J. Comput. Appl. Res. Mech. Eng. Vol. 11. No. 1, pp. 139-149, 2021 *DOI: 10.22061/JCARME.2019.5081.1622*

SRTTU
Journal of Computational and Applied Research in Mechanical Engineering
jcarme.sru.ac.ir
JCARME
ISSN: 2228-7922

Research paper

Finite element simulation of crack growth path and stress intensity factors evaluation in linear elastic materials

Abdulnaser M. Alshoaibi* and Omar G. M. Yasin

Department of Mechanical Engineering, Jazan University, P. O. Box 706, Jazan 45142, Kingdom of Saudi Arabia

Article info:

Article history:

Received: 11/04/2019
Revised: 22/10/2019
Accepted: 25/10/2019
Online: 26/10/2019

Keywords:

Finite element method,
Franc2D/L program,
ANSYS mechanical software,
Fatigue analysis,
Linear elastic fracture mechanics.

***Corresponding author:**
alshoaibi@gmail.com

Abstract

This paper proposes a combination of FRANC2D/L (2D crack growth simulation program) and ANSYS mechanical program (3D structural analysis for fracture mechanic analysis). The comparisons between the two software are performed for different case studies for stress intensity factors (SIFs) and crack growth trajectory. Crack growth is numerically simulated by a step-by-step 3D and 2D finite element method. The SIFs are calculated by using the displacement correlation technique. The procedure consists of computing SIFs, the crack growth path, stresses, and strain distributions via an incremental analysis of the crack extension, considering two and three-dimensional analysis. The finite element analysis for fatigue crack growth is performed for both software based on Paris's law, and the crack orientation is determined using maximum circumferential stress theory. The simulation results obtained in this study, using the finite element method, provide a good agreement with experimental results for all the case studies reviewed.

1. Introduction

Finite element analysis (FEA) has proved to be a better alternative to testing real-size structures for failures. With the help of numerical methods based on FEM, various parameters of fracture mechanics such as the stress intensity fractures, equivalent von Mises stress, and principal stresses can be computed. One of the most important parameters in fracture mechanics is the stress intensity factor (SIF) which should be accurately calculated. Many SIF calculation handbooks have been published aiming at regular geometry components. However, only a very small number of simple and special fracture problems have an analytical solution, and the vast majority of fracture problems encountered in engineering practice should be resolved with the numerical analysis method [1]. In cases where experimental work is not practical, numerical analysis by FEM can be ideal for finding the fracture mechanics parameters like SIFs. SIFscalculation's numerous handbooks are available [2-5] for specific geometries and loading. Due to the limitation of the analytical solution of SIFs, the vast majority of fracture problems encountered in engineering practice should be resolved with a numerical analysis method [6]. SIFs can be computed by various methods, such the boundary element method (BEM), FEM, both FEM and BEM [7]. Another numerical method used is the peridynamics theory which uses integral or integrodifferential equations instead of partial differential equations (PDEs). When a deformation exists in a system, PDEs cannot be used because they are not structured on variables such as the crack surface. This is where integral equations come to play because they can be used directly. Since peridynamics theory is a nonlocal extension of continuum mechanics, it is compatible with the physical nature of cracks as discontinuities [7-9]. FEM programs such as ANSYS [10, 11], ABAQUS [12, 13], can be used to add elements manually and perform analysis on complex structures. However, there are limitations when it comes to more complicated geometries and loading conditions. This is due to the density of the generated mesh as well as the element type used. Several studies have been conducted for the prediction of fatigue crack growth as well as the evaluation of stress intensity factors in two dimensional components [14-17]. Other researchers conducted a numerical analysis for fatigue crack growth in gear using ANSYS and FRANC3D [11, 18] programs. Their investigations show that the mode I, K_I is dominant during the fatigue crack growth in gear tooth pulled by the constant amplitude loading, and K_I raises progressively with the increasing length of the crack. Other researchers developed their own two- dimensional source code program to predict the fatigue crack growth, crack propagation under static loading, and the prediction of SIFs using adaptive mesh strategy [15, 19]. This study presents a comparison study for fatigue crack growth under constant amplitude loading between two-dimensional software (FRANC2D/L) and three-dimensional FEM (ANSYS Mechanical). The fracture parameters such as SIF and crack growth trajectory are compared on different types of modified compact tension geometries. The predicted results are validated by the corresponding experimental values for the crack growth trajectory.

2. Numerical computation and method

2.1. FRANC2D/L software

The software FRANC2D/L is a free two-dimensional fracture analysis, developed by Cornell Fracture Mechanics Group at Cornell University, funded by the U.S. National Science Foundation, NASA, the U.S. Navy, and other agencies [20]. This software is using for modeling, crack propagation, and fatigue crack growth based on linear elastic fracture mechanics assumption. The program is made of two parts: CASCA and FRANC2D/L [20]. The CASCA program is a simple mesh generating program for many types of mesh-like triangular and quadratic (T3, T6, Q4, and Q8). It can be used to generate initial meshes for FRANC2D simulations. After generating the mesh, the FRANC2D/L program uses the CASCA model, including mesh for further preprocessing (load, problem type, materials properties, fixity) and post-processing (crack definition, crack growth criteria, SIFs, fracture mechanics parameters, stresses, and strain distribution).

