Elasto-Plastic stress singularity field near the vertex of bi-material joint

Jirapat Malai[1] and Attaporn Wisessint[2,*]

[1] Department of Mechanical Engineering, Graduate School, Kasetsart University, Ladyao, Jatuchak, Bangkok, 10900
[2] Department of Mechanical Engineering, Engineering Faculty, Kasetsart University, Ladyao, Jatuchak, Bangkok, 10900
*Corresponding Author: fengapw@ku.ac.th, 02-942-8555 ext. 1861

Abstract

Bi-materials joints are being used in various engineering applications. In a bi-materials joint, a very high stress can occur at the corner of the joint even for low mechanical loadings, due to the differences of Young's modulus of each material which can lead to a structural failure. In this study, the finite element method (FEM) was used to determine stresses and displacement of the joints, and it was found that after low mechanical loading were applied to the bi-materials joints, the plastic deformation occurred. Consequently, to determine the behavior of elasto-plastic stress singularity field around the singular point of the bi-material joint, the ratio of different Young's modulus and the plastic property in stress-strain curve for the lower Young's modulus material were varied.

***Keywords**:* Elasto-Plastic materials joint, Stress singularity field, Finite Element Method

1. Introduction

In linear elasticity, stress solutions involving singular stresses have been investigated by many researchers. In contrast to elasto-plastic material, the bi-material joint were few to investigate. From previous studies, the stress singularity may occurs at the vertex of the bi-material joint due to the difference of dissimilar materials properties [1] . These difference can leads to ineffective of reliability for the bi-material joint, while some researchs clarify that stress singularity does not exist in the real materials. The materials are subjected to the plastic flow after reached the yield stress, causing the stress to increase rapidly and reaches the finite value [2],[3] and maybe disappear with various reasons. In bi-material joint contact, one can be assumed that the hard material will behaves as elastic response, only soft material will behaves as elasto-plastic material because the plastic strain in soft material can be considered to be very large [4]. In order to investigate the stress singularity, refined mesh with small element size is needed in order to collect the steep stress near the corner vertex of the bi-material. Many researchers have been investigating new method for large increment in order to decrease the mesh count [5]. There're many elasto-plastic materials models that have been used in various studies, in this study plastic-material was assumed to be in the form of rate-independent power law strain hardening materials [6]. The overall aim is to study the structure of the bi-material-joint stress field near the corner in two-dimensional stresses.

columns with single spacing and the following margins.

In a bi-material joint there are many factor which influence the crack initiations. Constraints in fracture mechanics is a term that widely used yet few has been investigated in bi-material joint problems, the near-tip stress state or stress distribution ahead crack-tip, as the factor or conditions which influence the crack driving force. These would lead to understand for the characteristics of the stress fields around a singular point on the stress singularity for elasto-plastic biomaterial joint problems. The effected of material response from nearly linear-elastic to elasto-perfect plastic material also the influence from the ratio of Young's modulus between two materials.

2. Method and range of investigation

The considered problems have been investigated by the finite element method, using the MSC.Marc/Mentat FEM Application developed by MSC.Software Corporation. The program can solve non-linear 2-D problems with elasto-plastic material response. This paper is focused and organized as follows: the studies of stress distribution ahead corner of the joint when plastic material variable "n-exponent" were varied and when vary Young's modulus ratio between two materials. The simulation setup value is shown in Table 1 and Table 2 for both studies cases.

2.1 Elastic and elasto-plastic material property

Considering the plastic strain near the vertex in lower young's modulus material has much higher plastic strain value than higher one. Therefore, it is reasonable to assume the material with higher young's modulus which is behaves as a very "hard" material to be elastic material.

In an elastic material, Young's modulus will be varied to investigate the influence of each variable. For elasto-plastic material, plastic deformation occurs if the yield strength of the material is reached.

