Structural Engineering and Mechanics, *Vol. 57, No. 6 (2016) 1143-1156*
DOI: http://dx.doi.org/10.12989/sem.2016.57.6.1143

The investigation crack problem through numerical analysis

Murat Yaylacı*

Department of Civil Engineering, Recep Tayyip Erdoğan University, 53100, Rize, Turkey

(Received June 8, 2015, Revised December 15, 2015, Accepted February 17, 2016)

Abstract. This paper presents a comparative study of finite element method (FEM) and analytical method for the plane problem of a layered composite containing an internal perpendicular crack in literature. The layered composite consists of two elastic layers having different elastic constants and heights. External load is applied to the upper elastic layer by means o a rigid punch and the lower elastic layer rests on two simple supports. Numerical simulations are realized by the world wide code ANYS software. Two dimensional analysis of the problem is carried out and the results are verified by comparison with solutions reported in literature. Main goal of the numerical simulation is to investigate the normal stress $\sigma_x(0,y)$, stress intensity factors at the crack factor and the crack opening displacements.

Keywords: fracture mechanics; crack; stress intensity factor; FEM

1. Introduction

The numerical methods represent many years leading computational utility. Initially uninteresting finite element method (FEM) has today become one of the main computing resources in the engineering industry. The main advantage FEM is graphic interpretation often very abstract phenomena in which classical technique solutions introduces considerable simplification at the expense of accuracy Hadju (2014).

Even though failure due to presence of crack or crack like defects has been observed both in huge structures such as airplanes and automobiles and in the micron level structures such as flip-chip and wire bond packages in semiconductor industry since the creation of the earliest man-made structures, however, the formulations of various fracture theories and the understanding of this phenomenon rapidly accelerated during the 20th century Khan *et al*. (2013).

In application of most of the current fracture criteria, the stress-intensity factor and the crack opening displacement are the mostly used quantities. The stress-intensity factor defines the stress field close to the tip of a crack and provides fundamental information of how the crack is going to behave, whether it is to expand causing the failure of a structural element or it is remain stable. This is reason why a great deal of research has been devoted to this topic in recent years. Using different methods, some of which are Integral transform technique, finite element method, and other computational methods, many problems relating to cracks, i.e., in a strip or layer, a half plane, bonded materials, and layered composites, have treated Birinci and Erdol (2004). Fracture

*Corresponding author, Professor, E-mail: murat.yaylaci@erdogan.edu.tr

http://www.techno-press.org/?journal=sem&subpage=8 ISSN: 1225-4568 (Print), 1598-6217 (Online)

mechanics can be used to assess materials resistance against cracking and its durability through a critical fracture parameter as the critical stress intensity factor or the critical energy release rate in the range of linear elastic material behavior. All three modes of crack propagation (Modes I, II, III) maybe present under general loading conditions, and the full set of fracture tests should measure the critical fracture parameter under opening, plane and anti-plane shear fracture modes Figiel *et al.* (2004).

Leverenz (1972) used numerical finite element solution techniques to determine stress intensity factors for a crack in one material of a bi-material plate, with the crack located perpendicular to the material interface. It was found that the finite element results were in well agreement with the numerical results of Isida (1970).

A two dimensional plane-strain finite element (FE) model was created with a Hertzian contact pressure that moves over a crack. Considering an elasto hydrodynamic (EHD) contact pressure profile and liquid entrapped between the crack faces, both KI and KII increase significantly, because of the liquid's pressure. Reduced FC lets the crack faces slide over each other, and the shear mode of fracture takes place at lower SIFs. A study done by Bogdanski (2002) shows the magnitude of KI under an EHD contact pressure is almost 250% greater than a similar dry contact condition. Figiel *et al*. (2004) studied computational analysis of a compression shear fracture test, proposed for interface fracture toughness determination and crack propagation analysis in curved layered composites. Numerical analysis using contact finite elements is paid to the near crack-tip displacements and stresses. The comparison between finite element method and analytically determined stresses is made. This study shows that energy release rate of the composite considered strongly depends on the interfacial friction coefficient. Qiao and Wang (2004) investigated novel interface deformable bi-layer beam theory to account for local effects at crack tip of bi-material interface by modeling a bi-layer composite beam as two separate shear deformable sub-layers with consideration of crack tip deformation. In this study, an elastic deformable crack tip model is presented for the first time which can improve the split beam solution. The present model is in excellent agreements with analytical 2-D continuum solutions and finite element analyses. Papadopoulos *et al*. (2008) solved crack problem by analytical and numerical methods. In the paper an experimental and an numerical determination of mode-I stress intensity factor (SIF), KI at the tip of an edge crack reinforced with bonded patches is undertaken by using the optical method of caustics and the finite element analysis (FEA). The experimental results are compared with the corresponding one obtained by FEA ANYS. Chaari *et al*. (2009) derived an analytical formulation of the time varying gear mesh stiffness and presented an original analytical modeling of tooth cracks, the gear mesh stiffness reduction due to this fault was quantified. A comparison with finite element model was presented in order to validate the analytical formulation. A simple method to identify multiple cracks in a beam using the vibration amplitudes is examined by Lee (2009). The cracks are modeled as massless rotational springs and the forward problem is solved using the finite element method. A two-dimensional finite element model is built to simulate the experimental results and to provide vibration amplitude measurements. Rolling contact fatigue cracks in railway track called squats are studied by Farjoo *et al*. (2014). The effects of an elastic foundation (sleepers and the ballast) on stress intensity factors obtained at a crack tip are studied. A simplified finite element model (FEM) and an extended finite element model (XFEM) were created to investigate these effects, the XFEM model being limited in geometrical size, but more able to model crack growth. Khan *et al.* (2013) investigated elastic-plastic solutions for the dynamically loaded cracked structures. The path independent integral originally developed for a two-dimensional dynamically loaded stationary circular arc crack in a homogeneous and isotropic material, is evaluated in the present study by taking elastic-plastic material properties to quantify the crack problem. All the numerical results presented in this study were evaluated using general purpose finite element commercial software ANSYS together with the post- processing program (using FORTRAN) developed by the authors. Shao *et al*. (2013) solved dynamic and three-dimensional finite element analytical models of cracked gears using the theory of fracture mechanics and the finite element method by ANSYS. The influence of crack position and length on the dynamic characteristics of the gear structure are simulated and discussed. Pandya and Parey (2013) adopted a 2-D finite element method with principle of linear elastic fracture mechanics to carry out crack propagation path studies for gear pair with different contact ratio. The computational method proposed here can predict the change of gear mesh stiffness for different crack propagation paths to provide some guidance for gear damage detection. The comparative studies of numerical solution and analytical solution of the contact problem are conducted by Yaylaci *et al*. (2014), Adiyaman *et al*. (2015), Birinci *et al*. (2015), Öner *et al*. (2015). Hajdu (2014) studied the fracture problem which is solved by ANSYS. The author's main goal of the numerical simulation is to investigate the stress state near the crack tip through determination of stress intensity factors.

