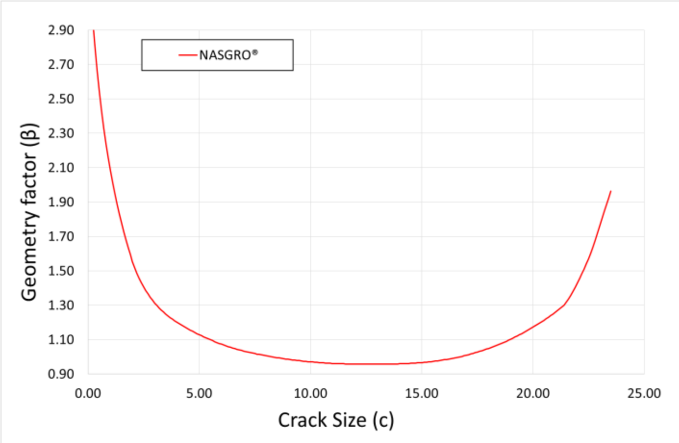

Fig. 7. NASGRO® geometry factor (β) curve.

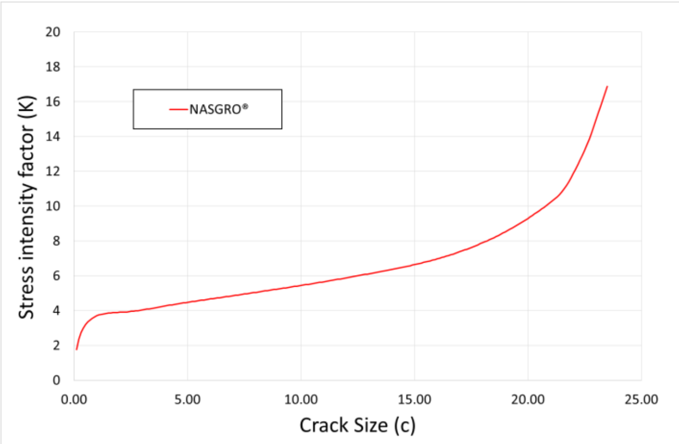

Fig. 8. NASGRO® stress intensity factor (K) curve.

Three finite element models were created with different mesh sizes: 2.0 mm, 1.0 mm and 0.5 mm. Fig. 9, Fig. 10 and Fig. 11 show the mesh around the hole and indicate the path for the crack propagation (gray line).

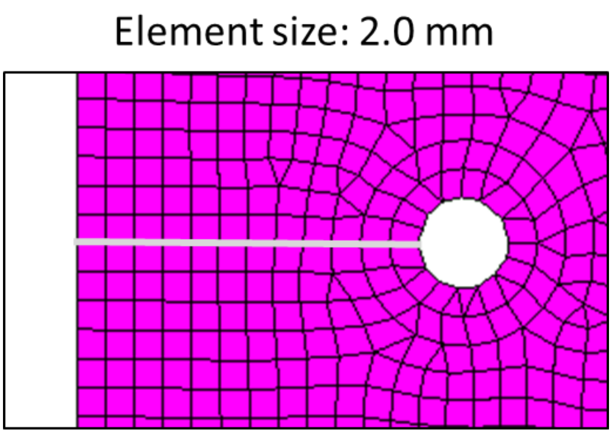

Fig. 9. Mesh for element size of 2.0mm.

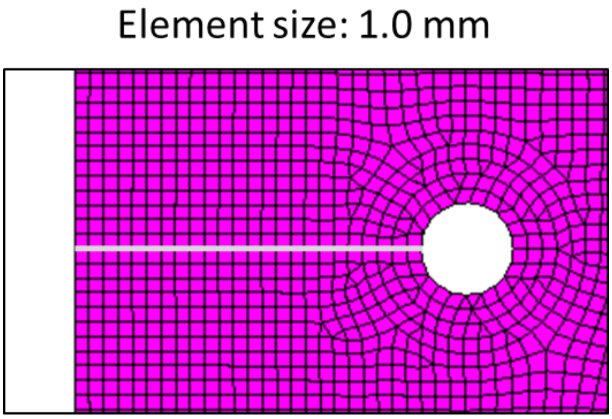

Fig. 10. Mesh for element size of 1.0mm.