The stress-strain relation of the power law hardening material can be expressed as

$$\sigma_f = \sigma_0 \left(1 + \frac{\bar{\epsilon}^p}{\epsilon_0} \right)^n \qquad (1)$$

Where σ_f is the plastic flow stress, σ_0 is the yield stress, $\bar{\epsilon}^p$ is the equivalent plastic strain, ϵ_0 is the yield strain, n is the plastic strain hardening exponent, and also shown in Fig. 1. These curves shown the behavior of elasto-plastic materials ranging from nearly elasto-perfectly plastic material to more linear elastic material. The material properties used in this study are given in table 1. The yield behavior is indicated by Von Misses's Stress yield criteria.

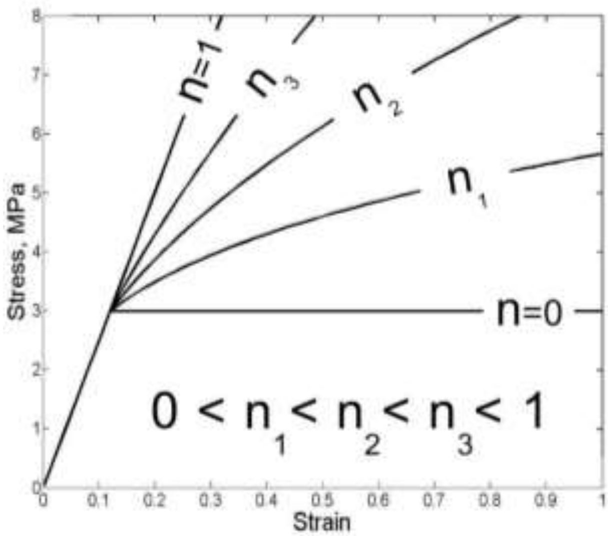

Fig. 1 stress-strain relationship

Table 1 Material properties used in *E* ratio comparison case

Type	Young's modulus (MPa)	Material response	Yield strength (MPa)	Poisson's ratio
Hard	250,300, 350,400, 500	Elastic	-	0.3
Soft	25	Elasto-Plastic (n=0.3)	3	0.3

Table 2 Material properties used in n-exponent comparison case

Type	Young's modulus (MPa)	Material response	Yield Strength (MPa)	Poisson's ratio
Hard	250	Elastic	-	0.3
Soft	25	Elasto-Plastic (n=0,0.2, 0.4,0.6, 0.8)	3	0.3

3.2 Finite element method

Finite element method (FEM) was used in this study to determine the approximate solution where the displacement formulation for an e nodes element is expressed as the following equation.

$$u_i = \sum_{n=1}^{e} N_n \bar{u}_{i\,n} \qquad (2)$$

Where u_i is the displacement in direction, $\bar{u}_{\text{in}}$ is the displacement at node, n associated with direction i. N_n is the interpolation function at node n.

The model used in this study is two-dimensional model with 2-D Quadrilateral Element; Quad 4 in MSC.Marc. This element class consists of four-noded isoparametric elements with bilinear interpolation. The element is formed by mapping from the x-y plane (Global Coordinates) to the ξ, η plane (Local Coordinates) see Fig. 2. There are 2 degrees of freedom for each node which is u_x (displacement in the global x direction) and u_y (displacement in the global y direction).

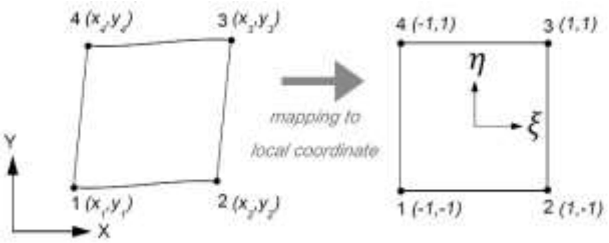

Fig. 2 Mapping global coordinate to local coordinate

3.3 Finite element analysis Model and boundary conditions

The finite element model of the problems is shown in Fig. 3. The element is two-dimensional and considering plane stress. A very fine mesh zone indicated in the Fig. 3 was used close to interesting area near the corner. These area will leads to stress singularity when bi-materials subjected to even small load.

