Comparative Analysis of 2D and 3D Software for Crack Growth in Linear Elastic Materials

Mohammed Y. Tharwan*

Mechanical Department, College of Engineering, Jazan university, Jazan, Saudi Arabia

* *Corresponding author; mtharwan@jazanu.edu.sa*

Abstract:
In this study, a comparison was made between two finite element programs, ANSYS Workbench 19.2 and FRANC2D/L for analyzing crack growth under linear elastic fracture mechanics. The study involved the computation of stress intensity factors, crack propagation paths, and fatigue life cycles using ANSYS Workbench 19.2 and FRANC2D/L software in both 2D and 3D finite element simulations. The predicted results for SIFs, fatigue life cycles, and crack path were found to be similar for both software. However, FRANC2D/L had a shorter computational time and was more flexible in terms of mesh generation. Therefore, the study suggests that FRANC2D/L could be a viable alternative to ANSYS Workbench 19.2 for analyzing crack growth problems under linear elastic fracture mechanics, particularly if three-dimensional visualization is not essential. To confirm the validity of the study's findings, reference solutions from existing literature are used for comparison and validation.

Key words: *Finite element analysis; Failure analysis; SIFs; fatigue crack growth; ANSYS; FRANC2D/L.*

1. INTRODUCTION

Cracks are a common type of material defect that can result in disastrous failure if not promptly detected and addressed. Thus, understanding the behavior of cracks is critical in preventing material failure and ensuring safety in various applications. The analysis of crack growth problems often relies on the widely adopted methodology of linear elastic fracture mechanics (LEFM). This approach entails the examination of various factors, including stress intensity factors (SIFs), crack growth paths, and fatigue life. Linear elastic fracture mechanics is a fundamental framework for understanding and analyzing crack propagation in various materials. To gain a comprehensive understanding of this field, researchers and engineers rely on a range of important references (Pook 2000, Lee, Pan et al. 2005, Broek 2012, Anderson 2017). When it comes to studying crack growth problems, finite element methods have gained significant popularity due to their effectiveness in conducting numerical simulations (Bouchard, Bay et al. 2000, Kim, Yang et al. 2006, Duchêne, El Houdaigui et al. 2007, De Jesus, da Silva et al. 2011). ANSYS Workbench 19.2 is a widely recognized software extensively employed for such analyses. In contrast, FRANC2D/L is a freely available code developed by Cornell University, specifically designed for fracture analysis with a foundation in the finite element method (FEM). This software possesses numerous advantages, including computational efficiency and great flexibility in mesh generation.

Consequently, the study suggests that FRANC2D/L could serve as a feasible alternative to ANSYS Workbench 19.2 for examining crack growth problems within the scope of linear elastic fracture mechanics. Notably, this alternative may be particularly suitable when three-dimensional visualization is unnecessary. The evaluation of fatigue crack growth is a crucial procedure to guarantee the safety and dependability of structures operating under fatigue loading conditions. This assessment encompasses determining the direction of crack propagation, the range of the equivalent stress intensity factor *(ΔKeq)*, and the rate at which the crack grows per cycle number (*da/dN*) (Suresh 1998, Zhan, Hu et al. 2017). The comparison of ΔKeq to material characteristics, such as fracture toughness or threshold stress intensity factor, is essential in predicting the failure of a structure or component due to crack growth under static or fatigue loading. Stress intensity factors play a critical role in assessing the behavior of cracks during their initiation and propagation stages. They serve as fundamental parameters in evaluating the potential failure mechanisms (Sih and Liebowitz 1968, Hellan 1985, Barsom and Rolfe 1999, Broek 2012). However, the determination of SIFs is not straightforward, especially when the crack is in a complex state. This level of complexity increases when the crack front is not planar, or when the crack experiences loading conditions that involve a combination of modes. In such cases, the determination of SIFs requires more advanced methods, such as numerical simulations using FEM.

The study of crack growth problems, specifically for long cracks exhibiting small-scale yielding features at the crack front (known as the Paris regime), often relies on the widely adopted approach of LEFM, which offers valuable insights into the understanding and prediction of crack growth phenomena in the Paris regime. The Paris regime refers to the stage in which the crack grows at a constant rate under cyclic loading conditions (Paris, Tada et al. 1999, Bang, Ince et al. 2019). Predicting the path of crack propagation is vital to guarantee the reliability of components in the field of engineering. However, conducting full-scale investigations to estimate fatigue can be prohibitively expensive.

As a cost-effective alternative, accurate estimation methods are necessary to calculate crack growth trajectory and fatigue analysis under static and cyclic loading (Demir, Ayhan et al. 2017, Dirik and Yalçinkaya 2018, Zhang and Guo 2018). Simulating fatigue crack growth requires the use of various numerical methods, including the phase-field method (Zhang and Tabiei 2020), Elements Free Galerkin method (Kanth, Harmain et al. 2018), extended finite element method (Belytschko and Black 1999, Rozumek, Marciniak et al. 2017, Huynh, Nguyen et al. 2019, Surendran, Natarajan et al. 2019), cohesive elements method (Rezaei, Wulfinghoff et al. 2017, Dekker, van der Meer et al. 2019), and Discrete Elements Method (Leclerc, Haddad et al. 2017, Shao, Duan et al. 2019, Li, Li et al. 2020). Each approach possesses its own distinct advantages and limitations, and the selection of a particular method depends on the specific demands of the analysis. Factors such as material properties, loading conditions, and crack geometry play a significant role in determining the most suitable approach for the given scenario. It is crucial to carefully consider these factors when choosing the appropriate method for crack analysis to ensure accurate and reliable results.

To optimize laboratory resources, minimize time requirements, and reduce costs associated with analyzing fatigue crack problems, an efficient approach is to employ a simulation methodology that integrates two- and three-dimensional numerical analysis tools, such as FRANC2D/L and ANSYS. This combined approach enables the simulation of both simple and complex geometries in both 2D and 3D analyses. It has gained widespread adoption in various computational techniques used for fatigue crack analysis, offering a streamlined and effective solution for investigating fatigue-related phenomena (Rozumek and Macha 2006, Alizadeh, Hills et al. 2007, Alshoaibi 2010, Dirik and Yalçinkaya 2018, Gomes and Miranda 2018, Alshoaibi 2019, Alshoaibi and Yasin 2019, Chen, Wang et al. 2019, Alshoaibi and Fageehi 2020, Fageehi and Alshoaibi 2020, Fageehi and Alshoaibi 2020).