In the Franc2D program, there are three methods for computing SIFs along the crack path, these methods include displacement correlation technique (DCT), potential energy release rate, which is computed by a modified crack-closure integral technique, J-integral, which is computed with the combination of equivalent domain integral (EDI) and a decomposition scheme [21]. All three methods produce almost the same results. In the present study, the SIFs are evaluated by using the DCT method. The analysis of a given geometry is divided into two parts. The first part creates and builds the mesh using CASCA [20]. It is a pre-processor for the Franc2D program [22, 23]. This doesn't limit the Franc2D program because other mesh generators can be used and translated to be used in Franc2D program. The second part is using the Franc2D to assign boundary conditions, create the initial crack, propagate cracks and perform stress analysis [24, 25].

2.2. ANSYS mechanical software

ANSYS Mechanical Software is a mechanical engineering software solution that uses FEA for structural analysis using the ANSYS Mechanical interface [26]. It covers an enormous range of applications and comes complete with everything you need from geometry preparation to optimization and all the steps in between. Ansys Mechanical implements a new feature namely Smart Crack Growth, for the analysis of the SIF, crack propagation, number of cycles, etc. [27, 28]. In this feature, there are two types of model to determine the crack growth, static, and fatigue. Static Mode uses a failure criterion option of either the critical rate of the SIF or J-integral. Fatigue uses the Paris' law to predict the crack path. The Paris' law [29] is expressed as:

$$\frac{da}{dN} = C\Delta K^m \qquad (1)$$

where da/dN is the crack growth rate, C and m are the material characteristics and ΔK is the range of the SIF during a fatigue cycle.

2.3. Mesh validation

The mesh structure dictates the accuracy of the results. It also is a deciding factor for the success and failure of a simulation. The Franc2D/L Software requires the use of an external program to generate a mesh on the structure. CASCA in this case is used to manually create a quadratic 8 nodes mesh by setting the numbers of nodes on each edge of the structure. Due to the limitation of this software, where the number of nodes on each edge exceeded ~50 nodes, the software would crash.

For Ansys Mechanical Program, a tetrahedron mesh is automatically generated with the element size decided by the user. The lower the element size, the denser the mesh, and vice versa.

In the Franc2D/L Software, reducing the number of nodes below 12 results in a completely wrong crack path, and hence wrong SIFs value. There is a negligible difference in the SIFs values when using nodes higher than 13 to 45 nodes. Ansys Mechanical Program acts the same way in terms of accuracy, and when the element size is below the 1 mm, the results are identical.

In this study, the meshing is increased with an element size of 0.6 mm in areas surrounding the crack area just to increase the accuracy of the results and reduce the simulation time.

3. Simulation results and discussion

In this section, the crack path and other fracture parameters are computed for some numerical examples.

3.1. Modified compact tension specimen [MCTS]

The modified compact tension specimen geometry and generated mesh in both ANSYS and Franc2D Softwares are shown in Figs. 1 and 2, respectively. The specimen is made from SAE 1020 carbon steel with Young's modulus E = 205 GPa, Poisson's ratio $\nu = 0.33$, yield strength σ_y = 285 Mpa, and tensile strength σ_u = 491 MPa. For the Paris equation, $c = 8.59 \times 10^{-11}$ and $m = 4.26$ with an average load ratio of R = 0.1. Three cases are displayed, in each case, the 7 mm diameter hole will change its position based on the variation of distances A and B, as shown in Fig. 1. Two loads (P) are applied at the upper and lower holes in the opposite direction as a point load (positive y-axis and negative y-axis). The magnitude of the point load is 250N. The number of nodes and elements will vary slightly in each case for both programs.

Case 1

In this case, there exists no thrid hole. As predicted, the crack propagation occurs approximately on a straight line, as seen in Fig. 3. The number of nodes and elements generated on the geometry in the Franc2D program are 4683 nodes and 2220 elements, whereas ANSYS Software generates a mesh with 108349 nodes and 72025 elements.

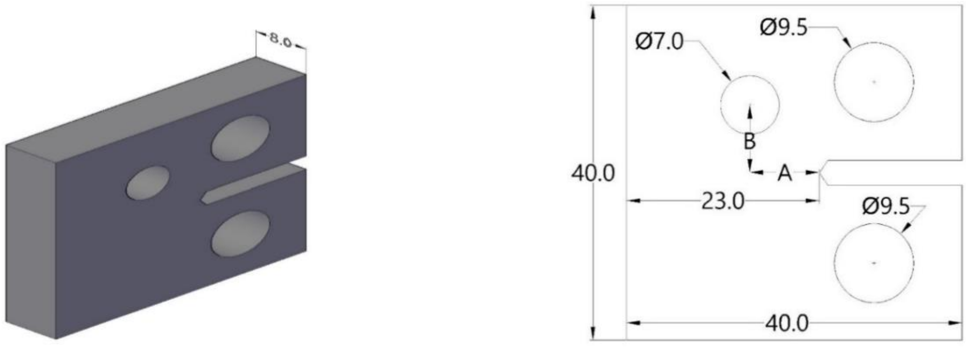
Case 2

Case 2 introduces a thrid hole to the geometry, in which its position according to Fig. 1 is A=8.3 mm and B=8.1 mm. The position of this hole makes the crack propagate towards it but misses it as shown in Fig. 4.

The crack path results are compared to that of experimental and numerical results using Quebra2D code performed by [21]. It can be noticed that the path produced is identical in both cases of simulation compared to the experimental results.

