Journal of Zhejiang University SCIENCE A
ISSN 1009-3095 (Print); ISSN 1862-1775 (Online)
www.zju.edu.cn/jzus; www.springerlink.com
E-mail: jzus@zju.edu.cn

Finite element simulation of stress intensity factors in elastic-plastic crack growth

ALSHOAIBI Abdulnaser M.†, ARIFFIN Ahmad Kamal
(*Computational Mechanics Research Group, Department of Mechanical and Materials Engineering, Faculty of Engineering, Universiiy Kebangsaan Malaysia, Bangi 43600, Malaysia*)
†E-mail: alhager01@yahoo.com
Received Mar. 16, 2006; revision accepted May 25, 2006

Abstract: A finite element program developed elastic-plastic crack propagation simulation using Fortran language. At each propagation step, the adaptive mesh is automatically refined based on a posteriori *h*-type refinement using norm stress error estimator. A rosette of quarter-point elements is then constructed around the crack tip to facilitate the prediction of crack growth based on the maximum normal stress criterion and to calculate stress intensity factors under plane stress and plane strain conditions. Crack was modelled to propagate through the inter-element in the mesh. Some examples are presented to show the results of the implementation.

Key words: Crack propagation, Nodal displacement, Stress intensity factor, Adaptive mesh, Finite element method (FEM)
doi:10.1631/jzus.2006.A1336 **Document code:** A **CLC number:** O346.1; TB303

INTRODUCTION

The finite element method (FEM) has been widely employed for solving linear elastic and elastic-plastic fracture problems. The evaluation of stress intensity factors in 2D geometries by FEM is a technique widely used for non-standard crack configurations. Basically, there are two groups of estimation methods, those based on field extrapolation near the crack tip (Chan *et al*., 1970; Shih *et al*., 1976) and those using the energy release when the crack propagates. However, the high stress and strain gradients near the crack tip require very refined meshes and special elements at the crack tip. The stress intensity factors are very crucial concepts of most important magnitude in fracture mechanics. These factors define the stress field close to the crack tip and provide fundamental information on how the crack is going to propagate. Nodal relaxation is frequently used to release nodes, one by one, in order to enable the crack tip to propagate through the mesh. In contrast, methods based on near-tip field fitting procedures require finer meshes to produce a good numerical representation of crack-tip fields. The most accurate methods are those based on nodal displacements, which comprise the primary output of the finite element program (Guinea *et al*., 2000). In adaptive mesh refinement, most analysis favour either the Delaunay technique or the advancing front method over other techniques when generating meshes due to the quality of unstructured meshes generated (El-Hamalawi, 2004). The main advantage of the advancing front method is that it produces nicely graded meshes and high quality triangles that are usually very close in shape to equilaterals. The boundary integrity is also preserved, since the discretisation of the domain boundary constitutes the initial front. This is in contrast to the Delaunay triangulation, where boundary integrity is not usually preserved for complicated domains, which is a key requirement for mesh generation procedures (El-Hamalawi, 2004).

Phongthanapanich and Dechaumphai (2004) used an FEM, with the adaptive Delaunay triangulation as mesh generator to analyze 2D crack propagation problems. They described the Delaunay triangulation procedure consisting of mesh generation, node creation, mesh smoothing, and adaptive remeshing, all with object-oriented programming. They also used the displacement extrapolation method to determine the values of stress intensity factors. Rao and Rahman (2001) developed the coupled meshless-finite element method for analyzing linear-elastic cracked structures subject to Mode-I and mixed-mode conditions. Their method was applied to calculate Mode-I and Mode-II stress intensity factors in a number of 2D cracked structures.

Fan *et al.*(2004) presented an enriched partition of unity finite element method (PUFEM), which is known as one of the meshless methods to calculate the stress intensity factor in linear elastic fracture mechanics under plane stress and plane strain conditions. In their method they predicted the values of stress intensity factors for different types of crack and different types of loading.

In this paper triangular mesh generation using the advancing front method was used. The mesh is finally optimised by smoothing, and associated boundary conditions are found by interpolation from the initial geometry conditions, and finally producing the output files. The remeshing algorithms place a rosette of quarter point elements around the crack tip, and then rebuild the mesh around the crack tip. A computer code has been developed using Fortran programming language for finite element analysis calculation processes, which is based on displacement control for elastic-plastic crack propagation modelling. The program that was developed consisted of three processes involving non-linearity in geometry, material and boundary conditions. The stress intensity factors during crack propagation steps were calculated by using the displacement extrapolation method, which shown to be highly accurate, with the direction of crack propagation being predicted by using the maximum normal stress criterion.

