FINITE ELEMENT MODEL DEVELOPMENT FOR SIMULTANEOUS CRACK PROPAGATION IN TWO COOPERATIVE LUGS

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

Keywords: *stress intensity factor, geometric factor, crack propagation, finite element model, lug*

Abstract

Multiple crack propagation scenarios are onerous to predict and calculate. Aircraft operation must be safe and damage tolerance design ensures residual resistance of structure by fracture mechanics analyses. There are commercial tools able to calculate simple scenarios but complex geometries are not covered by them, like simultaneous crack propagation in cooperative lugs. This scenario was studied using finite element method to obtain the geometric factor (β) and understand structural behavior.

1 Introduction

There are design methods used on aeronautic structures that must guarantee the absence of catastrophic failures during operating life. Safe life method orders replacement of components after an established life based on fatigue studies, and damage tolerance approach accepts the presence of a defect until a planned maintenance action can find and repair it, based on fracture mechanics that ensures residual resistance of structure [1].

Linear elastic fracture mechanics is used to prevent cracks to grow to a critical size, which usually nucleate in regions with stress concentration, like holes or notches [2]. One of factors that determine crack growth is the stress distribution on the crack tip, which depends of the stress intensity factor (K) that is presented on Eq. (1).

$$K = \beta \sigma_{ref} \sqrt{\pi a} \tag{1}$$

Where "a" is the crack length, "σ_{ref}" is the reference stress (value outside of stress concentration influence zone) and "β" is the geometric factor. The geometric factor (β) is a function that depends of geometry, failure mode and load path, and because of that it is a complex parameter to obtain [2].

A calculation method to obtain the stress intensity factor (K) is based on the strain energy release rate, presented in Eq. (2). In finite element models the crack length (a) is increased by uncoupling of nodes along the crack line. Experience has shown that an acceptable accuracy can be obtained without the necessity to go to an extremely fine mesh [3].

$$\frac{dU}{da} = \frac{K^2}{E} \tag{2}$$

Several damage scenarios are analyzed using fracture mechanics for aircraft development. There are commercial tools that can be used to calculate simple scenarios but complex geometries or multiple crack scenarios are not covered by them. That is the case of the component with two cooperative lugs (or clevis) with damage present in both lugs and influence of the crack in one lug on the crack of the other one.

A method to find the geometric factor (β) is obtained using Eq. (1) and Eq. (2) that results on Eq. (3).

$$\beta = \frac{\sqrt{\frac{1}{\pi a} E \frac{dU}{da}}}{\sigma_{ref}} \qquad (3)$$

To study the simultaneous crack propagation in different cooperative lugs, which share the same loading a finite element model was developed, using Hypermesh [4] as pre-processor, Hyperview [4] as post-processor and Nastran [5] as solver.

2 One lug model

The first step was simulating crack propagation in only one lug. As the main goal is evaluate a method for obtain the geometric factor (β), lug's dimension are smaller than the usually adopted on service. They are presented in Figure 1 and Table 1.

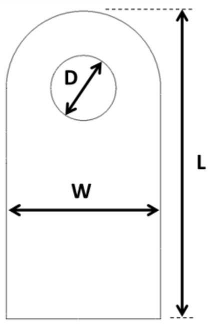

Figure 1: Geometry

Table 1: Dimensions

D	6.35 mm
W	15 mm
L	30 mm

Boundary conditions of finite element model are presented in Figure 2. Table 2 presents thickness of model, material properties (from a general aluminum), considered elastic and linear. Lug was modeled with shell elements, its base was restricted on the six degrees of freedom and the unidirectional load (P = 500 daN) was applied with a rigid element that represents a pin with 130° of contact, to avoid influence on nodes near to crack representation.

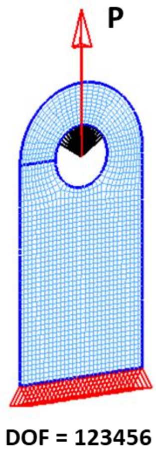

Figure 2: Boundary conditions – one lug model

Table 2: Model properties

Thickness (t)	2.4mm
E	7200 daN/mm²
ν	0.33

The initial crack and its propagation were represented by nodes separation on the model, on line mesh highlighted on Figure 2. The nodes were constrained through Multi-Point Constrain (MPC) and each pair was unrestricted in one increment of the analysis.