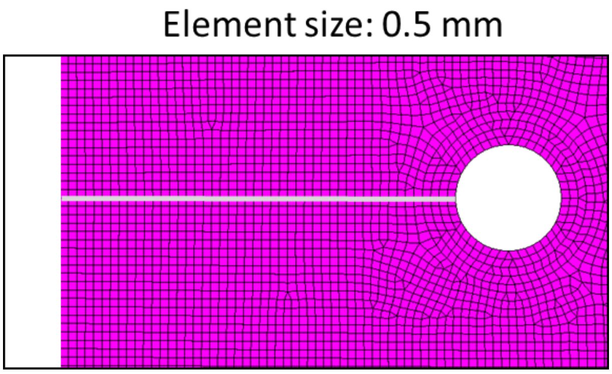

Fig. 11. Mesh for element size of 0.5mm.

From these models it was obtained the total energy of deformation for each step of the analysis. Equation 4 was used to get the value of β and the stress intensity factor was calculated using Equation 1. The results can be seen in Fig. 12 and Fig. 13.

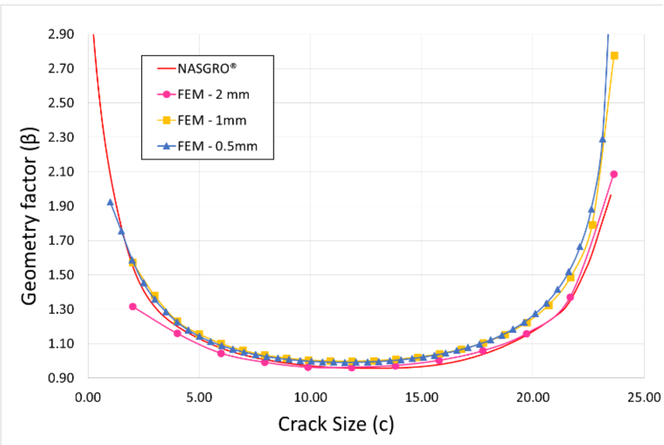

Fig. 12. Comparison between geometry factors (β) for different meshes.

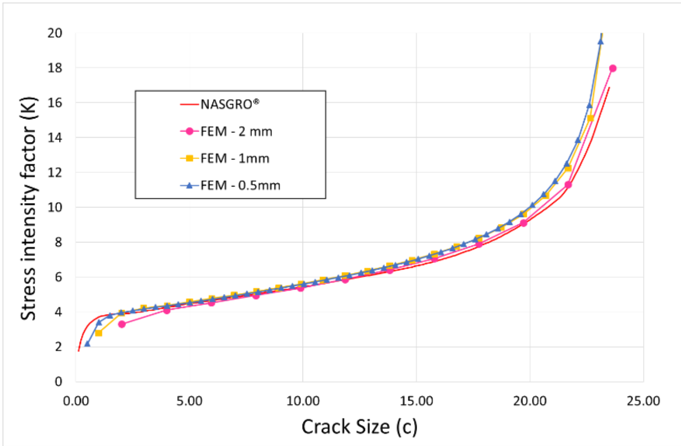

Fig. 13. Comparison between stress intensity factors (K) for different meshes.

From the analysis of these graphs it was concluded that cracks sizes from 0.1 mm to 7.0 mm have a better correlation with data from NASGRO® using element size of 0.5 mm. For cracks sizes greater than 7.0 mm, element size of 2.0 mm had a better correlation. Therefore more three models were built using element size varying from 0.5 mm to 2.0 mm, 0.25 mm to 2.0 mm and 0.1 mm to 2.0 mm. Fig. 14, Fig. 15 and Fig. 16 show the meshes.