In the present study, the problem of a layered composite which contains an internal or edge crack perpendicular to its boundaries in its lower layer and which rests on two simple supports is investigated. The layered composite consists of two elastic layers having different elastic constants and heights, and is subjected to a concentrated load $2P$ by means of a rigid rectangular stamp of which width is *2a*, shown in Fig. 1. It is assumed that all surfaces are frictionless .In the actual problem crack surface is free of traction as studied by Birinci and Erdol (2004). The problem is developed based on the FEM ANSYS (2013) software. The numerical results for the normal stress $\sigma_x(0,y)$, stress intensity factors and the crack opening displacements are obtained for various quantities and shown in the figures. The numerical results are verified by comparison with analytical results in literature .

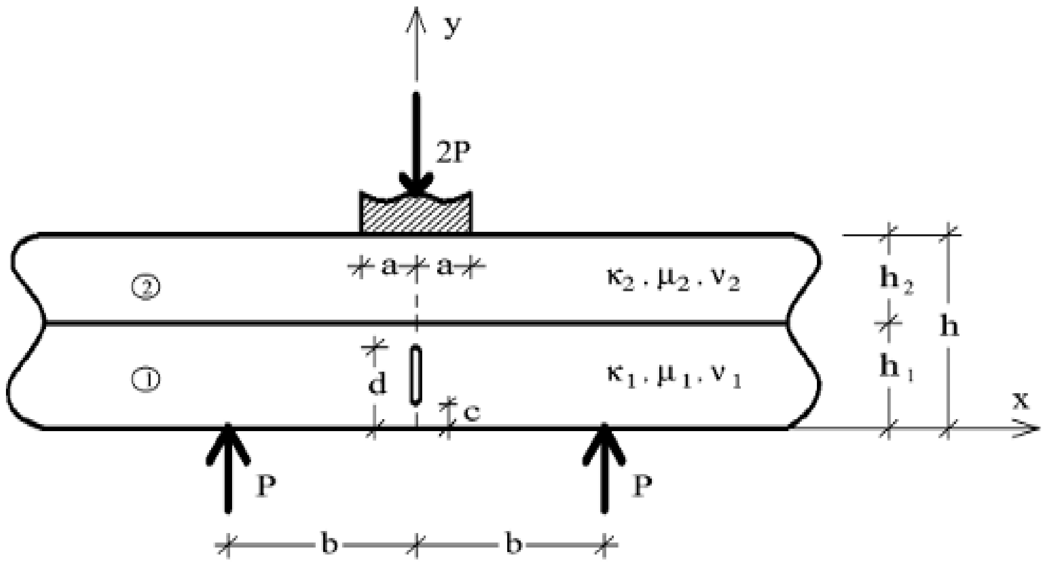
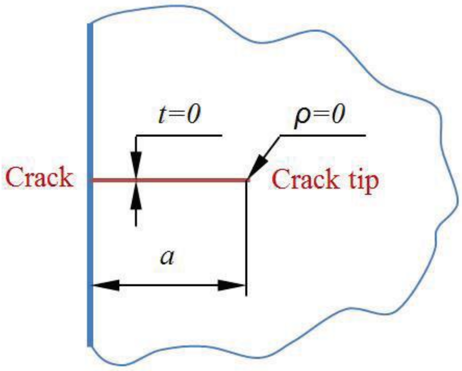


Fig. 1 Geometry of the layered composite containing a crack in its lower layer



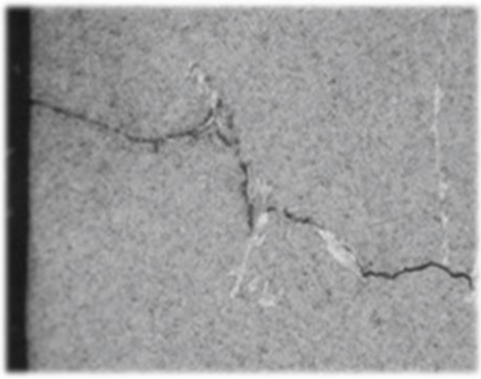
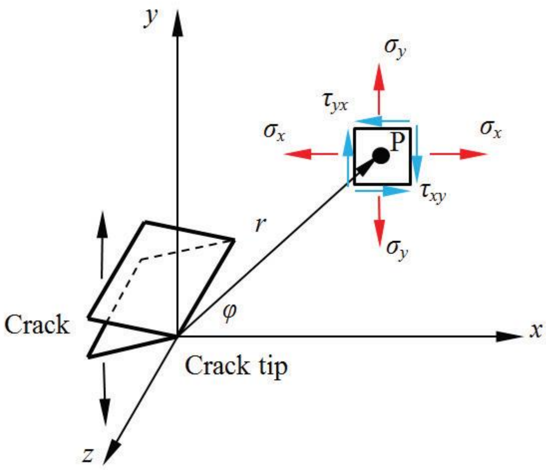