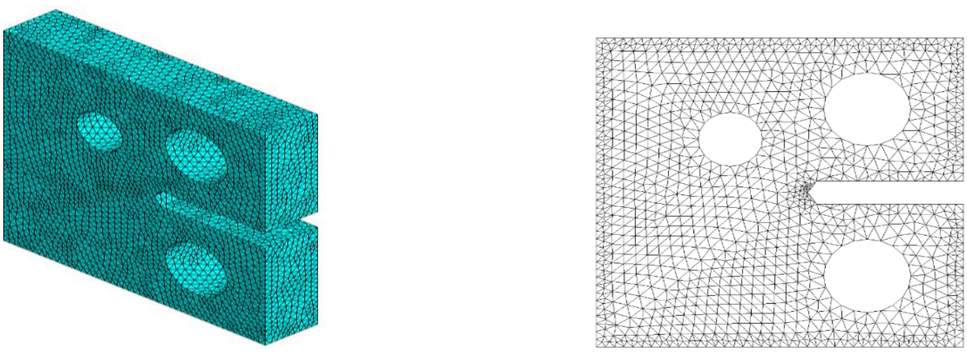
Case 3

In this case the position of the thrid hole, according to Fig. 1, is A=8.1 mm and B=8.1 mm. When the hole is located close to the crack, the crack tends to propagate towards it. As can be seen from Fig. 5, the crack sinks in the hole, and there is a good match with the work of [21] for both experimental and numerical results.



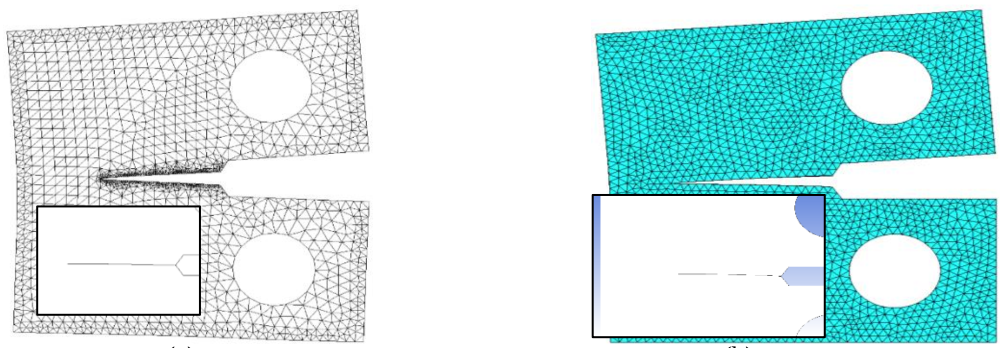
(a) (b)

Fig. 1. Geometric dimensions of the specimen (all dimensions are in mm); (a) 2D and (b) 3D.



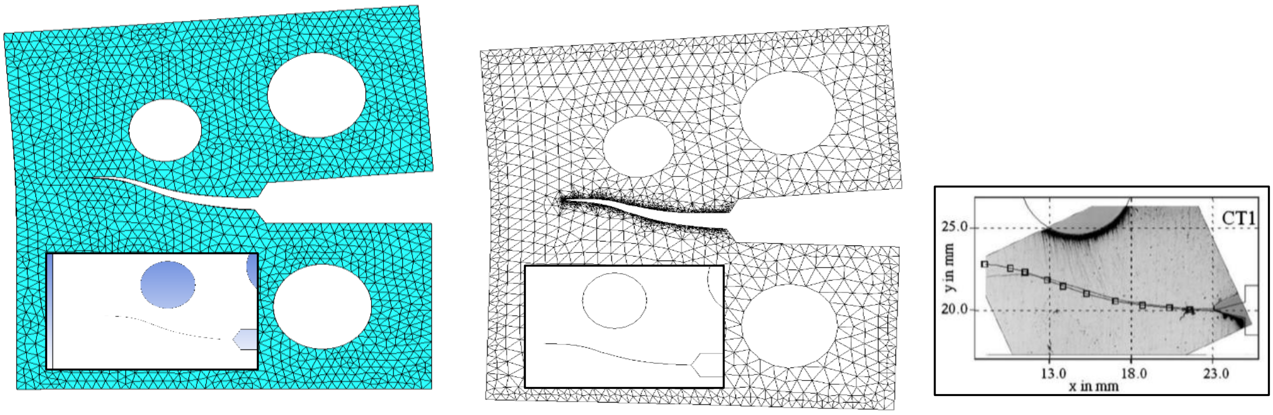
(a) (b)

Fig. 2. Meshed geometry; (a) Ansys Mechanical and (b) the Franc2D.



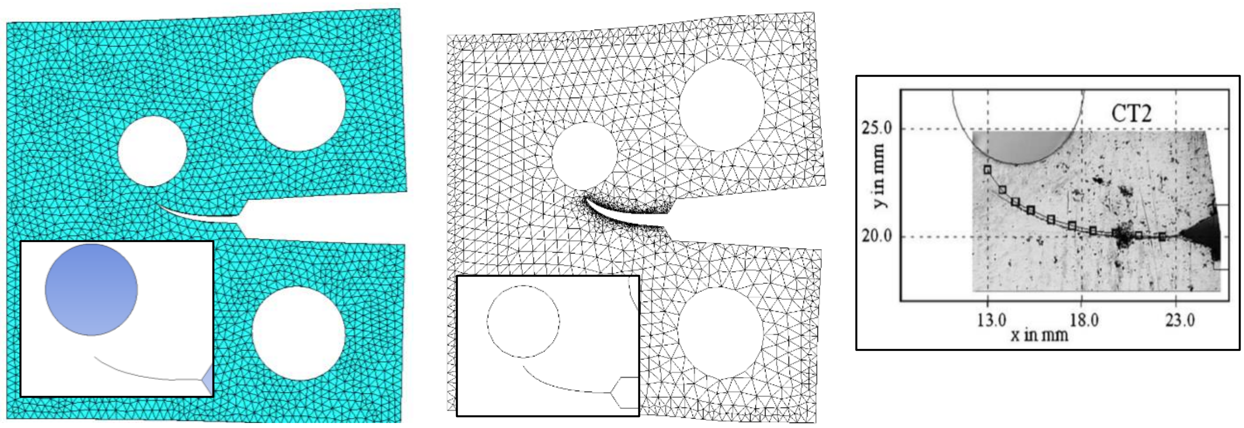
(a) (b)