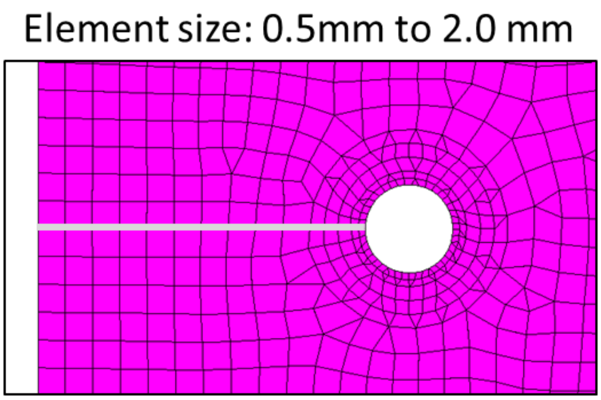

Fig. 14. Mesh for element size of 0.5 mm to 2.0 mm.

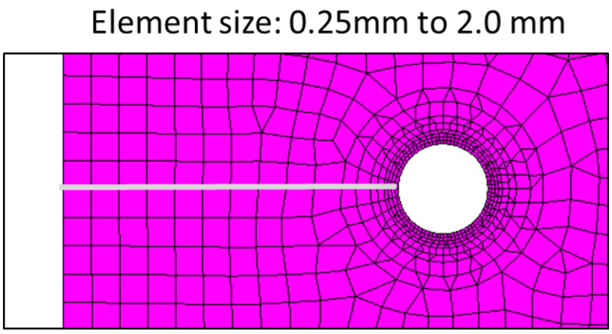

Fig. 15. Mesh for element size of 0.25 mm to 2.0 mm.

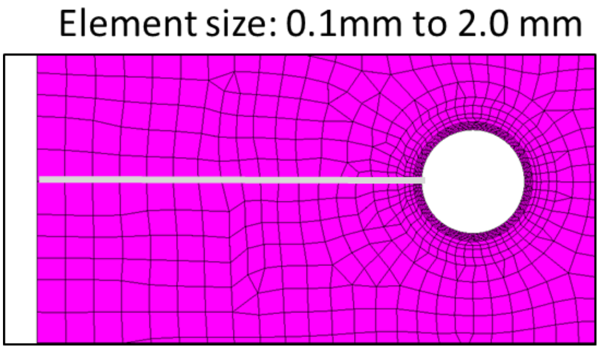

Fig. 16. Mesh for element size of 0.1 mm to 2.0 mm.

In the same way as before, Equation 4 and Equation 1 were used to obtain the value of β and K, and the results can be seen in Fig. 17 and Fig. 18.

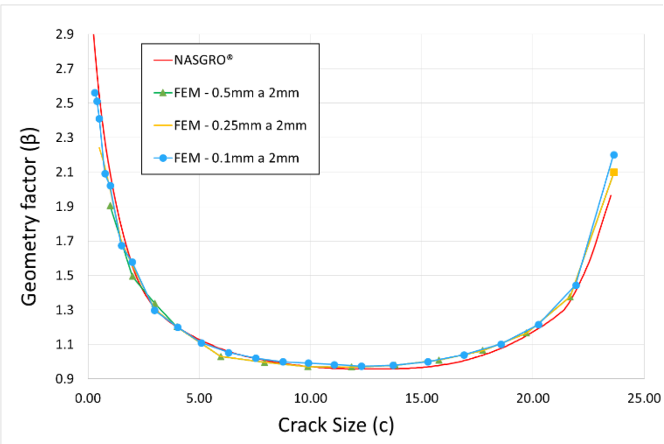

Fig. 17. Comparison between geometry factors (β) for different meshes.