The aim of this study is to compare the effectiveness of FRANC2D/L software with ANSYS in predicting fatigue crack propagation. The comparison between these two software tools primarily revolves around the accuracy and reliability of SIFs, crack growth paths, and fatigue life prediction. The study's findings will contribute to the development of a more efficient and cost-effective methodology for modeling fatigue crack growth in various industries. The recognition of challenges and considerations involved in using different software tools is an important aspect of research. By highlighting the limitations, complexities, and potential pitfalls associated with various software packages, it contribute to the understanding of the practical implications and limitations of these tools. This information is crucial for researchers and practitioners, enabling them to make informed decisions and avoid potential issues during their analyses.

2. ANSYS PROCEDURE FOR FATIGUE ANALYSIS

In the field of structural mechanics and engineering, accurately predicting the behavior of cracks and their propagation is of paramount importance for ensuring the integrity and safety of various components and structures. ANSYS, a widely renowned software package, has been instrumental in facilitating advanced analysis techniques for crack growth simulations. One of the notable features within ANSYS is the "Smart Crack Growth" analysis tool, which offers enhanced capabilities for modeling and simulating crack propagation. The "Smart Crack Growth" analysis tool in ANSYS provides engineers and researchers with a powerful solution for accurately simulating the growth and behavior of cracks. Unlike traditional methods, which often require extensive manual meshing and remeshing to account for crack propagation, the "Smart Crack Growth" tool automates this process, significantly streamlining crack growth simulations. By utilizing the SMART capabilities, ANSYS can provide an efficient and effective tool for simulating and analyzing 3D crack growth in a variety of materials and structures.

The analysis of crack growth was performed using the maximum circumferential stress criterion as the selected approach (Erdogan and Sih 1963). This criterion assumes that cracks propagate in the direction of maximum circumferential stress. According to this criterion, the angle of crack growth is determined by the following formula (Bjørheim 2019, ANSYS 2020):

$$\theta = \cos^{-1}\left(\frac{3K_{II}^2 + K_I\sqrt{K_I^2 + 8K_{II}^2}}{K_I^2 + 9K_{II}^2}\right) \tag{1}$$

where: K_I and K_{II} denote the first and second mode of SIFs.
The crack propagation rate (*da/dN*) can be predicted using a modified formula of the Paris law as (PC 1961):

$$\frac{da}{dN} = C(\Delta K_{eq})^m \tag{2}$$

where: ΔK_{eq} is corresponds to the equivalent SIF while C and m represent the Paris' law coefficient and exponent in Paris' law, respectively. The expression for the equivalent range of SIF formula utilized in both programs can be represented as follows (Bjørheim 2019):

$$\Delta K_{eq} = \frac{1}{2}\cos\frac{\theta}{2}\left[(\Delta K_I(1+\cos\theta)) - 3\Delta K_{II}\sin\theta\right] \tag{3}$$

where: ΔK_I is the range of K_I and ΔK_{II} denotes the range of K_{II}.

The ANSYS process for simulating fatigue crack growth is represented schematically in Figure 1. The smart crack growth analysis using ANSYS involves a systematic procedure that enables engineers to simulate and study the behavior of cracks in structures. The process typically begins with creating a finite element model of the structure, incorporating the necessary material properties and boundary conditions. The crack geometry is then defined, considering its shape, size, and location within the structure. Once the model is set up, the next step involves applying loading conditions to the structure. This can include static or dynamic loads, as well as thermal or mechanical loading. ANSYS provides a wide range of options to accurately represent the loading conditions in the analysis. The crack growth simulation is performed by iteratively advancing the crack incrementally, evaluating the stress intensity factors and determining the crack growth direction. ANSYS offers various crack growth criteria, such as the stress intensity factor-based criterion or the energy release rate-based criterion, which can be selected based on the specific analysis requirements. The software then calculates the updated stress distribution and modifies the mesh accordingly to account for the growing crack. By following this ANSYS procedure for smart crack growth analysis, engineers can gain valuable insights into the behavior of cracks in structures. This enables them to assess the structural integrity, predict the remaining useful life, and make informed decisions regarding maintenance and repair strategies. The advanced capabilities of ANSYS in simulating crack growth provide a powerful tool for optimizing the design and performance of structures in various industries, including aerospace, automotive, and civil engineering.

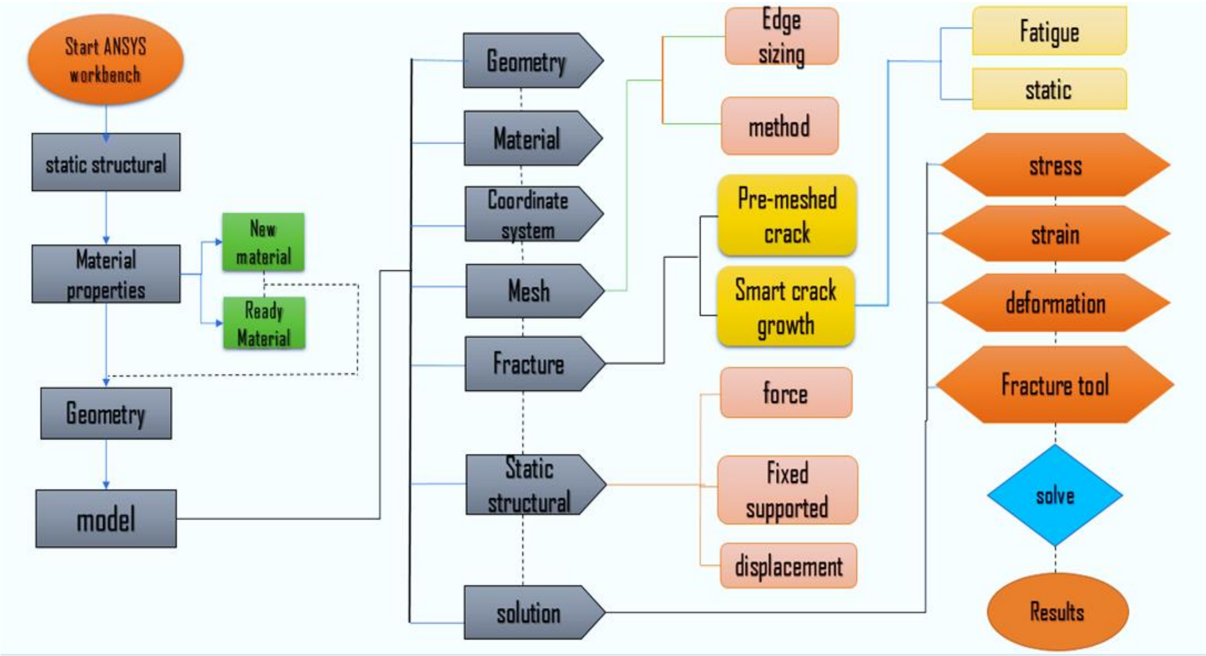


Figure 1. Schematic representation of ANSYS process for fatigue crack growth.