Fig. 3. Case 1 deformed shape and crack path; (a) Ansys Mechanical software and (b) the Franc2D program.



(a) (b) (c)

Fig. 4. Case 2 deformed shape and crack path; (a) Ansys Mechanical software, (b) Franc2D program, and (c) Quebra 2D with experimental results [20].



(a) (b) (c)

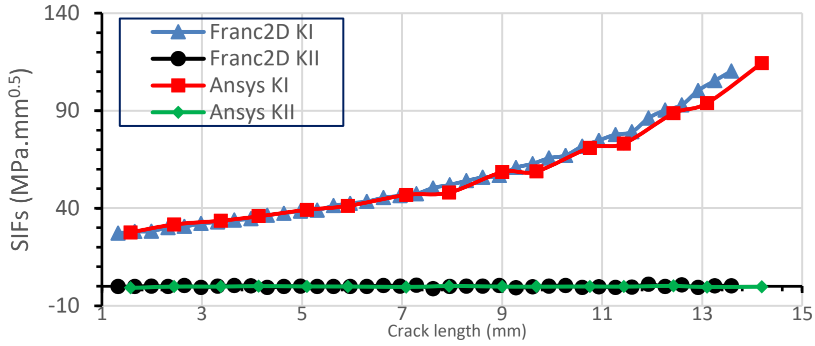
Fig. 5. Case 3 deformed shape and crack path; (a) Ansys Mechanical, (b) Franc2D program, and (c) Quebra 2D with experimental results [20].

The computed SIFs for each of the cases from both programs are compared, as shown in Fig. 6. The values are close in both programs. The mesh size and number of elements generated for the geometries in both software are the reason for the small difference in the values of SIFs. For case 3, the Franc2D generates a total of 4594 nodes and 2166 elements, whereas ANSYS generates a total of 147493 nodes and 99411 elements. Nonetheless, a good argument can be made for computed SIFs from both programs.

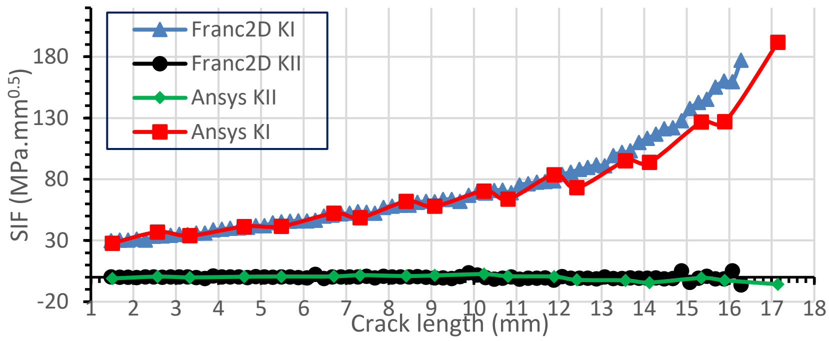
3.2. A cracked plate with four holes [CP4]

A plate rectangular in shape with a size of 100 mm × 100 mm × 1 mm containing 4 holes with a diameter of 10 mm and a crack length of 6 mm at the center edge of the plate, as shown in Fig. 7, is studied. The tensile stress with a magnitude of 10 MPa is applied at the top face of the plate. The plate is made of aluminum 7075-T6 and has Young's modulus of E= 72 GPa, the tensile strength of σ_y = 469 MPa, the ultimate tensile strength of σ_u = 538 MPa, and Poisson's ratio of 0.33. The generated mesh in both ANSYS and Franc2D programs with different mesh densities are shown in Fig. 8.

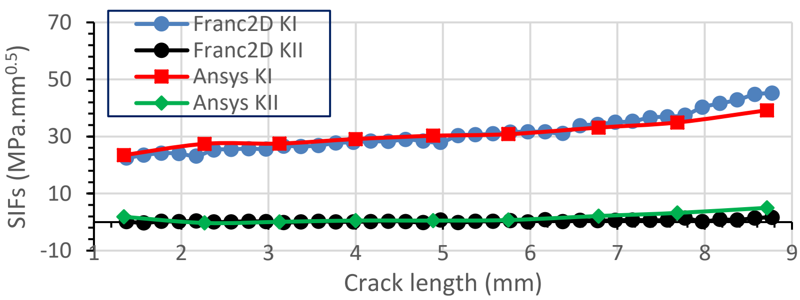
The geometry is fixed in the x and y directions at the bottom face in the FRANC2D/L program while in ANSYS software, the geometry is fixed in the x, y, and z directions, as shown in Fig. 7. The plate is fixed at the bottom in the x and y direction in the FRANC2/L program as well as fixed in the x, y, and z directions in ANSYS software.