The main reason why mesh needed to be refine was to capture the steep stress gradient expected at that region as shown in Fig. 4. The smallest elements in the finite element model, for this study had a 0.2648×10^{-2} nm radius.

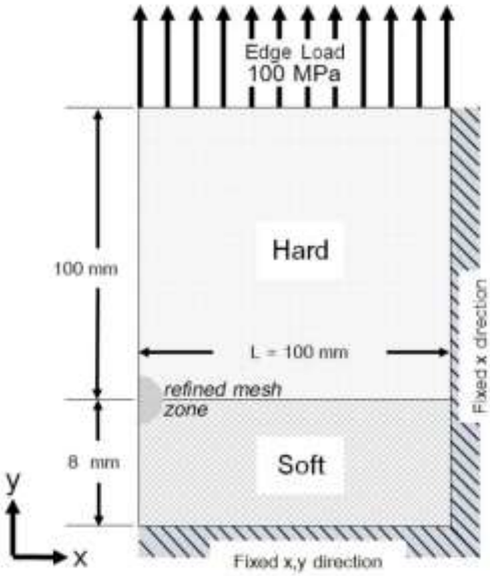

Fig. 3 Finite Element Model in this study with refined mesh zone at the corner of the bi-material joint.

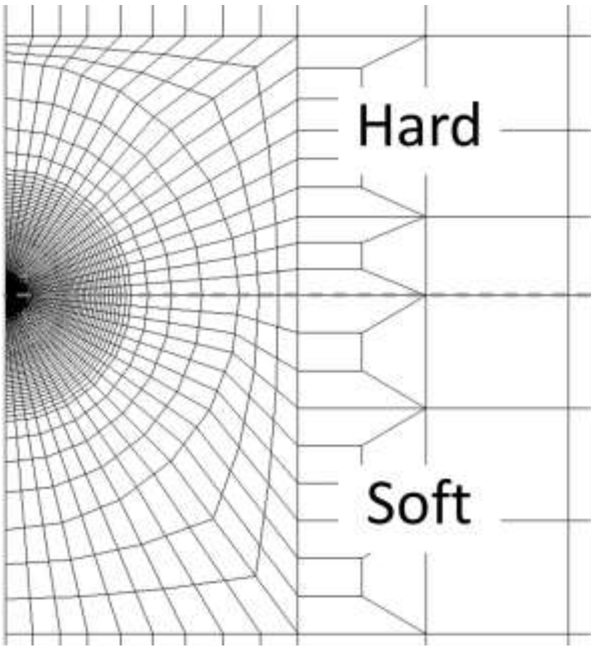

Fig. 4 refined mesh zone near the corner of the joint

4. Results and discussions

Fig. 5 shows the solutions for the stress distribution ahead the corner vertex of bi-material with different elasto-plastic material response. It can be found that the singularity in plastic zone becomes larger and stress distribution near the corner approaches infinite when increasing the n-component of power law hardening stress-strain relationship from equation (1). In contrast with plastic zone, in elastic deformation zone, stress singularity tends to occurs with much higher slope with decreasing of n-component in power law hardening stress-strain relationship.

One can see that for n-component is “0” (elasto-perfectly plastic), singularity in plastic zone were disappear.

Fig. 6 shows that when ratio of young’s modulus is increasing, the plastic deformation zone will larger, also the stress singularity is much higher in both elastic deformation zone and plastic deformation zone.

5. Conclusion

From an elasto-plastic stress analysis we can evaluate the plastic zone in and elasto-plastic material joint using FEM. Therefore we can conclude:

For elasto-plastic materials stress singularity in plastic deformation zone tends to increase with more linear like behavior material; where n approaching value of “1”.