MESH REFINEMENT

The mesh refinement is guided by a characteristic size of each element, predicted according to a given error rate and the degree of the element interpolation function. The error estimation for elastic-plastic simulation is based on stress smoothing. It is a point-wise error in stress indicator (ESI) to evaluate the accuracy of the finite element solution. Some authors (Sandhu and Liebowitz, 1995; Gallimard *et al.*, 1996) have controlled not only the discretization error but also the errors resulting from the incremental theory.

In general, the smaller mesh sizes in a finite element mesh, give more accurate finite element approximate solution. However, reduction in the mesh size leads to greater computational effort.

The error estimator used in this paper is based on stress error norm (Zienkiewicz and Zhu, 1989). The adaptivity remeshing technique is used when the element shape becomes highly distorted due to large displacement. After a few deformation increments, the whole domain is remeshed based on a stress error norm. The strategy used to refine the mesh during analysis process is adopted from (Ariffin, 1995) as follows:

(1) Determine the error norm for each element

$$\| e \|^{\mathrm{e}} = \int_{\Omega^{\mathrm{e}}} (\sigma - \sigma^*)^{\mathrm{T}} (\sigma - \sigma^*) \mathrm{d}\Omega, \qquad (1)$$

where e is the error norm, the superscript e refers to element, σ is the stress field obtained from the finite element calculation and σ^* is the smoothed stress field.

(2) Determine the average error norm over the whole domain

$$\| \hat{e} \|^{\mathrm{e}} = \frac{1}{m} \sum_{i=1}^{m} \int_{\Omega^i} \sigma^{\mathrm{T}} \sigma \mathrm{d}\Omega, \qquad (2)$$

where m is the total number of elements in the whole domain.

(3) Determine a variable, ε_{e}, for each element as

$$\varepsilon_{\mathrm{e}} = \frac{1}{\eta} \frac{(\| e \|^{\mathrm{e}})^{1/2}}{(\| \hat{e} \|^{\mathrm{e}})^{1/2}}, \qquad (3)$$

where η is a percentage that measures the permissible error for each element. If $\varepsilon_{\mathrm{e}}>1$ the size of the element is reduced and vice versa.

(4) The new element size is determined as

$$\hat{h}_{\mathrm{e}} = \frac{h_{\mathrm{e}}}{(\varepsilon_{\mathrm{e}})^{1/p}}, \qquad (4)$$

where h_e is the old element size and p is the order of the interpolation shape function.

The singular quarter point element has shown to give accurate results, but needs to have a radial concentric mesh at the crack tip. In the mesh refinement a concentric mesh around the crack tip was coupled with singular elements to model the stress field singularity as shown in Fig.1a, and the adaptive mesh refinement around the crack tip enables keeping a good precision in the vicinity of the crack. When the crack propagates through a mesh, the accuracy at the crack tip is of prime importance. In the present study, a nodal relaxation is combined with a remeshing technique. This enables avoiding the problem of distortion at the crack tip and continuing with a new, undistorted and well suited mesh. As the crack tip moves along, the areas which need to be refined will change; a new mesh is created and refined only in the areas where it is needed in order to optimise calculation time (Fig.1b). It is therefore more attractive to selectively refine the mesh in areas where the error in the approximate solution is the largest.

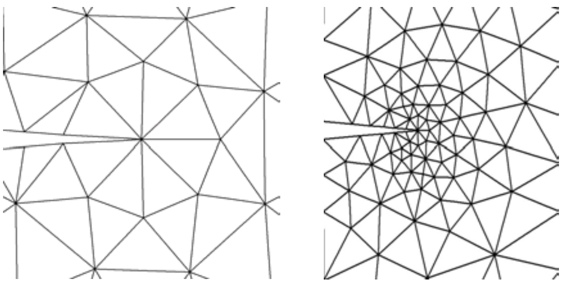


Fig.1 (a) Concentric mesh with singular element and (b) Evolutionary mesh refinement at the crack tip