(a)



(b)



(c)

Fig. 6. Relationship between the SIFs and crack length; (a) Case 1, (b) Case 2, and (c) Case 3.

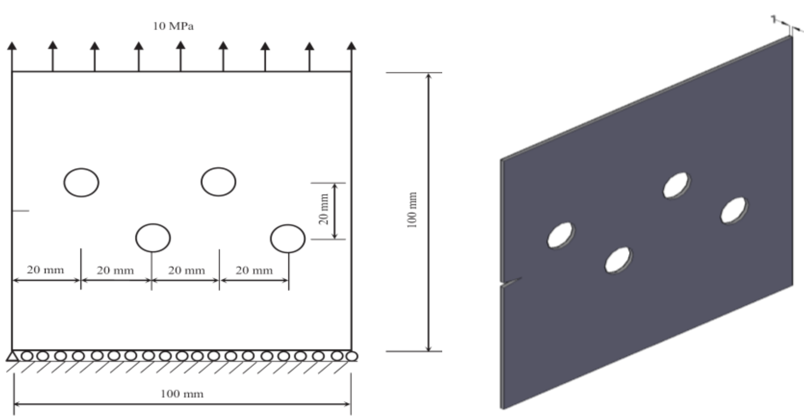
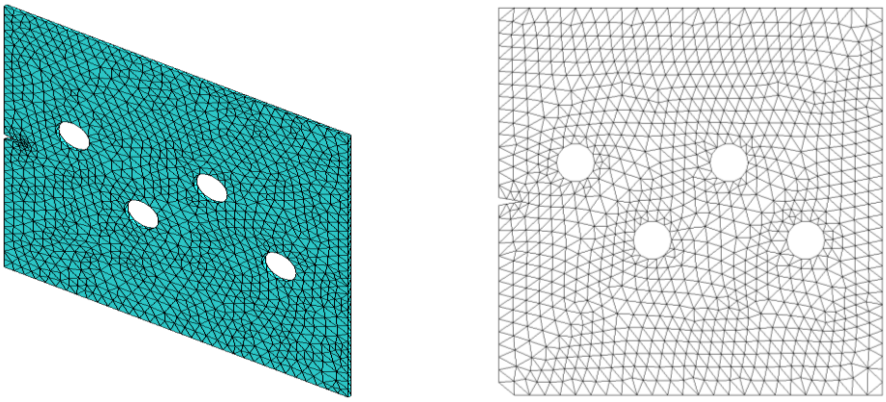
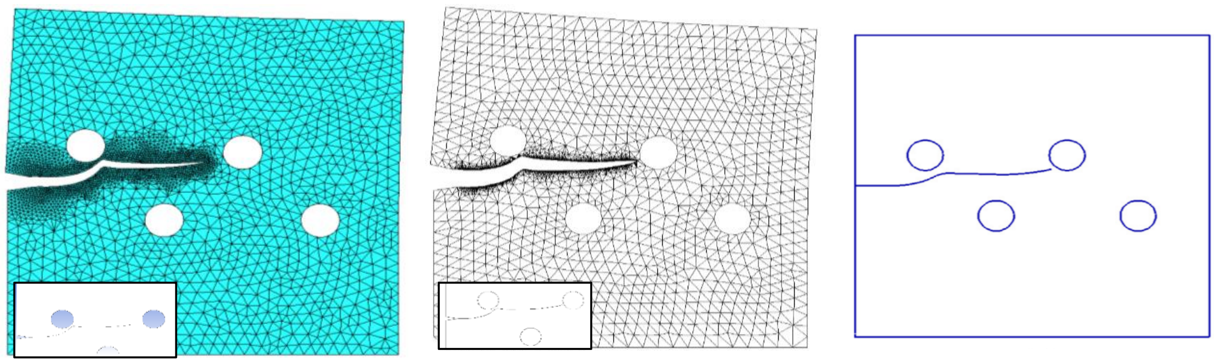


Fig. 7. A cracked plate dimensions with four holes.



(a) (b)

Fig. 8. Generated mesh; (a) Ansys Mechanical software and (b) Franc2D program.



(a) (b) (c)

Fig. 9. Crack propagation and deformed shape; (a) ANSYS Mechanical software, (b) FRANC2D program, and (c) fast multipole BEM [29].

The number of nodes and elements generated in the FRANC2/L program is 3933 nodes and 1911 elements, whereas ANSYS software generates a mesh with 14737 nodes and 7177 elements, as shown in Fig. 8. The crack path produced is identical in both programs, as shown in Fig. 9, and matches the work of [29] by using multipole BEM.

From Fig. 10, it can be seen that the SIF for mode I start to decrease when the crack length is between 15 and 20 mm, and when this happens, the SIF for mode II start to increase; hence, the crack path changes its trajectory and moves upwards. This is due to the influence of the hole on the crack trajectory.

3.3. A cracked plate with three holes [CP3]

Consider a rectangular plate with a dimension of 120 mm × 65 mm × 16 mm, which contains two holes, one with a diameter of 13 mm near both ends of the plate, and the other with a diameter of 20 mm near the center of the plate, as shown in Fig. 11. An initial crack of 10 mm is located at the center edge of the plate.

The plate is made from alumiuim 7075-T6 with Young's Modulus of E = 71.7 GPa, yield strength of σ_y = 469 MPa, ultimate yield strength of σ_u = 538 MPa, Paris equation parameters of c = 0.527 × 10^{-11} and m = 2.947, fracture toughness of K_{IC} = 938.25 MPa .$mm^{0.5}$, and a Poisson's ratio of 0.33.