3. FRANC2D/L PROCEDURE FOR FATIGUE ANALYSIS:

FRANC2D/L is a powerful software package that offers the capability to simulate and analyze crack growth in two-dimensional systems. The software employs a finite element method to effectively model the mechanical response of materials when subjected to stress. It has been specifically designed to accommodate a diverse array of materials, such as metals, ceramics, and composites, ensuring its versatility and applicability across various material types. One of the key strengths of FRANC2D/L for crack growth and fracture mechanics is its ability to accurately model the propagation of cracks in materials. Its ability to precisely model the mechanical response of materials under stress, and its advanced post-processing capabilities, make it a valuable asset for anyone interested in understanding the mechanics of materials.

In addition to its crack growth and fracture mechanics capabilities, FRANC2D/L also includes a variety of post-processing tools that can be used to analyze and visualize the results of simulations. These tools include contour plots, vector plots, and stress-strain curves, which can be used to explore the behavior of materials under different loading conditions. FRANC2D/L uses the maximum circumferential stress criterion to predict the direction of crack growth under mixed-mode loading conditions. To enable a more effective comparison between FRANC2D/L and ANSYS, the same criteria were used in both software for evaluating crack growth, SIFs, and fatigue life cycles. By using consistent evaluation methods, researchers can obtain more reliable and accurate results, enabling them to make informed decisions about the performance of different software for simulating crack growth and fatigue life. The FRANC2D/L process for simulating fatigue crack growth is represented schematically in Figure 2.

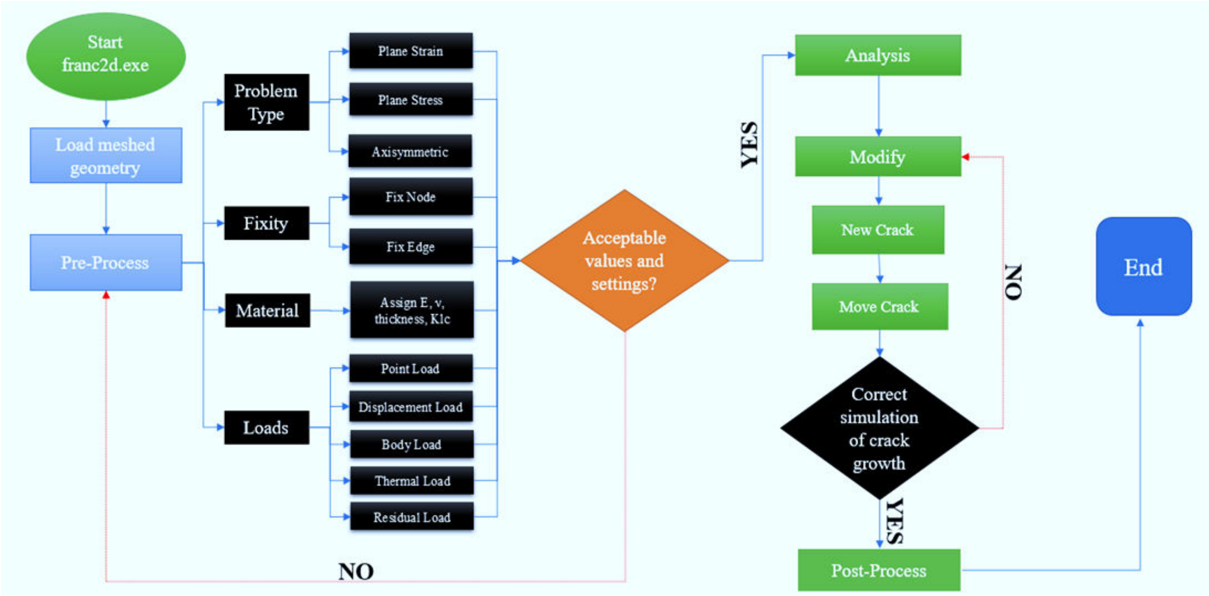


Figure 2. Schematic representation of FRANC2D/L Process for crack propagation analysis.

4. RESULTS AND DISCUSSIONS

4.1. Plate with a Hole and Single Edge Crack:

In this model, a rectangular aluminum 7075-T6 plate, with dimensions of 40 mm x 120 mm x 10 mm with a diameter of 10 and an initial edge crack measuring 6 mm as shown in Figure 3. The specimen exhibited the following material properties: $E =$ 72 GPa, $\nu = 0.33$, $K_{IC} = 29$ MPa m$^{0.5}$, $\sigma_y = 469$ MPa, and σ_u 538 MPa. A tensile load of 10 MPa was applied to the top of the specimen, while the bottom was kept fixed.