(1) In elastic deformation zone stress singularity tends to increase with more perfectly-plastic like behavior material; where n approaching value of “0”.
(2) For elasto - perfectly plastic materials singularity in plastic zone were disappear and required further investigation.
(3) The intensity of stress singularity may be applicable to bi-material models where non-linear materials are involved, except

for elastic-perfectly plastic case.

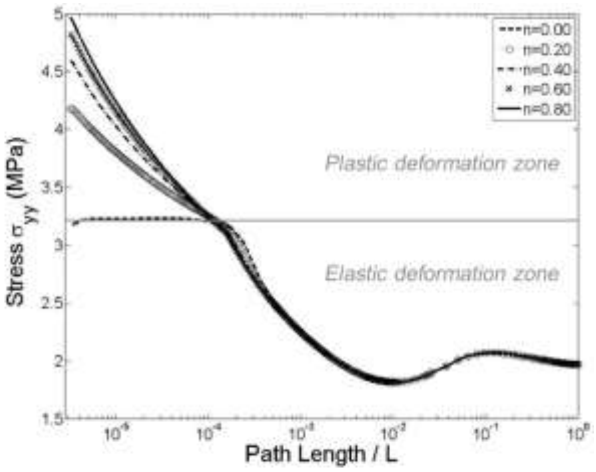

Fig. 5 Stress distribution for differrent n-exponent with Young's Modulus for hard material = 250MPa

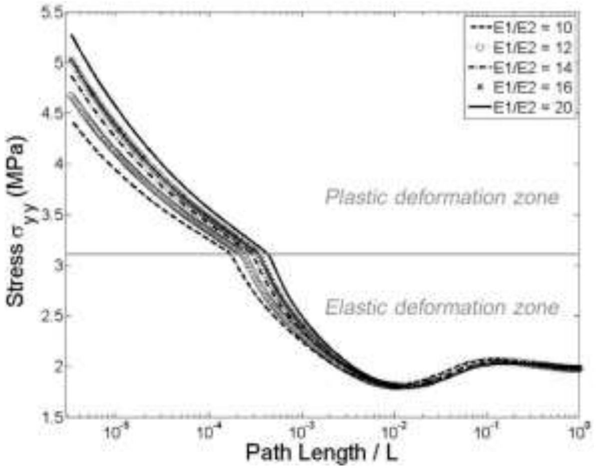

Fig. 6 Stress distribution for differrence Young's modulus ratio with n-exponent = 0.30

6. References

[1] Groth, H.L. and Brottare, (1988). Evaluation of Singular Intensity Factors in Elastic-Plastic Materials, *Journal of Testing and Evaluation*, JTEVA, Vol. 16, No. 3, May 1988, pp. 291-297.

[2] Marek Bijak-Zochowski and Piotr Marek (1994). Development of Plastic Zones and Residual Stress In Elastic-Plastic Contact Problems with Stress Singularities in Elastic Range, *Int. J. Mech. Sci.* , Vol. 38, No. 2, 1996, pp. 175-190.

[3] Y. Y. Yang, D.Munz and M. A. Sckuhr (1997). Evaluation of the plastic zone in and elastic-plastic dissimilar materials joint, *Engineering Fracture Mechanics*, Vol. 56, No. 5, 1997, pp. 691-710.

[4] Jin-Quan XU, Lie-Dong Fu and Yoshiharu Mutoh (2002). A Method for Determining Elastic-Plastic Stress Singularity at the Interface Edge of Bonded Power Law Hardening Materials, *JSME International Journal*, Series A, Vol. 45, No. 2, 2002, pp. 177-183.

[5] W.S. Barham, A.J. Aref and G.F. Dargush (2005). Development of the large increment method for elastic perfectly plastic analysis of plane frame structures under monotonic loading, *International Journal of Solids and Structures*, 42, 2005, pp. 6586-6609.

[6] X.B.Ren, Z.L.Zhang and B. Nyhus (2009). Effect of residual stresses on the crack-tip constraint in a modified boundary layer model, *International Journal of Solids and Structures*, , Vol. 46, 2009, pp. 2629-2641.