In the FRANC2D/L program, the mesh is generated with 9104 nodes and 4378 elements, whereas for ANSYS software, the mesh element size is set at 2.5 mm, hence generating a mesh of 215212 nodes and 145624 elements, as shown in Fig. 12.

Fig.13 shows the deformed geometry done using both software as well as the crack path predicted during the simulation. The predicted crack path is almost identical to the experimental crack path in the experimental work performed by [30]. As shown in Fig. 14, the SIFs computed in both Franc2D and ANSYS programs are almost identical, and the small level of error between the values is due to the variation in mesh density.

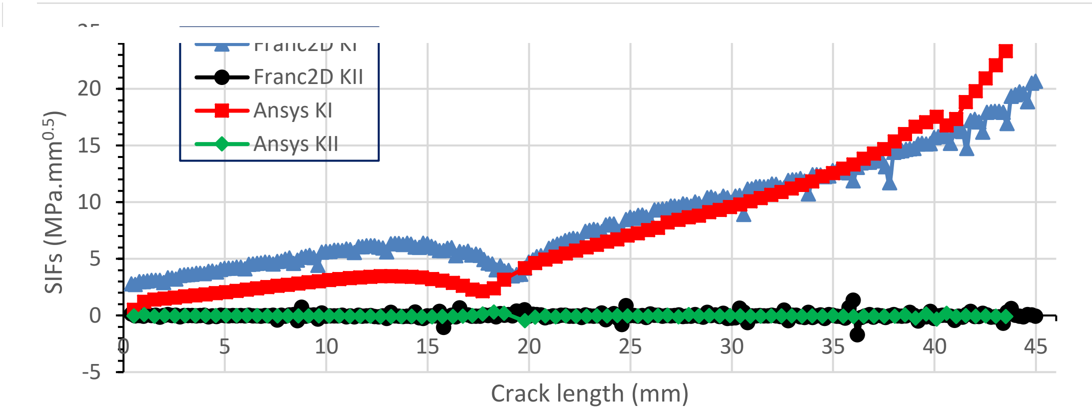


Fig. 10. The variations of SIFs versus crack length.

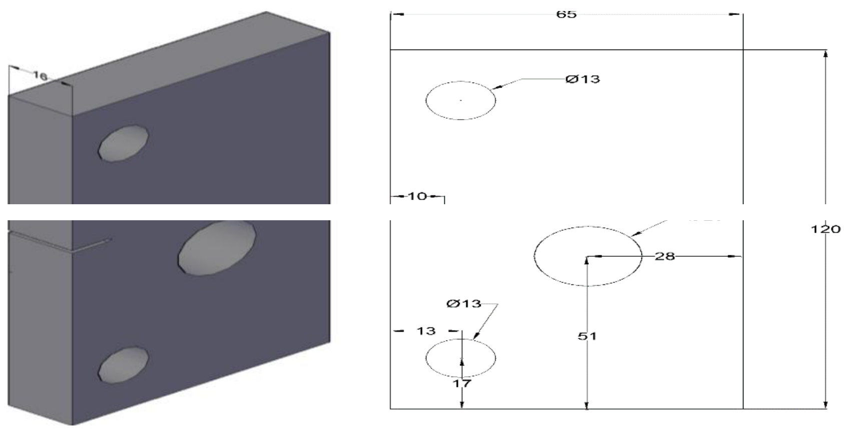
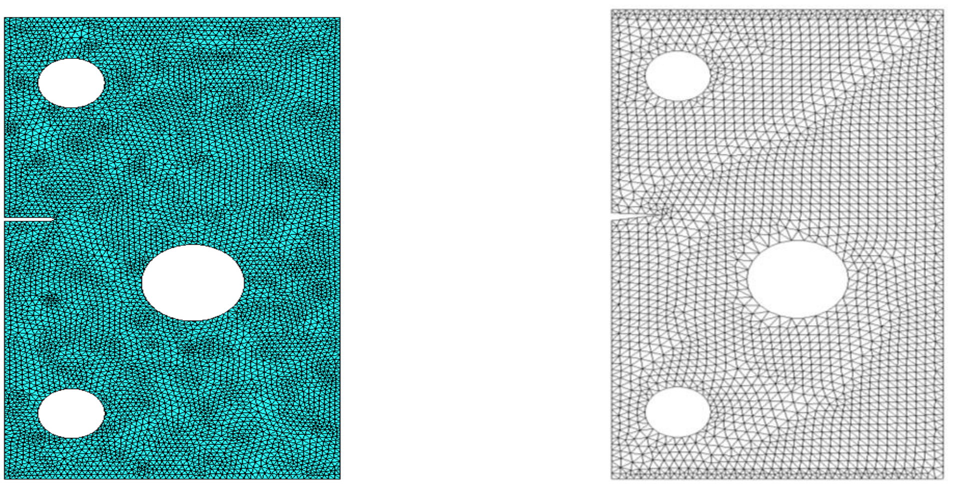
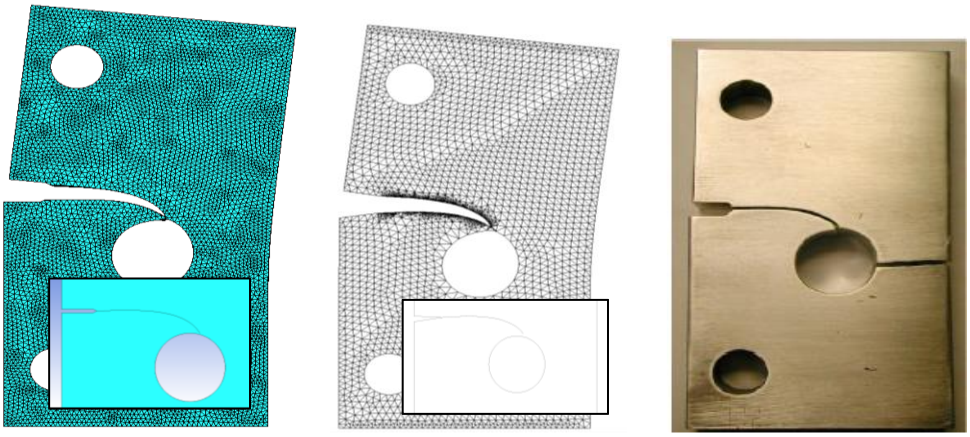