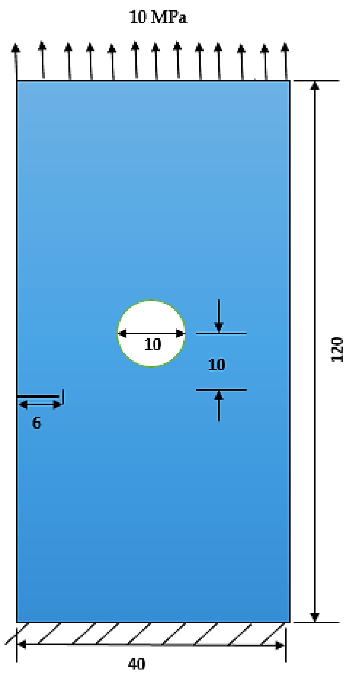
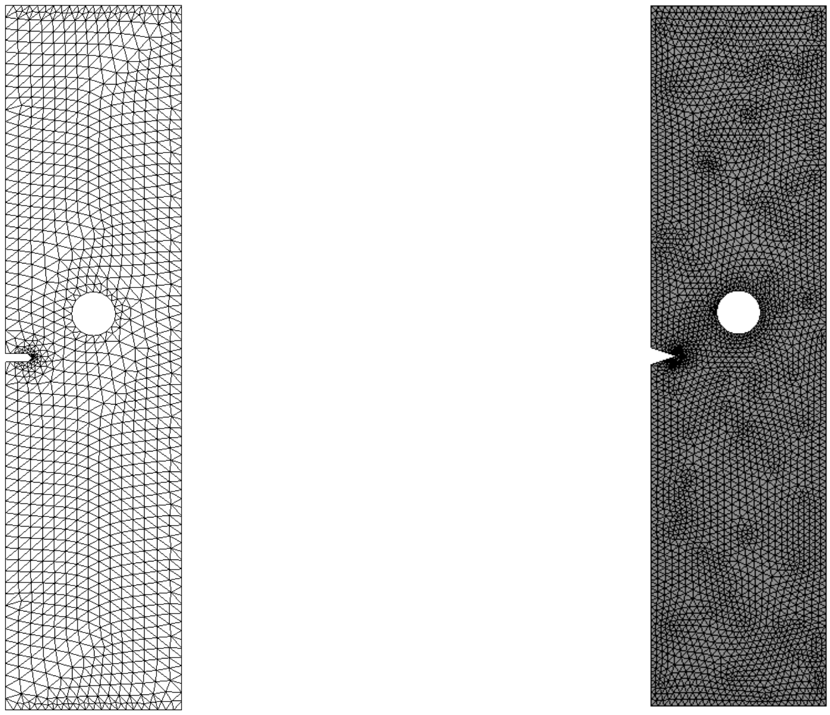


Figure 3. Geometric dimensions of the plate with a hole and single edge crack (dimensions in mm).

For the simulation, Franc2D/L employed a triangular mesh consisting of approximately 5530 nodes and 2657 elements, while ANSYS used an unstructured mesh with the SOLID187 tetrahedral element, generating 45016 nodes and 21216 elements as shown in Figure 4. The unstructured mesh conforms to the surface of the structure and the crack path through the use of tetrahedral elements. By using the higher-order SOLID187 tetrahedral element, ANSYS can accurately model the complex geometries and crack propagation behavior of various materials and structures, making it a popular choice for simulating crack growth.



(a) (b)

Figure 4. Initial mesh: a) Franc2D/L, b) ANSYS.

In both the CASCA program and ANSYS, the initial crack geometry and mesh may slightly differ. This discrepancy arises due to the limitations of the CASCA program, particularly when dealing with small crack fronts. Additionally, the mesh density in CASCA may have limitations compared to the ANSYS program, while both software programs follow similar principles for crack growth analysis It is worth noting that while the initial crack geometry and mesh may differ, the most crucial aspect is that the initial crack length remains the same in both programs.

The study's results obtained from both software were compared to both experimental and numerical data acquired by Varfolomeev et al.(Varfolomeev, Burdack et al. 2014) at a stress ratio R = 0.1. Whereas their numerical results were used the XFEM algorithm in ABAQUS. Additionally, the comparison included Wiragunarsa et al.'s (Wiragunarsa, Zuhal et al. 2021) numerical method, which employed smoothed particle hydrodynamics. Furthermore, the numerical results obtained by Liu et al.(Liu, Li et al. 2017) using fast multipole boundary element method, as depicted in Figure 5, were also considered. The comparison showed clear agreement between the current study's results and all of the aforementioned methods. According to the depiction in Figure 5, the crack initially grwos in a straight path until it approaches the hole. Once the crack reaches near the hole, it changes its direction and starts moving towards it. Due to insufficient proximity, the hole fails to generate enough attraction to draw the crack towards it, causing the crack's movement to be altered and diverted towards a different direction. This diversion may be attributed to the lack of a strong driving force that could pull the crack towards the hole. Therefore, the crack deviates from its initial trajectory and moves towards another direction.