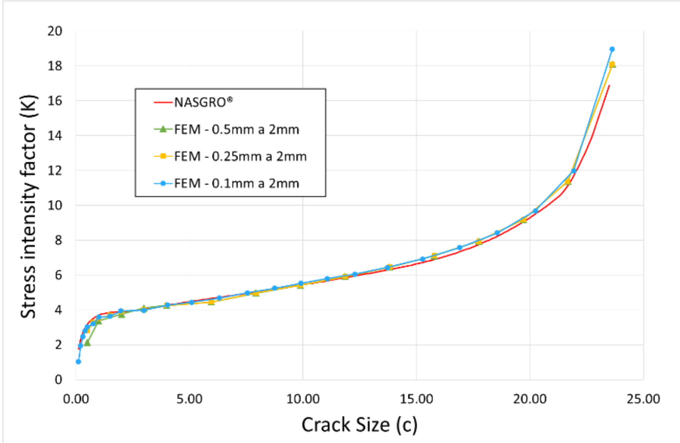

Fig. 18. Comparison between stress intensity factors (K) for different meshes.

A trend line was built for these three results for the geometry factor (β) and the stress intensity factor (K) curves. Two polynomials were created, one using data of crack size until 2.0 mm and the other using crack size greater than 2 mm. The polynomials obtained were used to calculate the values of β and K for a crack size starting at 0.1 mm, growing to 23.5 mm with an increment of 0.1 mm (the same used to build NASGRO® curves).

The best results were reached with the model having element size from 0.25 mm to 2.0 mm. The differences between the values of β and K got from the results of this polynomial and NASGRO® ones is shown in Fig. 19. This mesh was considered validated as its results reached an error below 5% for a crack size starting at 1 mm.

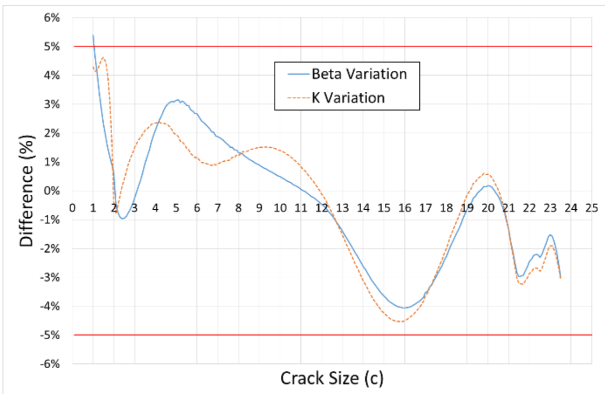

Fig. 19. Results differences between β and K taken from polynomial and NASGRO®.

The mesh of this model was used to propagate a crack to the other side of the plate, in order to simulate crack propagation for plate's largest side. Fig. 20 shows the mesh around the hole and indicate the path for the crack propagation (gray line).

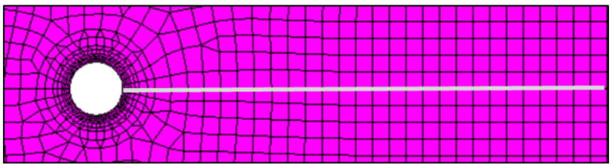

Fig. 20. Mesh for element size of 0.25 mm to 2.0 mm.

As performed before, it was obtained the total energy of deformation for each step of the analysis from this model. Equation 4 was used to get the value of β and the stress intensity factor was calculated using Equation 1. The results can be seen Fig. 21 and Fig. 22.

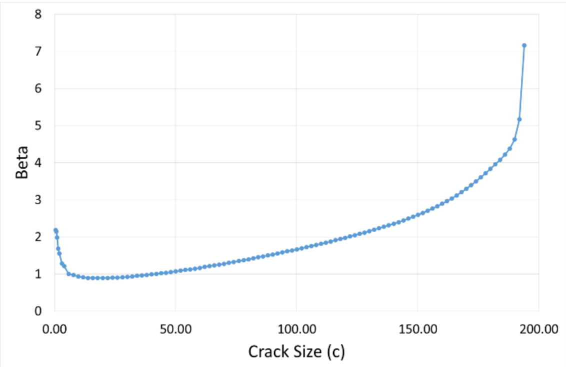

Fig. 21. Geometry factor (β) curve for a crack propagating to plate's largest side.