STRESS INTENSITY FACTOR AND CRACK PROPAGATION

In linear elastic fracture mechanics the important parameters used are the stress intensity factors in various modes. Several methods have been proposed to determine the stress intensity factors, such as the displacement extrapolation near the crack tip (Chan *et al*., 1970), the *J*-integral (Parks, 1974) and the energy domain integral (Moran and Shih, 1987). In this paper, the displacement extrapolation method (Phongthanapanich and Dechaumphai, 2004) is used to calculate the stress intensity factors as follows:

$$K_{\mathrm{I}}=\frac{E}{3(1+\upsilon)(1+\kappa)}\sqrt{\frac{2\pi}{L}}\left[4(v_b-v_d)-\frac{(v_c-v_e)}{2}\right], \quad (5)$$

$$K_{\mathrm{II}}=\frac{E}{3(1+\upsilon)(1+\kappa)}\sqrt{\frac{2\pi}{L}}\left[4(u_b-u_d)-\frac{(u_c-u_e)}{2}\right], \quad (6)$$

where E is modulus of elasticity, υ is Poisson's ratio, κ is elastic parameter defined by

$$\kappa=\begin{cases}(3-4\upsilon), & \text{plane stress,}\\ (3-\upsilon)/(1+\upsilon), & \text{plane strain,}\end{cases}$$

and L is element length. u and v are the displacement components in the x and y directions, respectively; the subscripts indicate their positions as shown in Fig.2.

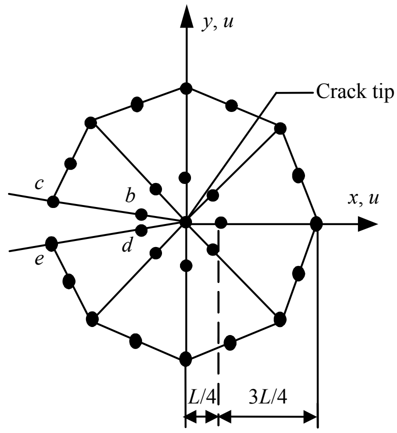


Fig.2 Quarter-point triangular elements around the crack tip

In order to simulate crack propagation under linear elastic condition, the crack path direction must be determined. There are several methods used to predict the direction of crack trajectory such as the maximum circumferential stress theory (Erdogan and Sih, 1963), the maximum energy release rate theory (Nuismer, 1975) and the minimum strain energy density theory (Sih, 1974). In the maximum circumferential stress theory, the direction of crack propagation θ is computed from

$$K_{\mathrm{I}}\sin\theta+K_{\mathrm{II}}(3\cos\theta-1)=0, \quad (7)$$

which was used in this paper to predict the crack propagation direction. Although the three criteria specify different aspects, they all yield similar results and no experimentally distinguishable differences have been observed (Akisanya and Fleck, 1992; Geubelle and Knauss, 1994; Hutchinson and Suo, 1992).

Analysis of Eq.(7) for the two pure modes showed that for pure Mode I K_{II}=0, $K_I \sin\theta$=0 and θ=0°, and for pure Mode II K_I=0 and θ=±70.5°. These values of θ are the extreme values of the crack propagation angles. The intermediary values are found by solving Eq.(7) for θ considering the mixed mode, resulting in

$$\theta = 2\arctan\left(\frac{1}{4}\frac{K_I}{K_{II}} \pm \frac{1}{4}\sqrt{\left(\frac{K_I}{K_{II}}\right)^2 + 8}\right). \quad (8)$$

NUMERICAL ANALYSIS AND VALIDATION

The method developed in this paper was applied to compute the stress intensity factors in elastic-plastic crack growth in plane stress and plane strain problems.

Single edge cracked plate under plane stress condition

The geometry of the single edge cracked plate under plane stress condition and its final adaptive mesh in the first step before crack propagation are shown in Fig.3. The plate has an initial crack length a=0.4 units, plate length L=2 units, plate width W=1 unit and the thickness t=1 unit. The far-field tensile stress σ=1 unit.

