Available online at www.sciencedirect.com

ScienceDirect

Procedia Structural Integrity 5 (2017) 225–232

Structural Integrity Procedia

www.elsevier.com/locate/procedia

2nd International Conference on Structural Integrity, ICSI 2017, 4-7 September 2017, Funchal, Madeira, Portugal

FE Mesh Generation – Automated Crack Grow Modeling with a View to Stress Intensity Factor Computing

Jan Raška[a],*

[a]*Aeronautical Research and Test Institute (VZLÚ),Strength of Structures Dept., Beranových 130, Praha – Letňany, 199 05, Czech Republic*

Abstract

The damage tolerance design approach, expects the existence of the initial flaw in any structure. To manage the extension of this flaw during service, the fracture mechanics principles are applied. For this purpose, two domains of analysis must be accomplished: First, the crack growth due to variable, periodical load (fatigue load), second, the residual strength of the damaged structure statically loaded. For both analysis, the computing of the fracture mechanics characteristics in each state of the crack opening is indispensable.

In practice, the application of the FE method is necessary. By the help of currently used FEM software NASTRAN, the stress intensity factor – K_I and K_{II} – can be compute by application of the special element, called CRAC2D. Unfortunately, CRAC2D element is not implemented into the currently used FE pre-processors. Moreover, one application of the CRAC2D element result only in a single value of the K_I and K_{II} for the modeled crack length.

For these reasons, the fully automated crack growing modeling was developed. Based on the virgin FE model (base FE model without crack), the developed software application generate the appropriate FE model for each desired crack opening. User have only to indicate the crack initiation node and the crack growing path, so-called crack growing scenario (see fig.1). After the data check, the crack is modeled, the CRAC2D element is applied at the crack tip, the FE model is saved, the related job is executed and the resultant K_I and K_{II} values are associated with modeled crack length. This step is repeated for each desired crack opening.

Peer-review under responsibility of the Scientific Committee of ICSI 2017

Keywords: Crack Growth, Crack Path, FE Model, FEM, Fracture Mechanics, Stress Intensity Factor

* Corresponding author. Tel.: +420-225-115-459; fax: +420-225-115-335.
E-mail address: raska@vzlu.cz

2452-3216
Peer-review under responsibility of the Scientific Committee of ICSI 2017
10.1016/j.prostr.2017.07.121

Nomenclature

K_I	Stress intensity factor – mode 1
K_{II}	Stress intensity factor – mode 2
a	Crack length
CRAC2D	Special fracture mechanics NASTRAN element

1. Introduction

The approach to engineering design to account for damage tolerance, which is a property of a structure relating to its ability to sustain defects safely until repair can be effected, is based on the assumption that flaws can exist in any structure and such flaws propagate with usage [1]. This approach is commonly used in aerospace engineering to manage the extension of cracks in structure through the application of the principles of fracture mechanics[2]. For this purpose, two domains of analysis must be accomplished: First, the crack growth due to variable, periodical load (fatigue load), second, the residual strength of the damaged structure statically loaded. For both analysis, the computing of the fracture mechanic characteristics (as stress intensity factor [5], [8] or J-integral) in each state of the crack opening is indispensable.

In practice, for the real complex aeronautical structures, the application of the finite element method is necessary [4]. By the help of the currently used software NASTRAN, the fracture mechanics characteristics of the plate and shell structures can be compute by application of the special element, called CRAC2D [6]. Concretely, the first and second mode of the stress intensity factor – K_I and K_{II} – result this FE analysis. Unfortunately, CRAC2D element is not implemented into the currently used FE pre-processors. Moreover, one application of the CRAC2D element result in a single value of the K_I and K_{II} for the modeled crack length. However, for above-mentioned analysis is indispensable to compute the stress intensity factor in each crack growing state (for each crack length) [9].

For these reasons, the fully automated crack growing modeling was developed. Based on the virgin FE model (base FE model without crack), the developed software application generate the appropriate FE model for each desired crack opening. After, the job is executed for each crack opening and finally, the result computed values of stress intensity factor are automatically read and link with the modeled crack length. Also, the stress intensity factor as a function of crack length is determinate (see example on fig.1).

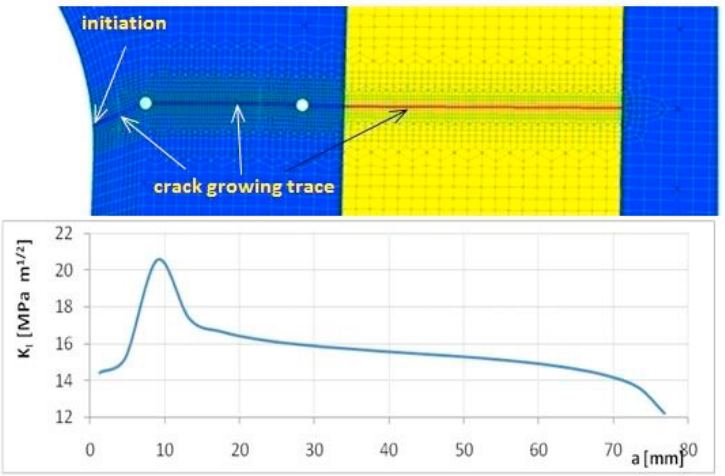

Fig. 1. Stress intensity factor vs. crack length – crack grow modeling example [7]