Fig. 11. Geometry of cracked plate with three holes (all dimensions in mm).



(a) (b)

Fig. 12. Generated mesh; (a) ANSYS Mechanical software and (b) the FRANC2D/L program.



(a) (b) (c)

Fig. 13. Deformed shape and crack path; (a) ANSYS Mechancical software, (b) the Franc2D program, and (c) experiemental result [31].

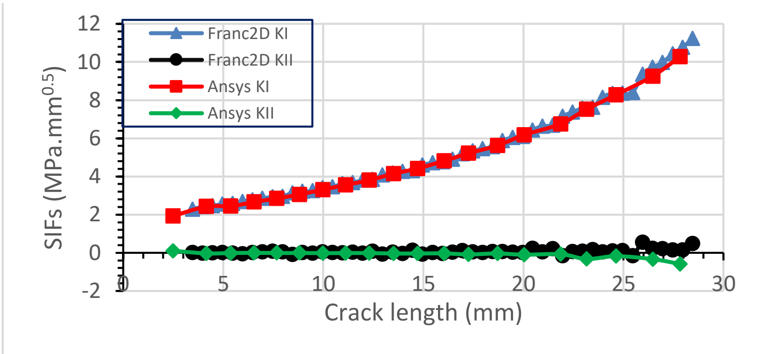


Fig. 14. SIFs relationship versus crack length.

Table 1. Difference between Mode I and simulation durations in geometries modelled in 2D and 3D.

Geometry	Difference Between 2D and 3D in Mode I [%]	Simulation Time in 2D	Simulation Time in 3D
MCTS-Case 1	6.8	< 1 min	~ 30 mins
MCTS-Case 2	7.9	< 1 min	~ 30 mins
MCTS-Case 3	7.4	< 1 min	~ 30 mins
CP4	15.8	< 1min	~ 150 mins
CP3	0.49	< 1 min	~ 240 mins

4. Conclusions

The mesh density is always a factor in obtaining accurate results, as seen in Table 1. From Table 1 it can be concluded that with an extremely high mesh density in 3D and 2D simulated geometries, the differences in SIF, Mode I, is less than 1%. But when the node density is low in both simulations, the accuracry reduces significnatly. This also comes at a cost, simulations in 2D is much faster than simulations in 3D, as seen in Table 1. The time also increases significantly when the mesh density increases in 3D simulations. Sometimes it wouldn't be ideal to perform a simulation on a complex geometry in 2D and hence the need is for 3D. The two-dimensional analysis provides a better computational time than 3D. It also gives the ability to generate a finer mesh on the geometry due to avalable computational power.

References

[1] V. Infante and J. Silva, "Case studies of computational simulations of fatigue crack propagation using finite elements analysis tools", *Eng. Fail. Anal.*, Vol. 18, No. 2, pp. 616-624, (2011).

[2] D. P. Rooke and D. J. Cartwright, Compendium of stress intensity factors. *Procurement Executive, Ministry of Defence. H. M. S. O. 1976, 330 p(Book).* (1976).

[3] H. Tada, P. C. Paris, G. R. Irwin and H. Tada, *The stress analysis of cracks handbook*; ASME press New York: (2000).

[4] Al Laham and S. I. Branch, *Stress intensity factor and limit load handbook*; British Energy Generation Limited Gloucester, UK:, Vol. 3, (1998).

[5] J. E. Srawley, "Wide range stress intensity factor expressions for ASTM E 399 standard fracture toughness specimens", *Int. J. Fract.*, Vol. 12, No. 3, pp. 475-476, (1976).

[6] M. E. Mobasher and H. Waisman, "Adaptive modeling of damage growth using a coupled FEM/BEM approach", *Int. J. Numer. Methods Eng.*, Vol. 105, No. 8, pp. 599-619, (2016).

[7] E. Madenci, E. Oterkus and E. Peridynamic Theory In *Peridynamic Theory and Its Applications*, Springer, pp. 19-43, (2014).

[8] M. Ghajari, L. Iannucci and P. A. Curtis, "Peridynamic material model for the analysis of dynamic crack propagation in orthotropic media", *Comput. Methods Appl. Mech. Eng.*, Vol. 276, pp. 431-452, (2014).

[9] B. Kilic and E. Madenci, "Structural stability and failure analysis using peridynamic theory", *Int. J. Non-Linear Mech.*, Vol. 44, No. 8, pp. 845-854, (2009).