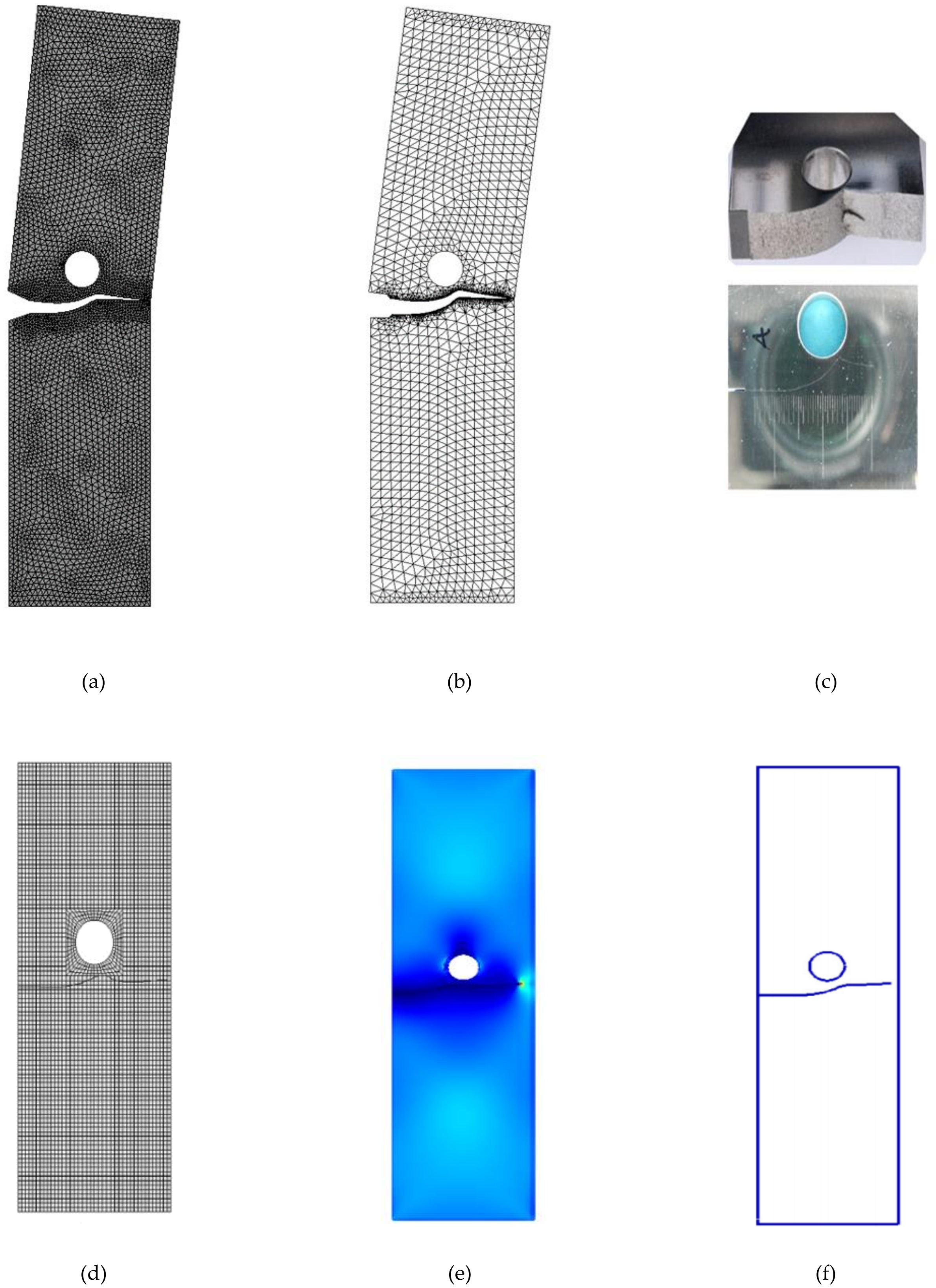


Figure 5. Comparison of crack growth trajectory (a) ANSYS, (b) Franc2D/L, (c) Experimental (Varfolomeev et al.(Varfolomeev, Burdack et al. 2014)), (d) Numerical (Varfolomeev et al.(Varfolomeev, Burdack et al. 2014)), (e) Numerical (Wiragunarsa et al.'s (Wiragunarsa, Zuhal et al. 2021)) and (f) Numerical (Liu et al. (Liu, Li et al. 2017)).

Figure 6 illustrates that the estimated values of K_I and K_{II} obtained from both software, exhibit a substantial level of agreement. Upon conducting an analysis, it was determined that there was a slight reduction in K_I within the range of crack lengths between 15.5 mm and 20 mm. This reduction was attributed to a change in the crack's trajectory towards the hole. However, after this interval, the value of the first mode of SIF started to increase gradually until the last step of the crack growth.

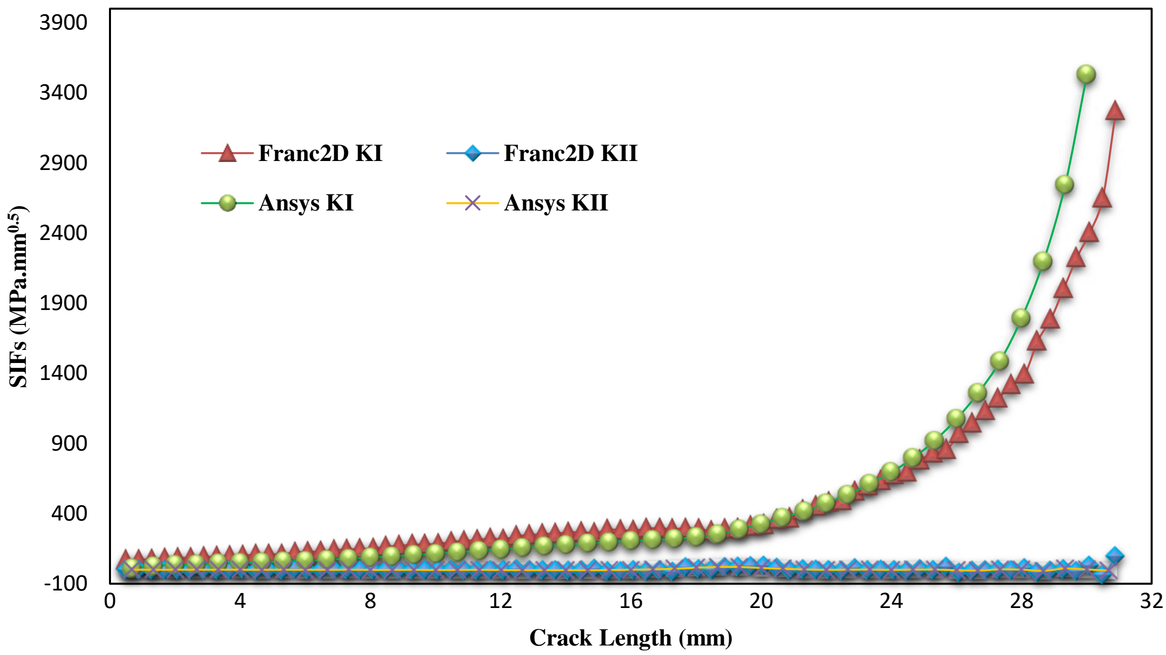
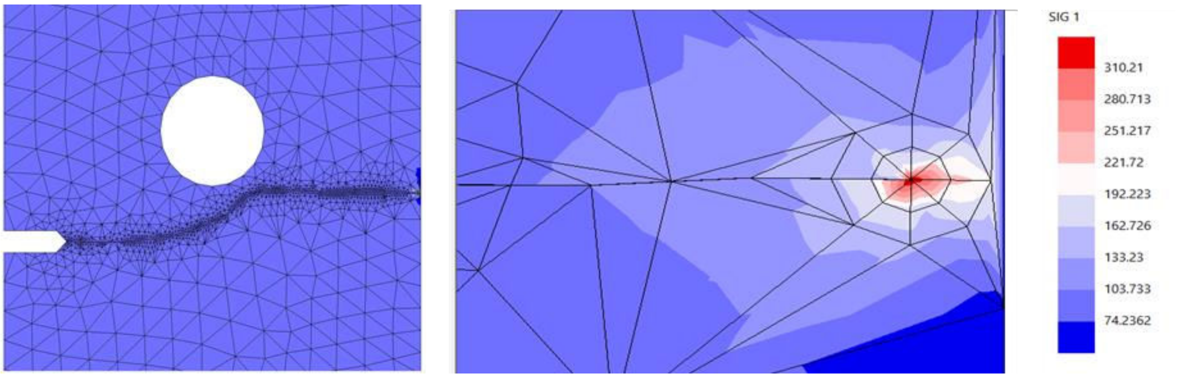


Figure 6. Comparing the estimation of K_I and K_{II} using ANSYS and FRANC2D/L Software.

