Engineering Fracture Mechanics 148 (2015) 42–59

Contents lists available at ScienceDirect

Engineering Fracture Mechanics

journal homepage: www.elsevier.com/locate/engfracmech

# Development of a varying mesh scheme for finite element crack closure analysis

Chien-Yuan Hou

*Department of Civil Engineering, National Chi Nan University, Puli, Taiwan, ROC*

ARTICLE INFO

*Article history:*
Received 22 October 2014
Received in revised form 22 July 2015
Accepted 3 September 2015
Available online 9 September 2015

*Keywords:*
Finite element analysis
Crack closure
Plasticity
Fatigue crack growth

ABSTRACT

Large amount of computation time is inevitable for finite element crack closure analysis when the crack propagates a long distance. This is because numerous elements are needed along the crack path. A method using a varying mesh technique is proposed to reduce the number of elements. In the method, a new finite element model is constructed before crack tip release, thus, compared to the conventional scheme, two additional tasks are required for the new approach: remeshing and state variable mapping. The proposed technique using a simple variable mapping produces unacceptable results for the plane strain condition. Investigations show that the residual nodal forces resulted from the mapped stresses cause the difficulty. A new remeshing method is proposed to overcome the difficulty. Also, significant computation time saving by the new technique is confirmed.


## 1. Introduction

Closure effects on the growth behavior of fatigue cracks have been studied for several decades. Since Newman [1] developed a finite element technique to calculate the closure effects caused by plasticity, numerous researches have been conducted to study the factors affecting closure behavior using the technique. These factors include loading level [1–3], stress ratio [1–5], plane stress and strain conditions [1,3–11], criteria for opening stress determination [10,12,13], notch plasticity [11,14], overload and variable amplitude load [15–18], material law [9,11,19], and 3D cracks [20–24] and their shape development [23,25,26]. In addition, the important effects of mesh size have also been studied [8,15,27,28]. The technique mainly contains the following steps: constructing the meshes of the cracked domain, loading and unloading the model step by step, releasing the crack tip node to simulate crack growth, monitoring the crack face node displacement, or contact forces, or the internal force of the elastic spring elements attached to the crack face nodes, so that the crack opening level can be determined.

Numerous results have shown that the calculated crack opening stress is significantly affected by the number of elements embedded in the crack tip plastic zone (elements in plastic zone, EIPZ). McClung and Sehitoglu [2] suggested using 10 elements in the forward plastic zone and Solanki et al. [27] suggested 3–4 elements in the reversed plastic zone. This size requirement (EIPZ requirement) has been adopted in many other studies to determine the element size ahead of the crack tip. Since the crack tip plastic zone size is usually small for metal fatigue, this requirement limits the element size near the crack tip. Thus, to fulfill the requirement and to reduce the model size to save computation time, during the mesh construction stage a fine mesh region is usually deployed near the crack tip and coarse meshes are used in the region away from crack tip: see Fig. 1 (note that the figure is simply a schematic illustration and the number of fine elements is not plotted in

*E-mail address:* cyhou@ms15.hinet.net

http://dx.doi.org/10.1016/j.engfracmech.2015.09.009
0013-7944/

**Nomenclature**

| | |
|---|---|
| EIPZ | elements in plastic zone |
| $N_{f,\min}$ | minimum required number of elements ahead of initial crack tip |
| $N_{\mathrm{EIPZ}}$ | number of elements for EIPZ requirement |
| $N_{LC}$ | number of loading cycles planned to be applied in the analysis |
| VM | varying mesh |
| CNV | conventional |
| $\Delta a$ | crack increment of each loading cycle |
| $u_x$, $u_y$ | nodal displacements |
| $h$ | shape function |
| $\xi$, $\eta$ | isoparametric coordinates |
| $\sigma_x$, $\sigma_y$, $\sigma_{xy}$, $\sigma_z$ | stress components |
| $\sigma_o$ | yielding stress |
| $\bar{\varepsilon}_p$ | effective plastic strain |
| $S_{\max}$ | maximum remote stress |
| $R$ | stress ratio |
| $\Delta S$ | stress increment of each loading step |
| $S_{\mathrm{open}}$ | crack opening stress |
| $\Omega$ | element domain |
| $[K]$ | stiffness matrix |
| $[f]$, $[f_I]$ | external and internal nodal force vectors |
| $[u]$ | nodal displacement vector |
| $[B]$ | strain–displacement matrix |
| $[\sigma]$ | stress vector |
| $[r]$ | residual nodal force vector |

accordance with the EIPZ requirement). When the analysis is performed, the crack tip node is released in a loading cycle to simulate crack growth. This procedure enforces the crack advancing a distance of fine element size for each cycle. As the crack tip moves forward, the EIPZ requirement is still needed to be fulfilled. Thus, the minimum required number of fine elements ahead of initial crack tip ($N_{f,\min}$), using the required number in the forward plastic zone as an example, should be the sum of the number of elements for EIPZ requirement ($N_{\mathrm{EIPZ}}$) and the number of loading cycles planned to be applied in the analysis ($N_{LC}$, also see Fig. 1):

$$N_{f,\min} = N_{\mathrm{EIPZ}} + N_{LC} = 10 + N_{LC} \tag{1}$$

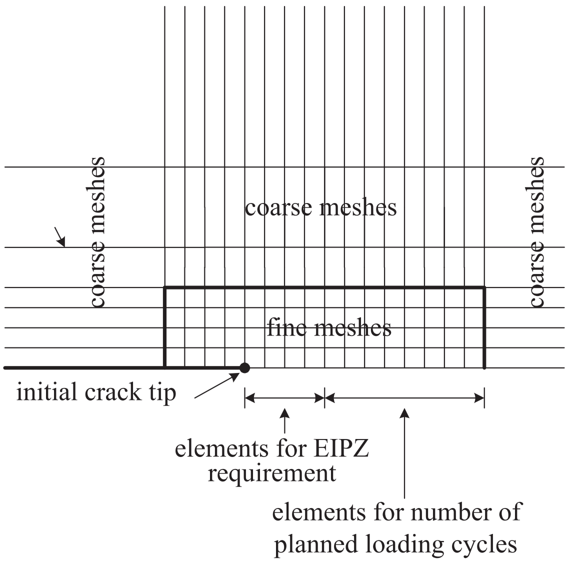


**Fig. 1.** Mesh arrangement of finite element closure analysis.