Fig. 2 Ideal computational model of a crack and the real crack in the material



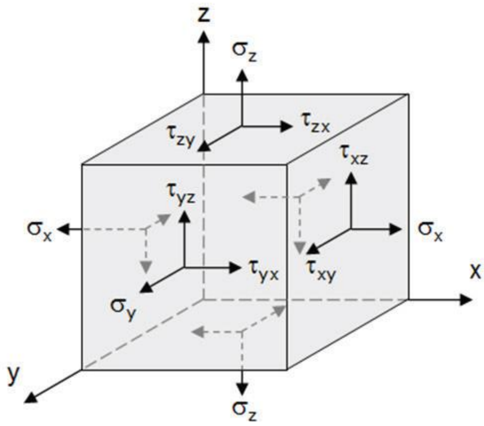


Fig. 3 Infinite plate with a central crack and the Cauchy stress tensor

2. Theoretical base

A crack is the most important case of a failure of the body continuity in terms of strength. A surface violation of continuity with the zero thickness and thus with the zero radius of the curvature on the tip of crack is the most preferable calculation model of a crack. Fig. 2 shows this case. The tip of the crack is in this case singular point in which is obtained infinite mechanical stresses. Therefore the state of stress near the crack tip is very inhomogeneous. Thus the plastic deformation in the crack tip will be occurred also at very small loading Hadju (2014). For a classification of a stress state around a crack is considered an infinite plate with a central crack (Fig. 3).

For the classification of a stress state around a crack is considered an infinite plate with a central crack (Fig. 3).

In the point P of the object which has coordinates r, φ with regard to the crack tip is the stress state given to the stress tensor T_{σ}. The individual components of the stress tensor can be expressed by the formula

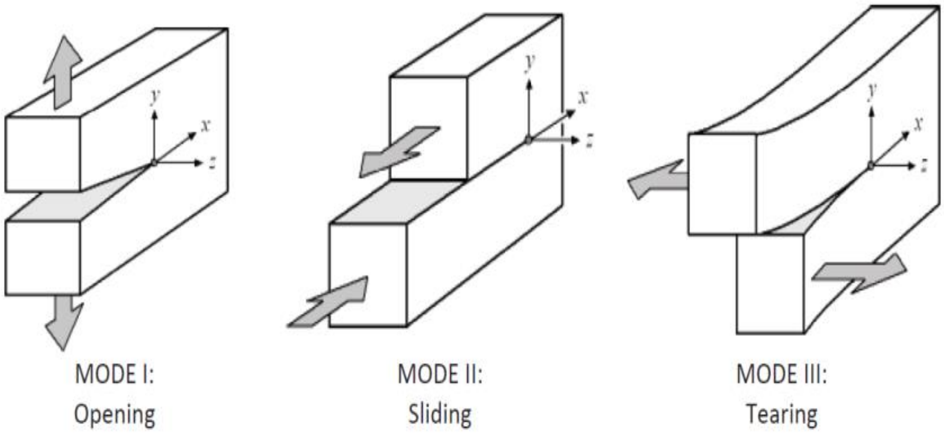


Fig. 4 Three modes of the crack deformation

$$\sigma_{ij} = \frac{K}{2\pi r} f_{ij}(\varphi) \tag{1}$$

Where $f_{ij}(\varphi)$ is function which depends only on a type of a crack opening mode. The parameter K depends on the geometry of the crack, geometry of the part itself and from the nominal stress in the sufficient distance from the crack tip. Irwin named this parameter as a stress intensity factor. For the objects of finite dimensions with variously situated cracks loaded with different modes can be expressed this coefficient in the form

$$K = \sigma\sqrt{\pi a}\, f_k(a,b) \tag{2}$$

where b is the dimension of the part in the crack seam direction and the function $f_k(a,b)$ depends on the boundary conditions of given configuration of the object with a crack.

There are three types of crack opening, termed modes I, II, and III which are shown in Fig. 4. Mode I is a normal opening mode while modes II and III are shear sliding modes Hadju (2014).

If stress intensity factors K_I, K_{II}, K_{III} are known for all these modes, the stress state near the tip of the crack for linear elastic material is described by the equation (the principle of superposition).

$$\sigma_{ij} = \frac{1}{2\pi r} \sum_{i=I}^{III} K_n f_{ij,n}(\varphi) \tag{3}$$

The stress intensity near a crack tip characterizes the stress intensity factor by the theory K-conception. If critical value of the stress intensity factor is known for the used material and the calculated stress intensity factors then can be said whether the crack will be grow or not. The critical stress intensity factor for mode I obtained by experimental test of the material $K_{IC}$is called as the plane strain fracture toughness Hadju (2014).

2. Numerical approach

The finite element method (FEM) is a numerical technique used for finding approximate solutions of partial differential and integral equations. The method works by assuming a continuous function for the solution and obtaining the parameters that govern this function which minimizes the error in the solution. For many engineering problems analytical solutions are not suitable because of the complexity of the material properties, the boundary conditions on the structure itself. The basis of the finite element method is the representation of a body or a structure by an assemblage of subdivisions called finite elements. The Finite Element Method translates partial differential equation problems into a set of linear algebraic equations.

$$[K]\{q\}=\{F\} \tag{4}$$

where $[K]$ is the global stiffness matrix, $\{q\}$ the structural nodal displacement vector and $\{F\}$ is the vector of structural nodal loads Delpero *et al*. (2010).

Recent developments in computer technology have made significant contributions toward the application of numerical methods, especially the FEM, which has become an essential solution technique in many areas of engineering. In the FEM, the structure is divided into a large number of predetermined elements, and the solution of the large number of equation sets, which is obtained by combining these elements with nodal points, is achieved. Determination of the unknowns in these equation sets and the values at the nodal points to be calculated requires computer use Altunsaray and Bayer (2013). There are many package programs based on the FEM. One of these programs is ANSYS software. The ANSYS software provides a comprehensive mechanism to combine numerical analyses of two or more different interrelated physical fields within a single model. Problems in different fields (e.g., structural, thermal, electromagnetic physics, acoustics, fluid dynamics) can be solved directly or iteratively Sabliov *et al*. (2007). The ANSYS (2013) software making solutions with the FEM is used in this study for comparing the analytical results.