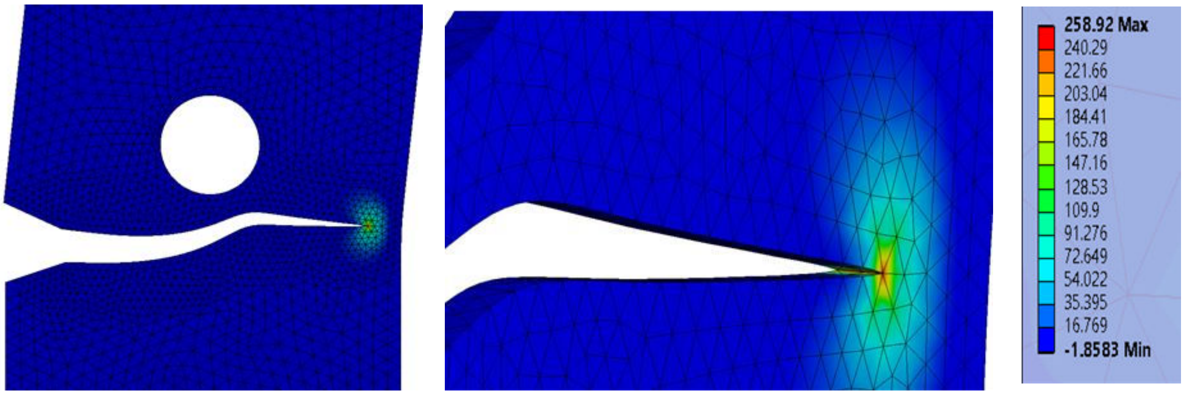
Figure 7 shows a side-by-side comparison of the maximum and minimum principal stresses obtained from the specimens analyzed by both programs. Figures 7a and 7b depict the maximum principal stress obtained from FRANC2D/L and ANSYS, respectively. On the other hand, Figures 7c and 7d illustrate the minimum principal stress obtained from FRANC2D/L and ANSYS, respectively. The maximum and minimum principal stresses estimated by FRANC2D/L are about 1.2 and 1.38 times greater than those obtained by ANSYS, respectively. The results obtained from the two programs exhibit a slight difference in values, which can be attributed to the variation in the number of steps employed and the numerical methods used in each program. However, even with this small percentage of error, the comparison offers valuable information regarding the precision and consistency of the stress analysis outcomes produced by the FRANC2D/L. program.

4.2. An Edge crack in a plate with a hole

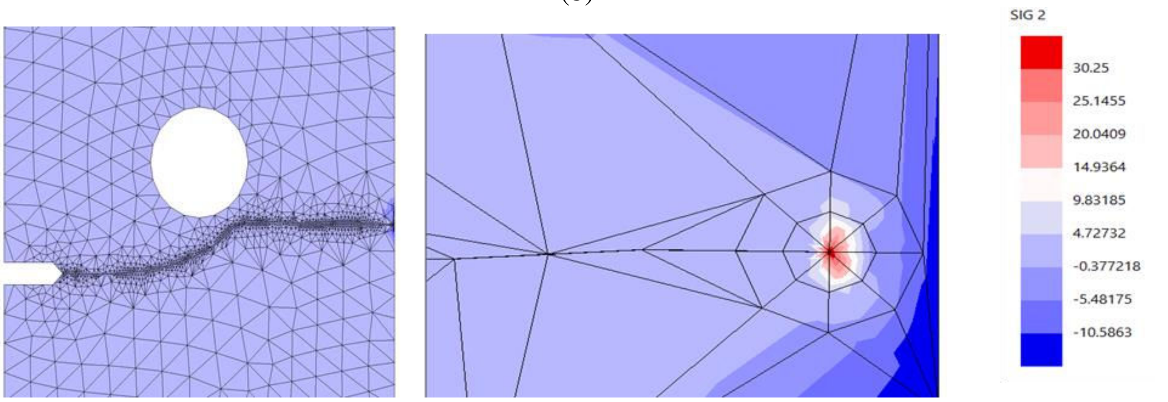
The problem illustrated in Figure 8 involves a rectangular plate made of Aluminum 7075-T6, with dimensions of 120mm x 65mm x 16mm, and linear elastic material behavior. The plate contains two holes with a diameter of 13 mm, while a 20 mm hole located near the center. The initial measuring 10 mm is loacated at the left center edge of the specimen. The material properties are listed in Table 1, and a cyclic load of 20 kN, with a load ratio of 0.1 (R = 0.1), is imposed on the plate.



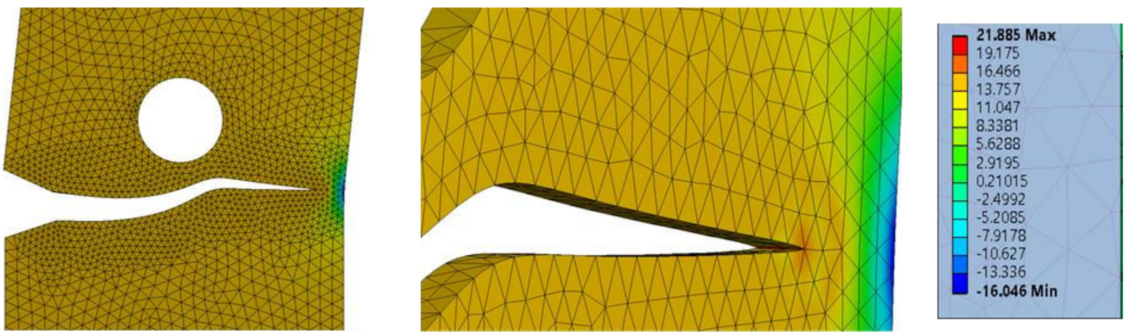
(a)



(b)



(c)



(d)

Figure 7. Comparison of Maximum and Minimum Principal Stresses (MPa) obtained in Franc2D/L (a and c) and Ansys (b and d).