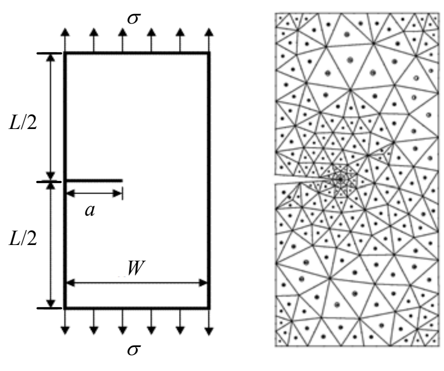


Fig.3 Problem statement (a) and the final mesh (b) of the initial crack before crack propagates

The stress intensity factor of 2.357 was calculated from (Tada *et al*., 2000) as

$$K_I = \sqrt{\pi} F \sigma a, \quad (9)$$

where F=1.12−0.231α+10.55α^2−21.72α^3+30.39α^4 and α=a/W.

Phongthanapanich and Dechaumphai (2004) evaluated the value of K_I=2.358 by using FEM with adaptive Delaunay triangulation as mesh generator. Also by using an efficient meshless method, Rao and Rahman (2001) calculated the values of stress intensity factors for the same geometry and boundary condition as shown in Table 1, and compared there results with the reference value of stress intensity factor (Tada *et al*., 2000).

Table 1 Values of K_I (Rao and Rahman, 2001)

L_{EFGM}/L	K_I
0.8	2.3441
0.7	2.3359
0.6	2.3423
0.5	2.3569
0.4	2.3724
0.3	2.3867
0.2	2.3701
0.1	2.1644

By using the developed Fortran code the predicted values of stress intensity factor using the displacement extrapolation method are presented in Table 2 and yield good agreement with the reference value of K_I (Tada *et al*., 2000).

The calculated values of stress intensity factors during crack propagation steps using the developed program were compared with values which were calculated by FRANC2D/L (http://www.cfg.cornell.edu) program with the same boundary condition, loading, crack growth criteria, and crack direction criteria. The results of this comparison are shown in Fig.4, and the agreement is clearly good.

Fig.5 shows the final configuration corresponding to the last evaluated crack length for the results obtained from the Fortran developed code in the present study, and that obtained from FRANC2D/L program. The behaviors of crack propagation are almost the same as shown in this figure.

Table 2 Values of stress intensity factors during crack propagation steps

Step	Crack length	K_I
1	0.50203	2.344634
2	0.90338	2.350420
3	1.30463	2.366343
4	1.70584	2.367035
5	2.10700	2.357465
6	2.50811	2.345653
7	2.90905	2.328344
8	3.31002	2.312559
9	3.71091	2.300761
10	4.11170	2.280170
11	4.51240	2.267931
12	4.91295	2.268662
13	5.31326	2.276166
14	5.71323	2.339646

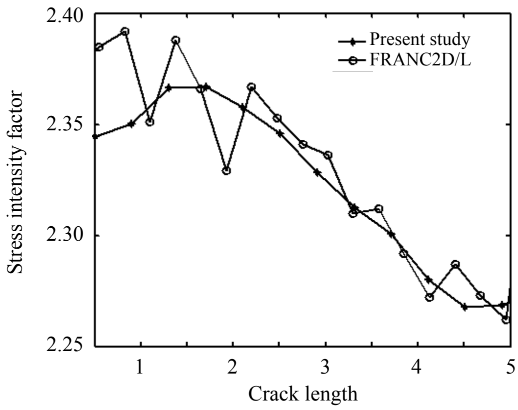


Fig.4 Comparison of calculated stress intensity factors with FRANC2D/L program

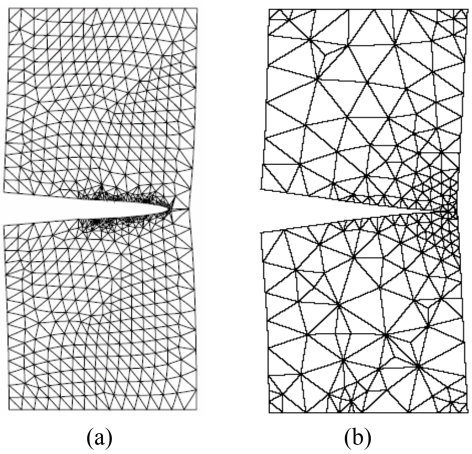


Fig.5 Final configuration corresponding to the last evaluated crack length. (a) FRANC2D/L; (b) Present study

Symmetrically cracked rectangular panel subjected to uni-axial tension

A rectangular panel having a half-through crack are subjected to a uni-axial tension. It is considered as a plane strain problem (Fan *et al*., 2004). The geometry of this panel is the same as that of the single edge cracked plate shown in Fig.3, difference exists only in the geometry dimensions. The width of the panel is W=2 units, the length is L=4 units, and the initial crack length a=1 unit. The modulus of elasticity E is 1, and Poisson's ratio is 0.3 (Tada *et al*., 2000). Uni-axial tension σ=1 is applied along the shorter edges. It leads to the symmetric Mode-I deformation.