The software consists of three parts: pre-processing module, analysis of the calculation module and post-processing module. Pre-processing module provides a powerful modeling and meshing tools, users can easily construct a finite element model; Analysis modules include structural analysis (linear analysis, nonlinear analysis, and highly non-linear analysis), fluid dynamics, electromagnetic field analysis, analysis of the sound field, piezoelectric analysis and multi-physics coupling analysis can simulate the interaction of a variety of physical media, with a sensitivity analysis and optimization analysis capabilities; post-processing module can calculate the results in color contour gradient vector particle flow trace display, three-dimensional slice, transparent and translucent (can be seen within the structure) and graphically displayed, the calculation results are displayed in the graphs, form or outputs. The software provides more than 100 kinds of cell types used in the analog engineering structures and materials Ji *et al*. (2010).

The problem is considered as a two-dimensional contact problem and the material of the layers are assumed elastic and isotropic. The physical system under consideration exhibits symmetry in geometry, material properties and loading. Taking advantage of symmetry, only one half of the geometry of the problem is to be modeled. . In the analyses, geometric properties are taken as $L = 2$ m (length of the layer in x direction), $h_2 = 20$ cm (thickness of the lower layer in y direction), $P = 1000$ N load and material properties are taken as $E_1 = 30000$ MPa $, \nu_1 = 0.2$. Other parameters are chosen such that b/h , a/h , d/h , c/h , h_1/h and E_2/E_1 ratios are compatible with dimensionless values which are obtained analytical solution. The geometry and the applied load are shown schematically in Fig. 5.

In ANSYS a concentration keypoint is generated at the crack tip. KCALC command is used for determination of stress intensity factors after the local coordinate system is defined and used at the

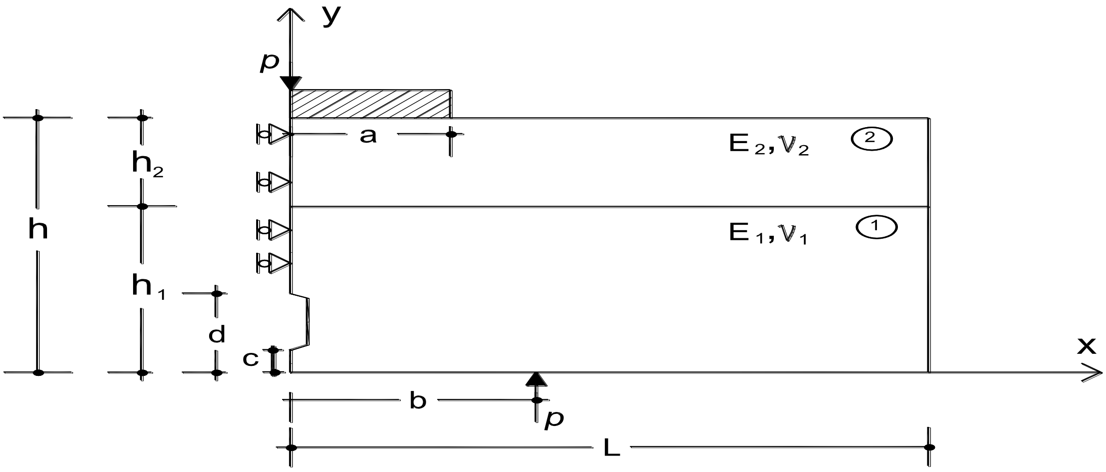


Fig. 5 The geometry for the analysis

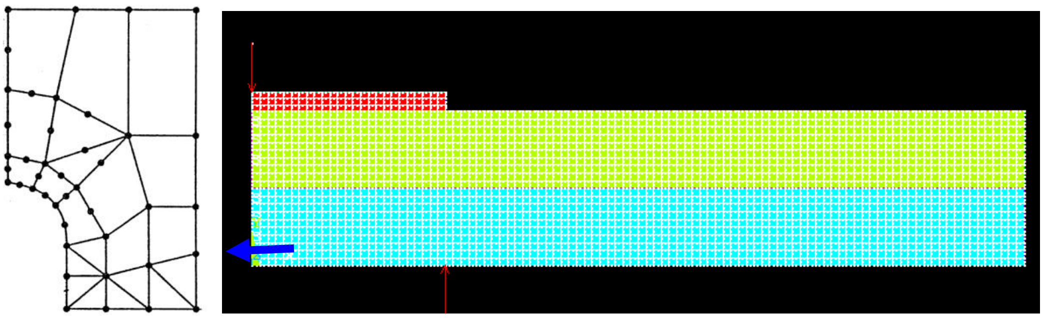


Fig. 6 An edge crack and the finite element mesh

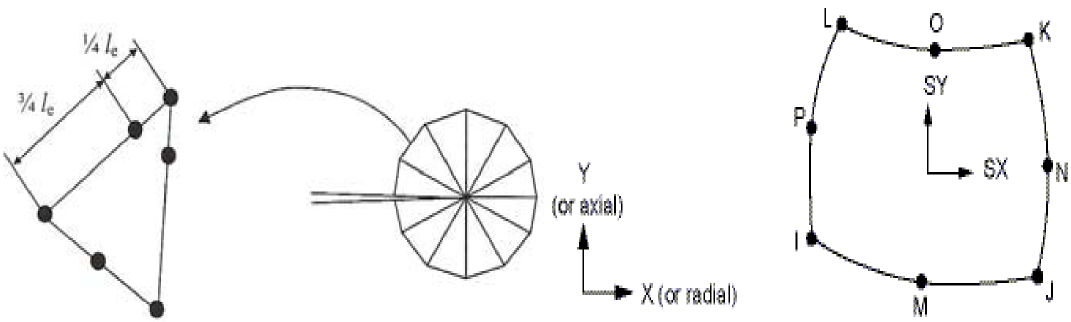


Fig. 7 Element 8-node PLANE 183

crack tip and the crack path defined in the post processer using path operations. Finite Element model of the problem before analysis and the model of crack as modeled using ANSYS are shown in Fig. 6.