Table 1. Mechanical properties of Aluminum 7075-T6.

Properties	Metric Units Value
Elasticity Modulus, E	71.7 GPa
Poisson's ratio, υ	0.33
Yield strength, σ_Y	496 MPa
Ultimate strength, σu	538 MPa
Paris' law coefficient, C	5.27×10^{-10}
Paris law exponent, m	2.947

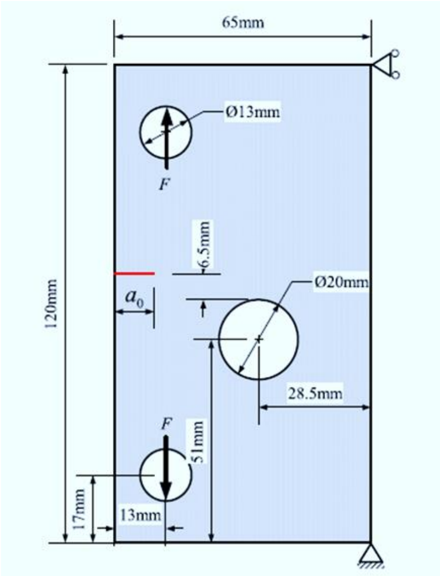
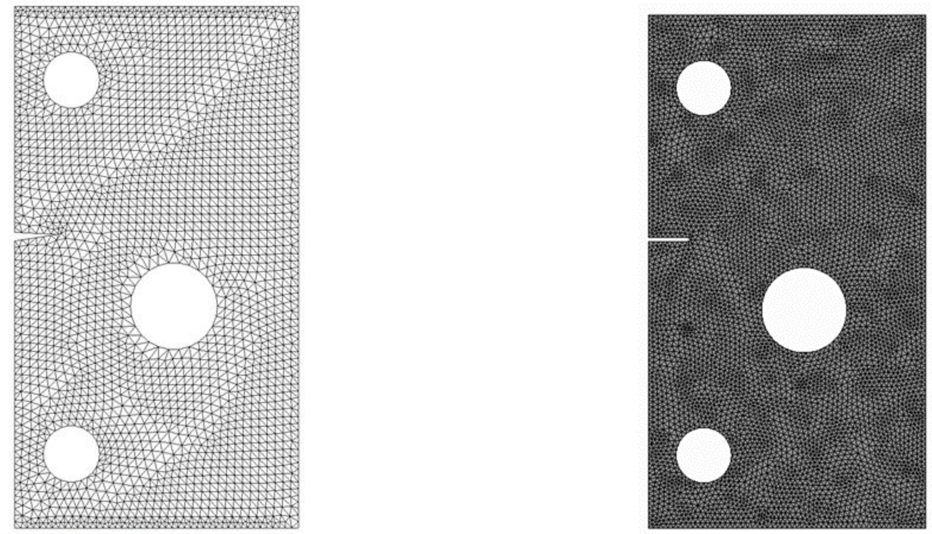


Figure 8. Geometric dimensions of the second problem.

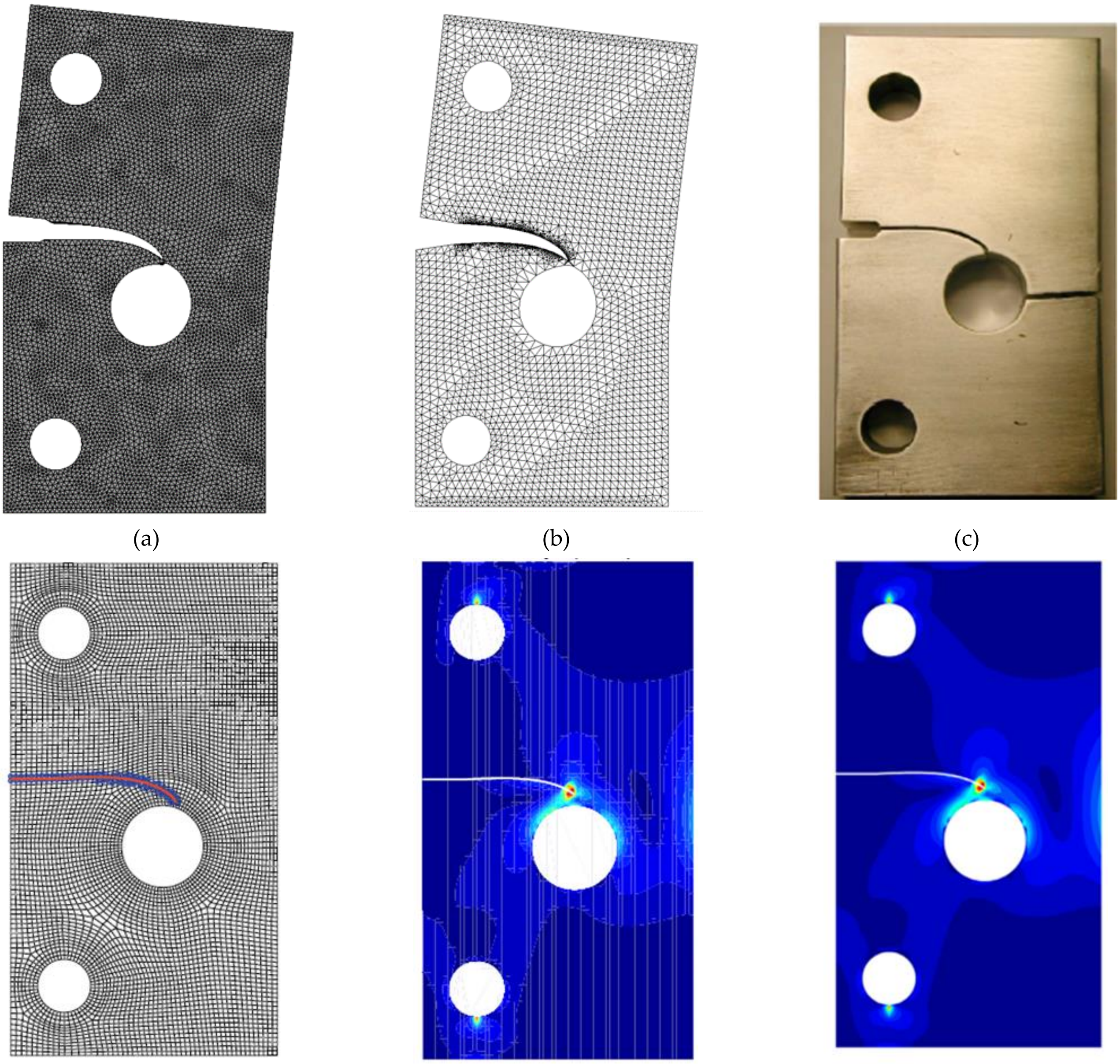
The mesh for the simulation was created using tetrahedrons with 8 nodes in FRANC2D/L, resulting in a mesh with 10,107 nodes and 5,127 elements, as shown in Figure (9a). On the other hand, Ansys generated an initial mesh with an element size of 1 mm, resulting in a mesh with 486,542 nodes and 339,463 elements, as depicted in Figure (9b).