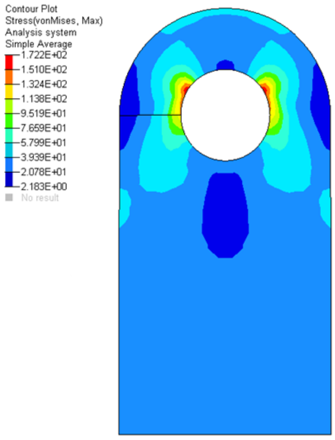

Figure 3: Finite element model, Von Mises stress, one lug: closed

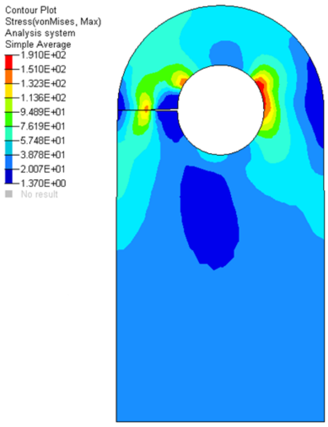

Figure 4: Finite element model, Von Mises stress, one lug: crack of 2.21 mm

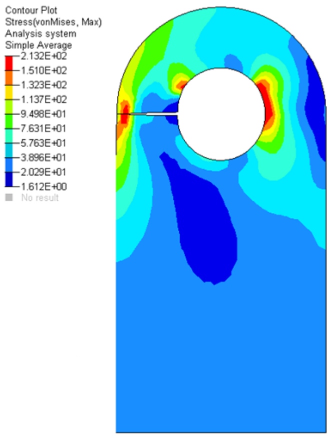

Figure 5: Finite element model, Von Mises stress, one lug: crack of 3.67 mm

From this model, the outputs were strain energy (U) and crack length (measured from the disconnected elements). Reference stress is presented in Eq. (4). Using these values, curve for the geometric factor was built using Eq. (3). It was compared with Nassif [6] curve, using same input data. Results are presented on Figure 6.

$$\sigma_{ref} = \frac{P}{Dt} \qquad (4)$$

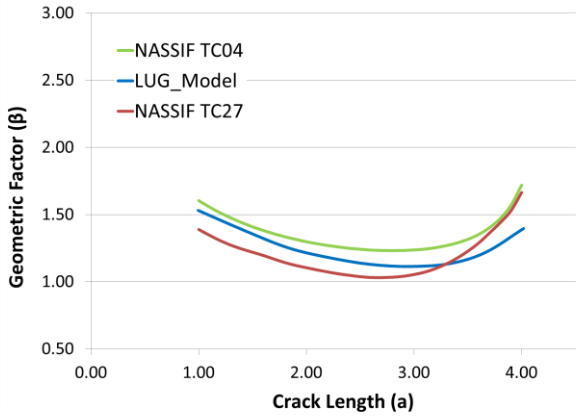

Figure 6: Geometric factor, one lug model

The result of model has the same behavior of results from Nassif [6]. Nominally, the new solution TC27 has the same geometry and loading condition as TC04, but TC27 employs nonlinear stress variation and contact between the pin and the lug, so presents different results [6]. The difference between the curves, besides the module, is the moment that the derivate changes.

3 Two lugs model

The mesh presented on Figure 2 was duplicated and boundary conditions were adapted for the model of two cooperative lugs, presented on Figure 7. The model is still linear elastic, its base is restricted on the six degrees of freedom and the unidirectional load (P = 1000 daN) was applied with a rigid element that represents a pin.

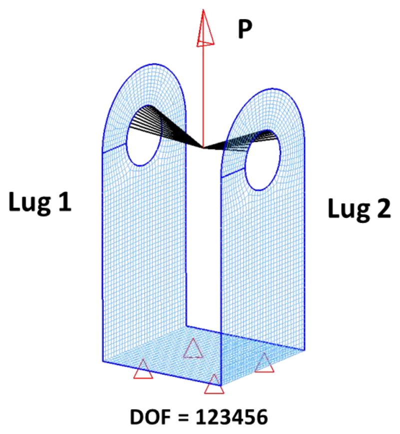

Figure 7: Boundary conditions – two lugs model

Again, initial crack and its propagation were represented by nodes separation on the model, but now with two cracks simultaneously, one in each lug, on lines highlighted on Figure 7. Thereby it was possible to verify component's behavior and the influence on the geometric factor (β).