The program ANSYS is used in the finite element analysis (FEA) modeling. The mesh is generated using two dimensional solid 8-node PLANE 183 for the crack and the crack-tip of the model (Fig. 7). PLANE183 is defined by eight nodes having two degrees of freedom at each node:

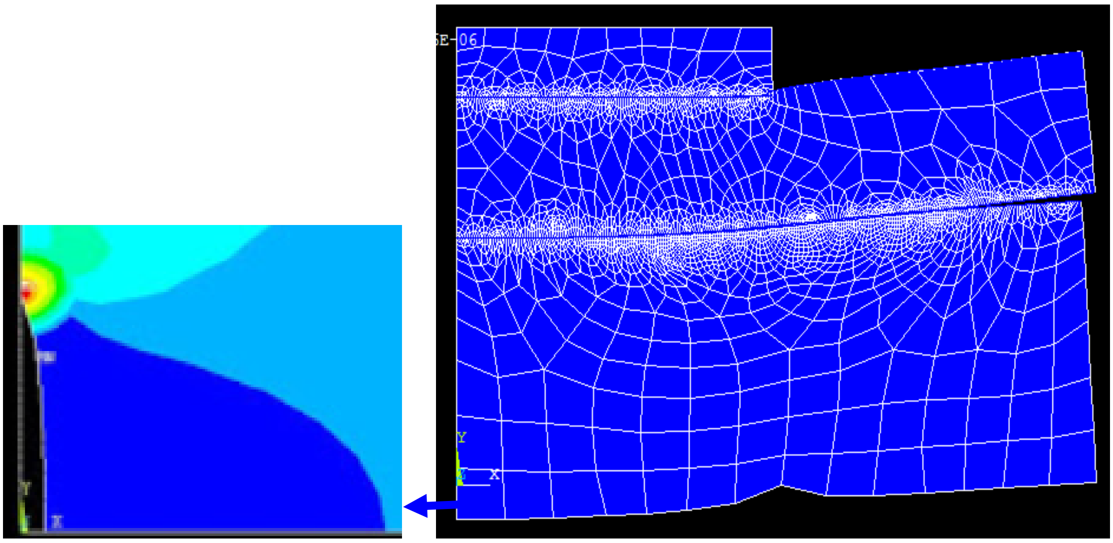


Fig. 8 Deformed geometry for the preliminary analysis

translations in the nodal x and y directions. In addition this element has the capability, plasticity, elasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities. Mesh size and configuration are important parts of modeling with precise mesh refinement being necessary in regions of high-stress intensity. However, as the mesh was made finer, the number of elements increased resulting in increased memory and computational time requirements. The software was limited to 150,000 nodes. Therefore, some modifications (smaller dimension) were required in the modeling process Zandi and Akpinar (2012). The preliminary analysis is meshed with 4435 elements and 8444 nodes. Deformation shape after analysis by using these elements is shown in Fig. 8.

3. Numerical results and discussion

Finite element analysis is briefly explained for the analytical results in literature Birinci and Erdol (2004). Stress intensity factors, axial stress and crack surface displacement are obtained for various values using finite element analysis with the aid of ANSYS (2013). The results from the running ANSYS codes, a full two-dimensional model, for the problem are shown in Fig. 9-16. The presents results compared with already available in the literature which is studied by Birinci and Erdol (2004). As shown in the results given below, the results coincide well with the known theoretical results which indicate the combination finite element method used here appears to be a good approximation method to solve actual crack problems. The comparison of results with those in literature and with the finite element software ANSYS are found in good agreement. The results consist of three groups; the first group is concerned with stress intensity factor for various values $d/h, c/b$ and β conditions. The second group is concerned with crack surface displacement for various values $d/h, c/b$ and β conditions. The third group is concerned with axial stress for various values b/h conditions.

It is important to compare the stress-intensity factors as assumed in the analytical model with

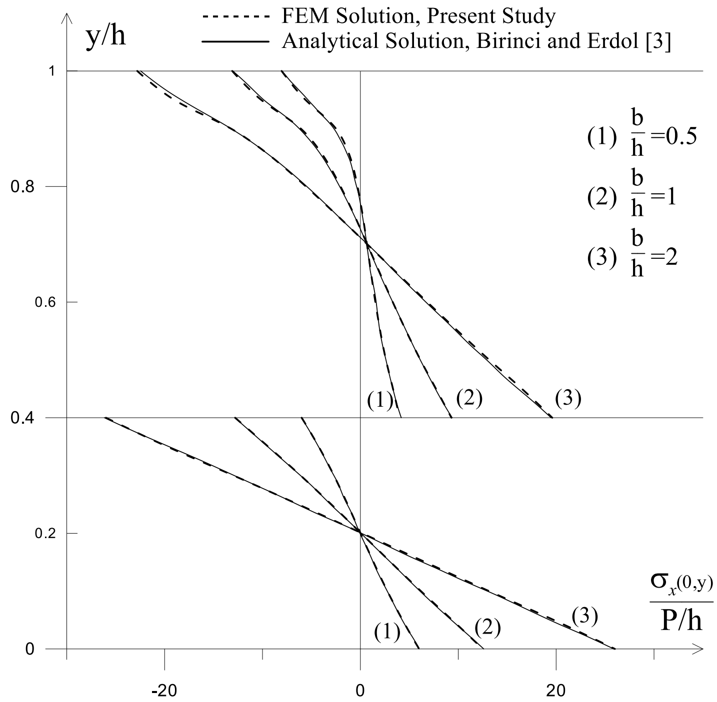


Fig. 9 Comparisons of analytical solution with FEM solution for the normalized axial stress $\sigma_x(0,y)/(\frac{P}{h})$ in the layered composite without a crack for various values of b/h $(a/h = 0.1,\ h_1/h = 0.4,\ \beta = 0.5)$