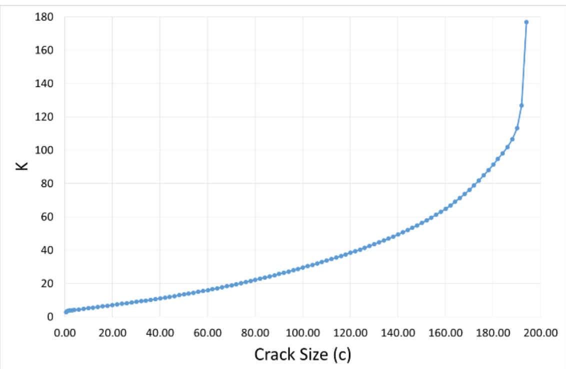

Fig. 22. Stress intensity factor (K) curve for a crack propagating to plate's largest side.

4. Conclusion

Three finite element models were created with different mesh sizes: 2 mm, 1 mm and 0.5 mm to represent a crack propagation initiating from the edge of a hole. Using the total energy of deformation for each step of the analysis is was built the geometry factor and the stress intensity factor curves. The results were compared with the curves obtained from NASGRO®, using NASSIF® modules.

It was observed that cracks sizes from 0.1 mm to 7.0 mm have a better correlation with data from NASGRO® using element size of 0.5 mm. For cracks sizes greater than 7.0 mm, element size of 2.0 mm had a better correlation. Therefore more three models were built using element size varying from 0.5 mm to 2.0 mm, 0.25 mm to 2.0 mm and 0.1 mm to 2.0 mm. And in the same way as before the curves for β and K were built.

The conclusion was that the model with mesh varying from 0.25 mm to 2.0 mm had the best correlation with NASGRO® results. So this mesh was used to propagate a crack to the largest side of the plate initiating from the hole's edge.

The geometry factor (β) and the stress intensity factor (K) curves obtained can now be used to perform calculations for crack propagation in this scenario using fracture mechanics.

Acknowledgements

The authors of this paper would like to acknowledge Embraer for the support given to develop this research.

References

[1] Broek D. *The practical use of fracture mechanics*. 1st edition, Kluwer Academic Publishers, 1988.

[2] Dowling N. *Mechanical Behavior of Materials*. 2nd edition, Prentice Hall College Div, 1993.

[3] Peng D, Jones R. A simple method for calculating the stress intensity factors for complex 3D cracks at a notch. *Engineering Fracture Mechanics*, 158: 81–86, 2016 (doi:10.1016/j.engfracmech.2016.02.042),<https://doi.org/10.1016/j.engfracmech.2016.02.042>

[4] Han Q, Wang Y, Yin Y, Wang D. *Determination of* stress intensity factor for mode I fatigue crack based on finite element analysis. *Engineering Fracture Mechanics*, 138, 118–126, 2015 (doi:10.1016/j.engfracmech.2015.02.019),<https://doi.org/10.1016/j.engfracmech.2015.02.019>

[5] NASGRO, *Fracture mechanics and fatigue crack growth analysis software* (version 8.01), reference manual, NASA Johnson Space Center and Southwest Research Institute; 2010.

[6] Altair HyperMesh 12.0, *Programmer's Reference Guide*, Reference Manual, 2012.

[7] Altair HyperMesh 12.0, *User's Guide*, Reference Manual, 2012.

[8] MSC Nastran, *Quick Reference Guide*, MSC Software, 2012.

[9] MSC Nastran, *Reference Manual*, MSC Software, 2012.

[10] Schijve J. *Fatigue of structures and materials*. 1st edition, Kluer Academic Publishers, 2004.

[11] Tada H, Paris P, Irwin G. *The stress analysis of cracks handbook*. 1st edition, Del Research Corporation, 1973.

5. Contact Author Email Address

Author: Bruna Luiza Nolli
Email address: nolli.bruna@gmail.com

Author: Ana Thais Almeida de Melo
Email address: melo.atm@gmail.com

Author: Carlos Alberto Cimini Júnior, Email address: carlos.cimini@gmail.com

Copyright Statement