Several simulations were done with different scenarios, in which the crack is always propagating in one lug and in the other the condition varied according to Table 3.

Table 3: Scenarios for two lugs model

Lug 1	Lug 2	Figure
	Open	Figure 8
	Closed, without crack	Figure 9
	Crack growth on increments	Figure 10
	Fixed crack of 0.54 mm	-
	Fixed crack of 0.99 mm	-
Crack growth on increments	Fixed crack of 1.41 mm	Figure 11
	Fixed crack of 2.21 mm	-
	Fixed crack of 2.58 mm	-
	Fixed crack of 2.96 mm	-
	Fixed crack of 3.34 mm	-
	Fixed crack of 3.67 mm	-
	Fixed crack of 4.02 mm	-

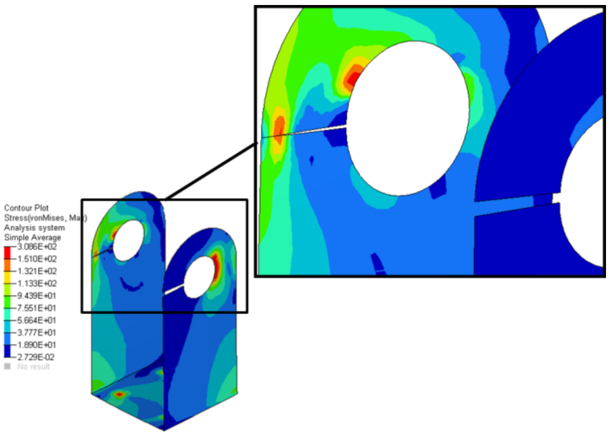

Figure 8: Finite element model, Von Mises stress, two lugs – Lug 1: crack of 3.34 mm, Lug 2: open

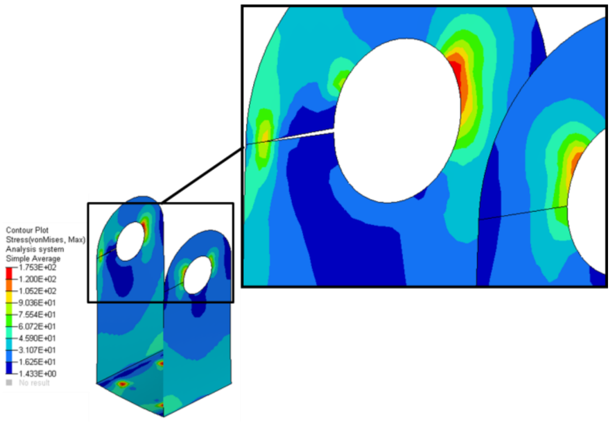

Figure 9: Finite element model, Von Mises stress, two lugs – Lug 1: crack of 3.34 mm, Lug 2: closed

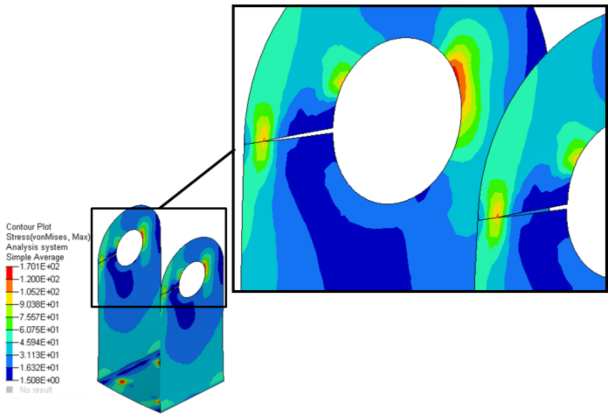

Figure 10: Finite element model, Von Mises stress, two lugs – Lug 1: crack of 3.34 mm, Lug 2: crack of 3.34 mm

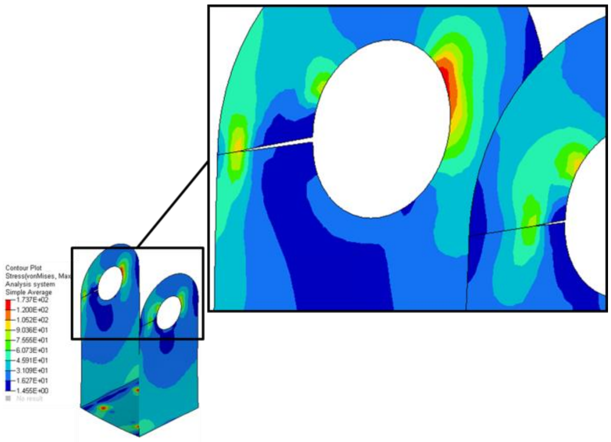

Figure 11: Finite element model, Von Mises stress, two lugs – Lug 1: crack of 3.34 mm, Lug 2: fixed crack of 1.41 mm

The curves for geometric factor obtained from those models are presented in Figure 12 and Figure 13.