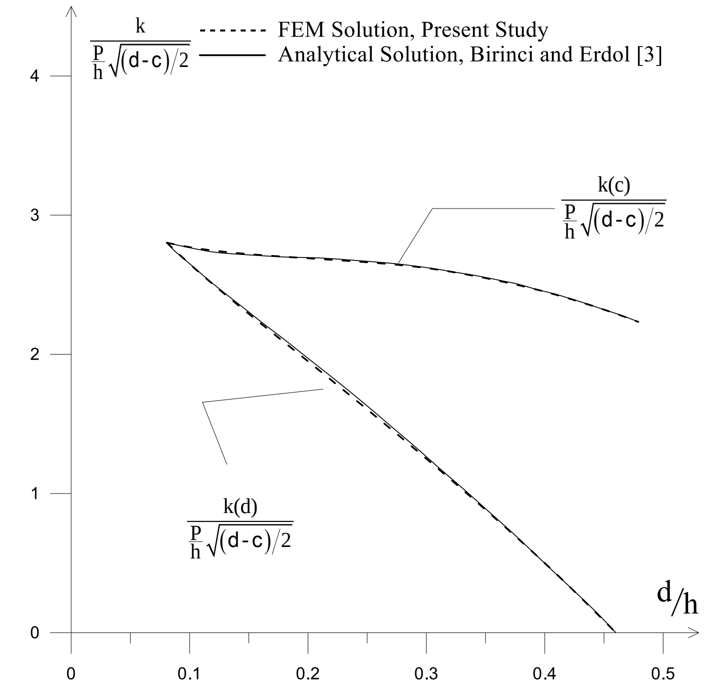


Fig. 10 Comparisons of analytical solution with FEM solution for the normalized stress-intensity factors with d/h for an internal crack $(a/h = 1,\ h_1/h = 0.7,\ b/h = 1,\ c/h = 0.08,\ \beta = 0.1)$

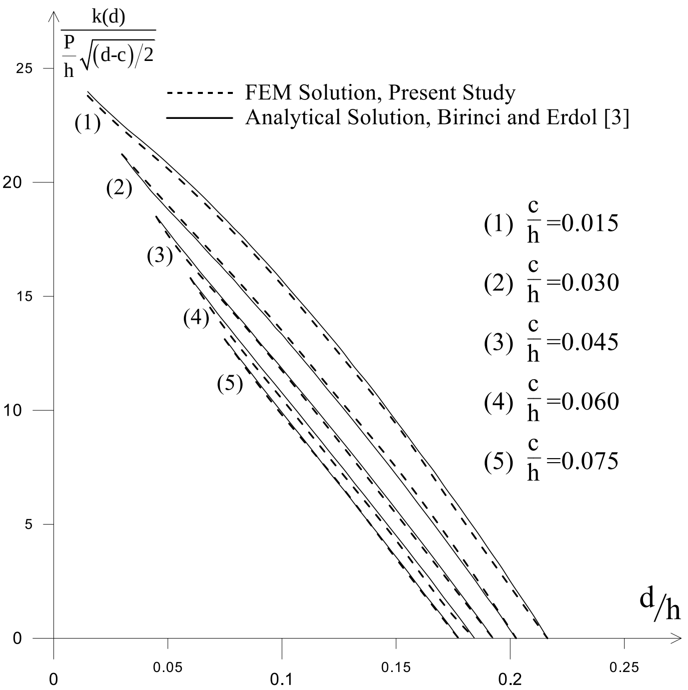


Fig. 11 Comparisons of analytical solution with FEM solution for the normalized stress-intensity factors $k(d)$ with d/h for various values of c/h for the case of an internal crack ($a/h = 1, h_1/h = 0.3,\ b/h = 1,\ \ \beta = 0.1$)

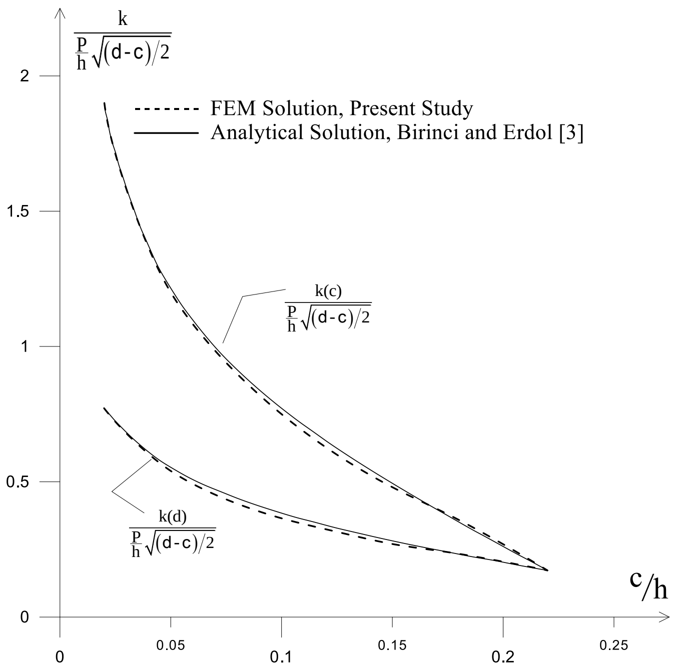


Fig. 12 Comparisons of analytical solution with FEM solution for the normalized stress-intensity factors with c/h for an internal crack ($a/h = 2,\ h_1/h = 0.5,\ b/h = 1,\ \beta = 0.22$)

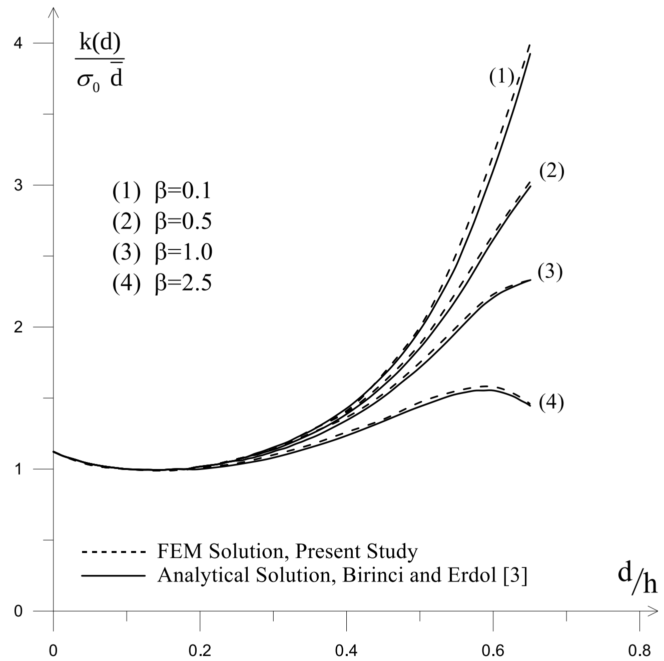


Fig. 13 Comparisons of analytical solution with FEM solution for the normalized stress-intensity factors $k(d)$ with d/h for various values of β for the case of an edge crack ($a/h = 0.2$, $h_1/h = 0.8$, $b/h = 1, \sigma_0 = \sigma_{1x}^*(0,0)$)