Duarte (1996) previously solved the same problem when they calculated the stress intensity factors. His results (K_I=5.10) was used as benchmarks for (Fan *et al*., 2004). The results of K_I obtained by Fan *et al*.(2004) are shown in Table 3.

Table 3 K_I obtained by Fan *et al*.(2004)

Order of enrichment (P)	K_I
2	2.595
3	4.528
4	5.045
5	4.996
Benchmark solution (Duarte, 1996)	5.100

By using the developed Fortran code the predicted values of stress intensity factor using the displacement extrapolation method are presented in Table 4 and yield good agreement with the reference values in (Duarte, 1996).

Table 4 Values of stress intensity factors during crack propagation steps for plane strain condition

Step	Crack length	K_I
1	0.50329	5.170447
2	1.00555	5.204885
3	1.50777	5.208066
4	2.00991	5.188438
5	2.51195	5.151423
6	3.01368	5.111819
7	3.51496	5.067196
8	4.01605	5.018521
9	4.51673	4.989732
10	5.01698	4.988881
11	5.51656	5.090815

The calculated values of stress intensity factors during crack propagation steps using the developed program were compared with that of the values which were calculated by FRANC2D/L program with the same boundary condition, loading, crack growth criteria, and crack direction criteria. The results of this comparison are shown in Fig.6, and there is also good agreement.

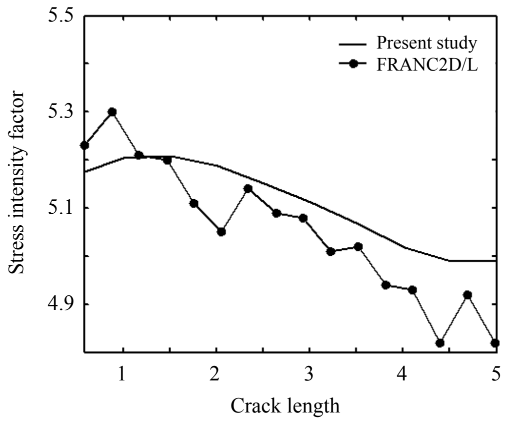
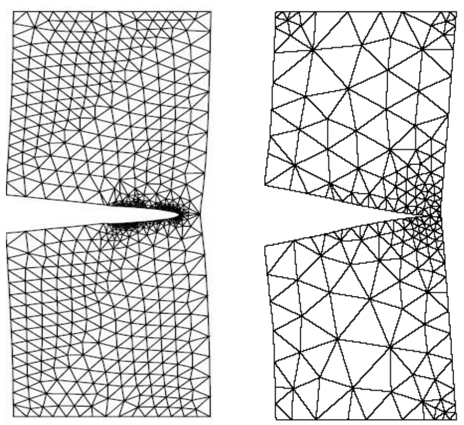


Fig.6 Comparison of calculated stress intensity factors with FRANC2D/L program

Fig.7 shows the final configuration corresponding to the last evaluated crack length for the results obtained from the Fortran developed code in the present study, and that obtained from FRANC2D/L program. The behaviors of crack propagation in plane strain are almost the same as shown in this figure.



(a) (b)

Fig.7 Final configuration corresponding to the last evaluated crack length. (a) FRANC2D/L; (b) Present study

CONCLUSION

The adaptive finite element program using the advancing front method has been developed to simulate elastic-plastic crack propagation using Fortran language. The isoparametric six-node triangular elements, with mid-side nodes displaced from their nominal position to quarter points at the crack tip, were employed to form a circular zone surrounding the tip in order to better capture the stress field. The stress intensity factors (SIFs) were predicted by using the displacement extrapolation method for plane stress and plane strain in pure Mode I (plane tensile). The predicted values of SIFs were compared with the standard reference (Tada *et al.*, 2000), some relevant publications (Phongthanapanich and Dechaumphai, 2004; Rao and Rahman, 2001; Fan *et al.*, 2004; Duarte, 1996), and also with a crack propagation software (http://www.cfg.cornell.edu), yielding good agreement. The predicted SIFs demonstrated the capability of the developed program for solving crack propagation problems under plane stress and plane strain conditions.