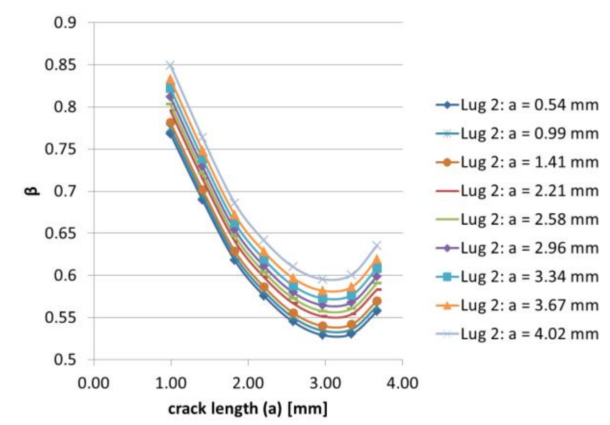

Figure 12: Geometric factor, two lugs model – Lug 1: crack growth on increments, Lug 2: fixed crack

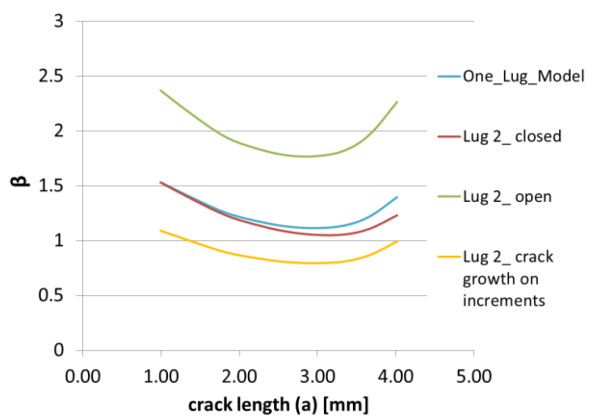

Figure 13: Geometric factor, two lugs model – Lug 1: crack growth on increments, Lug 2: variable condition

4 Conclusions

The "one lug model" geometric factor presented the same behavior of geometric factors from Nassif [6]. If mesh refinement was done, the values could match to TC04 results, but it is not true on TC27 case, because "one lug model" has linear behavior and do not represent contact between lug and pin.

The "two lugs model" was evaluated in many situations, all of them with crack propagation on Lug 1, considering the same reference stress and variable condition for Lug 2. Geometric factor curves obtained from "one lug model" and "two lugs model – lug 2 closed" would match, but in the end of curve the model concentration had influenced.

This method based on the strain energy release rate using finite element models could generate geometric factors for a large number of scenarios, with a simple approach and an acceptable accuracy [3], as seen in this study of lug in comparison with a commercial tool.

For the case of two cooperative lugs the conclusion is that the lugs can be evaluated separately. The influence of one crack in the other one could be neglected since the reference stress considers the redistribution of the load, because the shape of curves was not affected so a factor on reference stress is enough to guarantee the integrity of structure.

Acknowledgements

The support for this work by Embraer is gratefully acknowledged.

References

[1] Lazzeri, R. A comparison between safe life, damage tolerance and probabilistic approaches to aircraft structure fatigue design. *Aerotecnica Missili e Spazio*, Vol. 81, 2-2002, 2002.

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

[3] Schijve, J. *Fatigue of structures and materials*. 1st edition, Kluwer Academic, 2001.

[4] Altair Engineering, Inc. *Hyperworks reference manual.* 2018

[5] MSC Software. *Nastran reference manual.* 2018.

[6] Southwest Research Institute. *Nasgro reference manual.* 2018.

Contact Author Email Address

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

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

Author: Carlos Alberto Cimini Júnior
Email address: carlos.cimini@gmail.com
Mailling address: Av. Antônio Carlos, 6627 - Escola de Engenharia - Bloco 1 - 4o andar, sala 4215, Pampulha, Belo Horizonte – MG – Brazil.

Copyright Statement