[10] A. Grbović, G. Kastratović, A. Sedmak, I. Balać and M. D. Popović, "Fatigue crack paths in light aircraft wing spars", *Int. J. Fatigue*, Vil. 123, pp. 96-104, (2019).

[11] Y. Liao, Y. Li, M. Huang, B. Wang, Y. Yang and S. Pei, "Effect of hole relative size and position on crack deflection angle of repaired structure", *Theor. Appl. Fract. Mech.*, Vol. 101, pp. 92-102, (2019).

[12] M. Naderi and N. Iyyer, "Fatigue life prediction of cracked attachment lugs using XFEM", *Int. J. Fatigue.*, Vol. 77, pp. 186-193, (2015).

[13] A. Sedmak, "Computational fracture mechanics: An overview from early efforts to recent achievements", *Fatigue Fract. Eng. Mater. Struct.*, Vol. 41, No. 12, pp. 2438-2474, (2018).

[14] X. Zhang, L. Li, X. Qi, J. Zheng, X. Zhang, B. Chen, J. Feng and S. Duan, "Experimental and numerical investigation of fatigue crack growth in the cracked gear tooth", *Fatigue Fract. Eng. Mater. Struct.*, Vol. 40, No. 7, pp. 1037-1047, (2017).

[15] A. M. Alshoaibi, "Finite element procedures for the numerical simulation of fatigue crack propagation under mixed mode loading", *Struct. Eng. Mech.*, Vol. 35, No. 3, pp. 283-299, (2010).

[16] A. A. Alshoaibi and A. K. Ariffin, "Evaluation of stress intensity factor using displacement correlation techniques", *Jurnal Kejuruteraan.* Vol. 20, pp. 75-81, (2008).

[17] A. M. Alshoaibi, "An Adaptive Finite Element Framework for Fatigue Crack Propagation under Constant Amplitude Loading", *Int. J. Appl. Sci. Eng.*, Vol. 13, pp. 261-270, (2015).

[18] T. Ingrafea, "FRANC3D, 3D fracture analysis code", *The Cornell University Fracture Group, Cornell University, Ithaca, NY*, (1996).

[19] A. M. Alshoaibi, "A Two Dimensional Simulation of Crack Propagation using Adaptive Finite Element Analysis", *J. Comput. Appl. Mech.*, vol. 49, No. 2, pp.335-341, (2018).

[20] Cornell Fracture Group, F. D. V., http://www.cfg. cornell.edu.

[21] A. C. O. Miranda, M. A. Meggiolaro, J. T. P. Castro, L. F. Martha and T. N.

Bittencourt, "Fatigue life and crack path predictions in generic 2D structural components", *Eng. Fract. Mech.*, Vol. 70, No. 10, pp. 1259-1279, (2003).

[22] A. Ural, "*Modeling and simulation of fatigue crack growth in metals using LEFM and a damage-based cohesive model*", Cornell University, (2004).

[23] J. Hochhalter, "*Finite element simulations of fatigue crack stages in AA 7075-T651 microstructure*". (2010).

[24] A. M. Al-Mukhtar and B. Merkel, "Simulation of the crack propagation in rocks using fracture mechanics approach", *J. Fail. Anal. Prev.*, Vol. 15, No. 1, pp. 90-100, (2015).

[25] B. R. Davis, *"Three-dimensional simulation of arbitrary crack growth"*; Cornell University: (2014).

[26] T. Bittencourt, P. Wawrzynek, A. Ingraffea and J. Sousa, " Quasi-automatic simulation of crack propagation for 2D LEFM problems". *Eng. Fract. Mech.*, Vol. 55, No. 2, pp. 321-334, (1996).

[27] M. F. Yaren, O. Demir, A. O. Ayhan and S. İriç, "Three-dimensional mode-I/III fatigue crack propagation: Computational modeling and experiments", *Int. J. Fatigue.*, Vol. 121, pp. 124-134, (2019).

[28] A. Kotousov, F. Berto, P. Lazzarin and F. Pegorin, "Three dimensional finite element mixed fracture mode under anti-plane loading of a crack", *Theor. Appl. Fract. Mech.*, Vol. 62, pp. 26-33, (2012).

[29] F. Erdogan and G. Sih, "On the crack extension in plates under plane loading and transverse shear", *J. basic Eng.*, Vol. 85, pp. 519-525, (1963).

[30] E. Giner, N. Sukumar, J. E. Tarancón and F. J. Fuenmayor, "An Abaqus implementation of the extended finite element method", *Eng. Fracture Mech.*, Vol. 76, No. 3, pp. 347-368, (2009).

[31] Y. J. Liu, Y. X. Li and W. Xie, "Modeling of multiple crack propagation in 2-D elastic solids by the fast multipole boundary element method", *Eng. Fract. Mech.*, Vol. 172, pp. 1-16, (2017).

Copyrights ©2021 The author(s). This is an open access article distributed under the terms of the Creative Commons Attribution (CC BY 4.0), which permits unrestricted use, distribution, and reproduction in any medium, as long as the original authors and source are cited. No permission is required from the authors or the publishers.

How to cite this paper:

Abdulnaser M. Alshoaibi and Omar G. M. Yasin, "Finite element simulation of crack growth path and stress intensity factors evaluation in linear elastic materials,", *J. Comput. Appl. Res. Mech. Eng.,* Vol. 11, No. 1, pp. 139-149, (2021).

DOI: 10.22061/JCARME.2019.5081.1622

URL: https://jcarme.sru.ac.ir/?_action=showPDF&article=1161