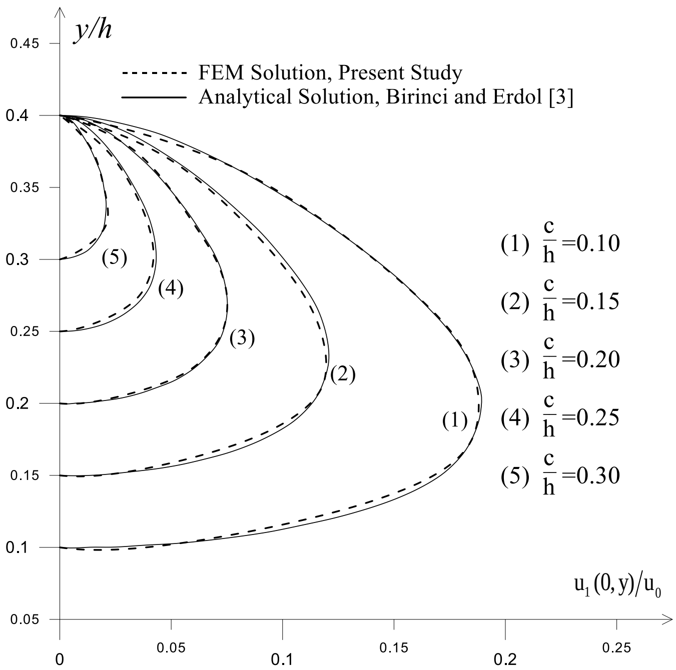


Fig. 14 Comparisons of analytical solution with FEM solution for the normalized crack surface displacement for various of c/h for the case of an internal crack ($a/h = 0.15$, $h_1/h = 0.8$, $b/h = 1$, $d/h = 0.4$, $\beta = 0.1$)

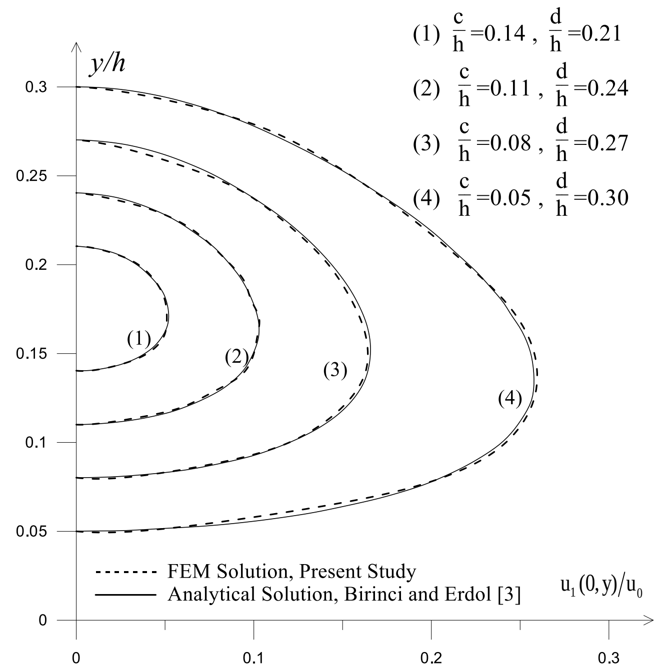


Fig. 15 Comparisons of analytical solution with FEM solution for the normalized crack surface displacement for various of c/h and d/h for the case of an internal crack ($a/h = 0.1$, $h_1/h = 0.7$, $b/h = 1$, $\beta = 1$)

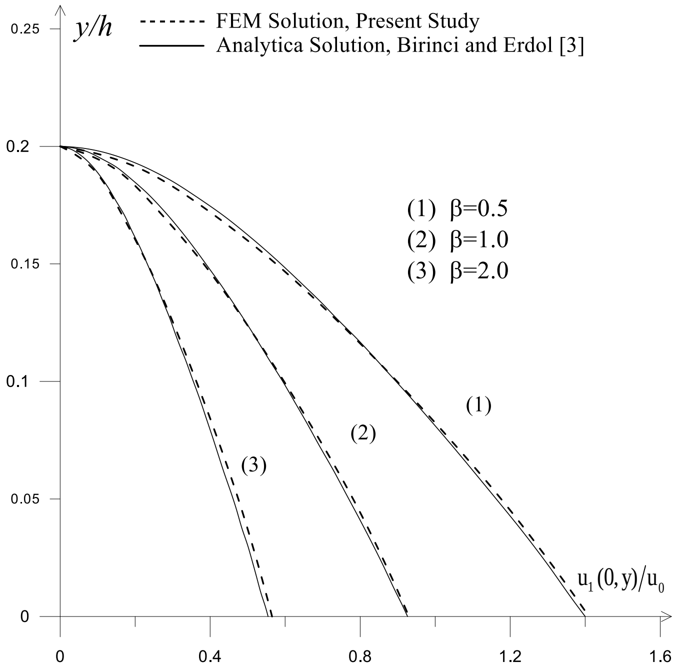


Fig. 16 Comparisons of analytical solution with FEM solution for the normalized crack surface displacement for various of β for the case of an internal crack ($a/h = 0.05$, $h_1/h = 0.5$, $b/h = 1$, $d/h = 0.2$)

that obtained from the FEM simulations to check the validity of the assumptions. Fig. 9-16 illustrates this comparison for various values (b/h), (d/h), (c/h) and β. The analytical normalized stress axial stress, stress-intensity factors and crack surface displacement has been received from literature Birinci and Erdol (2004) for the problem. It is interesting to see that the assumed axial stress, stress-intensity factors and crack surface displacement results are in very good agreement with the results obtained from the FEM simulations.