References

Akisanya, A.R., Fleck, N.A., 1992. Brittle fracture of adhesive joints. *International Journal of Fracture*, **58**(2):93-114. [doi:10.1007/BF00019971]

Ariffin, A.K., 1995. Powder Compaction, Finite Element Modelling and Experimental Validation. Ph.D Thesis, University of Wales Swansea.

Chan, S.K., Tuba, I.S., Wilson, W.K., 1970. On the finite element method in linear fracture mechanics. *Engineering Fracture Mechanics*, **2**(1):1-17. [doi:10.1016/0013-7944(70)90026-3]

Duarte, C.A., 1996. The Hp Clouds Method. Ph.D Thesis, University of Texas, Austin.

El-Hamalawi, A., 2004. A 2D combined advancing front Delaunay mesh generation scheme. *Finite Element in Analysis and Design*, **40**:967-989. [doi:10.1016/j.finel.2003.04.001]

Erdogan, F., Sih, G.C., 1963. On the crack extension in plates under plane loading and transverse shear. *Journal of Basic Engineering*, **85**:519-527.

Fan, S.C., Liu, X., Lee, C.K., 2004. Enriched partition-of-unity finite element method for stress intensity factors at crack tips. *Computers and Structures*, **82**:445-461. [doi:10.1016/j.compstruc.2003.10.019]

Gallimard, L., Ladevèze, P., Pelle, J.P., 1996. Error estimation and adaptivity in elastoplasticity. *International Journal of Numerical Methods in Engineering*, **39**(2):189-217. [doi:10.1002/(SICI)1097-0207(19960130)39:2<189::AID-NME849>3.0.CO;2-7]

Geubelle, P.H., Knauss, W.G., 1994. Crack propagation at and near bimaterial interfaces: linear analysis. *Journal of Applied Mechanics*, **61**:560-566.

Guinea, G.V., Planan, J., Elices, M., 2000. K_I evaluation by the displacement extrapolation technique. *Engineering Fracture Mechanics*, **66**(3):243-255. [doi:10.1016/S0013-7944(00)00016-3]

Hutchinson, J.W., Suo, Z., 1992. Mixed mode cracking in layered materials. *Advances in Applied Mechanics*, **29**:63-191.

Moran, B., Shih, C.F., 1987. A general treatment of crack tip contour integrals. *Engineering Fracture Mechanics*, **35**:295-310.

Nuismer, R.J., 1975. An energy release rate criterion for mixed mode fracture. *International Journal of Fracture*, **11**(2):245-250. [doi:10.1007/BF00038891]

Parks, D.M., 1974. A stiffness derivative finite element technique for determination of crack tip stress intensity factors. *International Journal of Fracture*, **10**(4):487-502. [doi:10.1007/BF00155252]

Phongthanapanich, S., Dechaumphai, P., 2004. Adaptive Delaunay triangulation with object-oriented programming for crack propagation analysis. *Finite Element in Analysis and Design*, **40**:1753-1771. [doi:10.1016/j.finel.2004.01.002]

Rao, B.N., Rahman, S., 2001. A coupled meshless-finite element method for fracture analysis of cracks. *International Journal of Pressure Vessels and Piping*, **78**(9):647-657. [doi:10.1016/S0308-0161(01)00076-X]

Sandhu, J.S., Liebowitz, H., 1995. Examples of adaptive FEA in plasticity. *Engineering Fracture Mechanics*, **50**:947-956. [doi:10.1016/0013-7944(94)E0043-G]

Shih, C.F., de Lorenzi, H.G., German, M.D., 1976. Crack extension modeling with singular quadratic isoparametric elements. *International Journal of Fracture*, **12**:647-651.

Sih, G.C., 1974. Strain-energy-density factor applied to mixed-mode crack problems. *International Journal of Fracture*, **10**(3):305-321. [doi:10.1007/BF00035493]

Tada, H., Paris, P.C., Irwin, G.R., 2000. The Stress Analysis of Cracks Handbook. ASME Press, New York.

Zienkiewicz, O.C., Zhu, J.Z., 1989. Error estimates and adaptive refinement for plate bending problems. *International Journal for Numerical Methods in Engineering*, **28**(12):2839-2853. [doi:10.1002/nme.1620281209]