Fig. 9 shows the distribution of the normalized axial stress $\sigma_x(0,y)$ in the layered composite without a crack for various values of width of the support (b/h). Figs. 10-12 show the variations of the normalized stress-intensity factors with the crack length for internal crack case. Variation of normalized stress-intensity factor $k(d)$ with d/h for various values of the case of the edge crack is shown in Fig. 13. Figs. 14 and 15 show normalized crack surface displacement, $u_1(0,y)/u_0$, for various crack length $(d-c)$ for the case of internal crack. Normalized crack surface displacement for the edge crack is shown in Fig. 16 for values of β.

4. Conclusions

This paper presents a finite element method for calculating stress intensity factors for crack problems in a elasticity layer. The results of the obtained FEM are included to show that the method is very efficient and accurate for calculating stress intensity factors of crack problem. From the FEA analysis is concluded that the exactly results of stresses intensity factors can be theoretically estimated. The engineer should always get in the mind that materials can contain cracks which are very large collections of mechanical stresses. The cracks can cause the state of unrepaired even though the element will be designed very well in conditions of the transfer ability. So the parts can be destroyed by fragile fracture even before some mark of plastic deformation will be able to clear. By the analytical method of the calculation of cracks parameters may be solved very exactly. Numerical approach to the calculation of the stress intensity factors is that a shape and position of cracks in a model are not limited. Solving of complicated cracks in a complex shaped model through the analytical approach is almost very inefficient in the terms of the time. From the obtained results can be seen that the numerical results give very exactly data if their compare with the analytically obtained results. Thus the numerical approach can be used for the solving of crack problem with very well accuracy of results. The numerical results had the deviate from the analytics results in the range very low.

References

Adıyaman, G., Yaylacı, M. and Birinci, A. (2015), "Analytical and finite element solution of a receding contact problem", *Struct. Eng. Mech.*, **54**(1), 69-85.

Altunsaray, E. and Bayer, I. (2013), "Deflection and free vibration of symmetrically laminated quasi-isotropic thin rectangular plates for different boundary conditions", *Ocean Eng.*, **57**, 197-222.

ANSYS (2013), Swanson Analysis Systems Inc., Houston, PA, USA.

Birinci, A. and Erdol, E. (2004), "A layered composite containing a crack in its lower layer loaded by a rigid stamp", *Eur. J. Mech. A/Solid.*, **23**, 909-924.

Birinci, A., Adıyaman, G., Yaylacı, M. and Öner, E. (2015), "Analysis of continuous and discontinuous cases of a contact problem using analytical method and FEM", *Latin Am. J. Solid. Struct.*, **12**(9), 1771-1789.

Bogdanski, S. (2002), “A rolling contact fatigue crack driven by Squeeze fluid film”, *Fatig. Fract. Eng. Mater. Struct.*, **21**, 1061-71.
Chaari, F., Fakhfakh, T. and Haddar, M. (2009), “Analytical modelling of spur gear tooth crack and influence on gear mesh stiffness”, *Eur. J. Mech. A/Solid.*, **28**, 461-468.
Delpero, T., Lepoittevin, G. and Sanchez, A. (2010), “Finite element modeling with ANSYS, centre of structure technologies”, Swiss Federal Institute of Technology Zurich Swiss.
Farjoo, M., Danie, l W. and Weehan, P.A. (2014), “Modeling a squat form crack on a rail laid on an elastic foundation”, *Eng. Fract. Mech.*, **85**, 47-58.
Figiel, L., Kaminski, M. and Lauke, B. (2004), “Analysis of a compression shear fracture test for curved interfaces in layered composites”, *Eng. Fract. Mech.*, **71**, 967-980.
Hadju, **Š.** (2014), “The investigation of the stress state near the crack tip of central cracks through numerical analysis”, *Procedia Eng.*, **69**, 477-485.
Isida, M. (1970), “On the determination of stress intensity factors for some common structural problems”, *Eng. Fract. Mech.*, **2**, 61-79.
Ji, J., Zhang, W., Zhao, W., Yuan, C. and Liu, Y. (2012), “Research of seismic testing and dynamic character of high-rise building structure based on ANSYS”, *Int. J. Digital Cont. Tech. Its Appl.*, **6**(8), 63-71.
Khan, D., Bhushan, A., Panda, S.K. and Biswas, K. (2013), “Elastic-plastic dynamic fracture analysis for stationary curved cracks”, *Finite Elem. Anal. Des.*, **73**, 55-64.
Lee, J. (2009), “Identification of multiple cracks in a beam using vibration amplitudes”, *J. Sound Vib.*, **326**, 205-212.
Leverenz, R.K. (1972), “A finite element stress analysis of a crack in a bi-material plate”, *Int. J. Fract. Mech.*, **8**, 311-24.
Oner, E., Yaylacı, M. and Birinci, A. (2015), “Analytical solution of a contact problem and comparison with the results from FEM”, *Struct. Eng. Mech.*, **54**(4), 607-622.
Pandya, Y. and Parey, A. (2013), “Failure path based modified gear mesh stiffness for spur gear pair with tooth root crack”, *Eng. Fail. Anal.*, **27**, 286-96.
Papadopoulos, G.A., Badalouka, V. and Souyıannıs, J. (2008), “Experimental study of the reduction at crack-tip stress intensity factor K_I by bonded patches”, *Lett. Fract. Micromech.*, **149**, 199-205.
Qiao, P. and Wang, J. (2004), “Partial Mechanics and fracture of crack tip deformable bi-material interface”, *Int. J. Solid. Struct.*, **41**, 7423-7444.
Shao, R., Dong, F., Wang, W. and Jia, P. (2013), “Influence of cracks on dynamic characteristics and stress intensity factor of gears”, *Eng. Fail. Anal.*, **32**, 63-80.
Sabliov, C.M., Salvi, D.A. and Boldor, D. (2007), “High frequency electromagnetism, heat transfer and fluid flow coupling in ANSYS multiphysics”, *IMPI*, **4**(4), 5-17.
Yaylacı, M., Ö ner, E. and Birinci, A. (2014), “Comparison between analytical and ANSYS calculations for a receding contact”, *Am. Soc. Civil Eng.*, ASCE, **140**(9), 71-80.
Zandi, Y. and Akpinar, M.V. (2012), “Evaluation of internal resistance in asphalt concretes”, *Int. J. Concrete Struct. Mater.*, **6**(4), 247-250.

CC