(a) (b)

Figure 9. Generated Mesh: a) FRANC2D/L, (b) ANSYS.

Figure 10 presents a visual comparison of five different crack paths, including those generated by ANSYS and FRANC2D/L software in this study. These paths include one obtained from experimental data (Giner, Sukumar et al. 2009) which was performed at a stress ratio R = 0.1, and three generated using extended finite element method with ABAQUS software (Giner, Sukumar et al. 2009), decomposed updating reanalysis based on the XFEM (Cheng and Wang 2019), extended finite element method with ABAQUS software (Giner, Sukumar et al. 2009), and extended finite element method with a controllable crack propagation strategy (Cheng and Wang 2018). Corresponding crack paths are presented in Figures 10a-f. The results of this study demonstrate that the predicted path of crack propagation is highly consistent with the experimentally observed path. The study's results indicate that the FRANC2D/L software is a reliable and trustworthy tool for simulating crack propagation, demonstrating the accuracy and effectiveness of this particular simulation method.



(a) (b) (c)

(d) (e) (f)

Figure 10. Crack growth path (a) ANSYS (b) FRANC2D/L (c) Experiemental work (Giner, Sukumar et al. 2009) (d) numerical (Giner, Sukumar et al. 2009), (e) numerical (Cheng and Wang 2019), (f) numerical (Cheng and Wang 2018).

As predicted, the presence of a hole induced unbalanced stresses at the tip of a fatigue crack, causing it to propagate towards the hole. Interestingly, holes played a dual role as both crack stoppers and focal points for crack growth. This phenomenon occurs because the stress concentration at the hole's edge attracts the crack trajectory towards it, leading to the crack growing towards the hole. In situations where the hole is not in the immediate vicinity of the fatigue crack, the deviation of the crack trajectory and its consequent shift away from the original path can cause the crack to bypass the hole entirely. These findings highlight the importance of considering the influence of holes on stress distribution and crack propagation in engineering designs. Therefore, it is crucial to consider the distance between holes and potential crack paths when designing structures to ensure that they can withstand the stresses and loads they will be exposed to during their operating lifetime.

The good agreement between the predicted results of K_I and K_{II} between ANSYS and FRANC2D/L software as shown in Figures 11 and 12 can be attributed to their use of the finite element method, advanced algorithms, user-friendly interfaces, and rigorous validation and verification processes. As observed in Figure 12, the presence of the lowest peaks suggests a sudden increase in the values of the second mode of SIFs when the crack tip approaches the hole. This indicates a significant change in the stress distribution around the crack tip area. These algorithms take into account various physical properties such as material strength, elasticity, and fracture toughness, which are essential factors in accurately predicting stress intensity factors.

Figure 13 demonstrates a high level of concordance between the fatigue life cycle predictions generated by two distinct computational tools, ANSYS and FRANC2D/L. The close alignment of ANSYS and FRANC2D/L outcomes underpins the reliability and validity of these two software packages in predicting fatigue life cycles. Both tools employ different theoretical models and algorithms, yet they converge on similar results, reinforcing the robustness of their respective approaches. This demonstrates that both tools are capable of accurately predicting fatigue life cycles under a wide range of scenarios, further attesting to their applicability and versatility in practical settings.

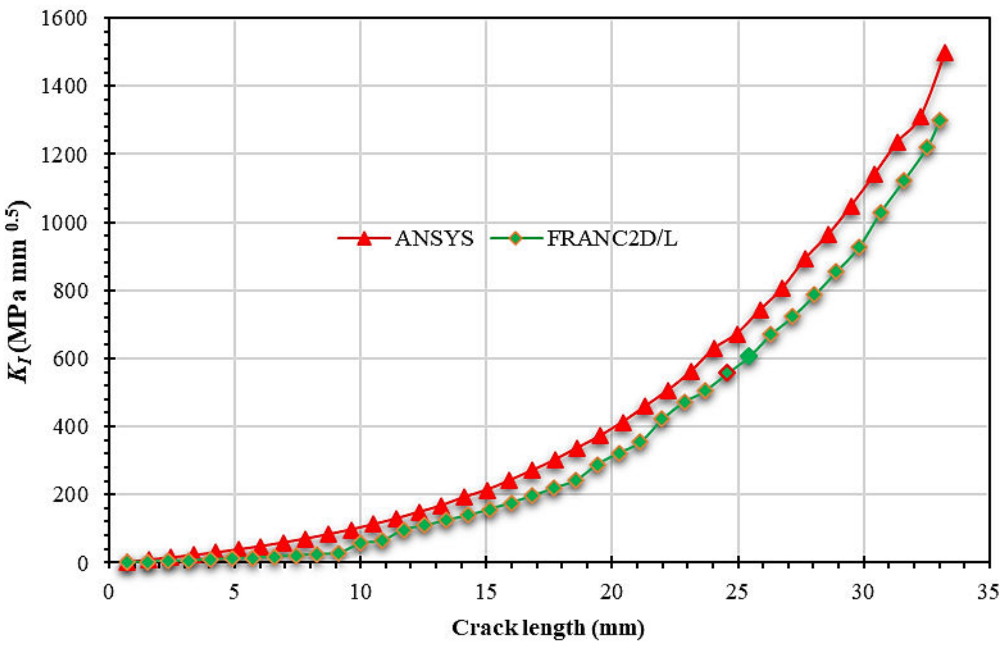


Figure 11. Predicted values of K_I by ANSYS and FRANC2D/L

.

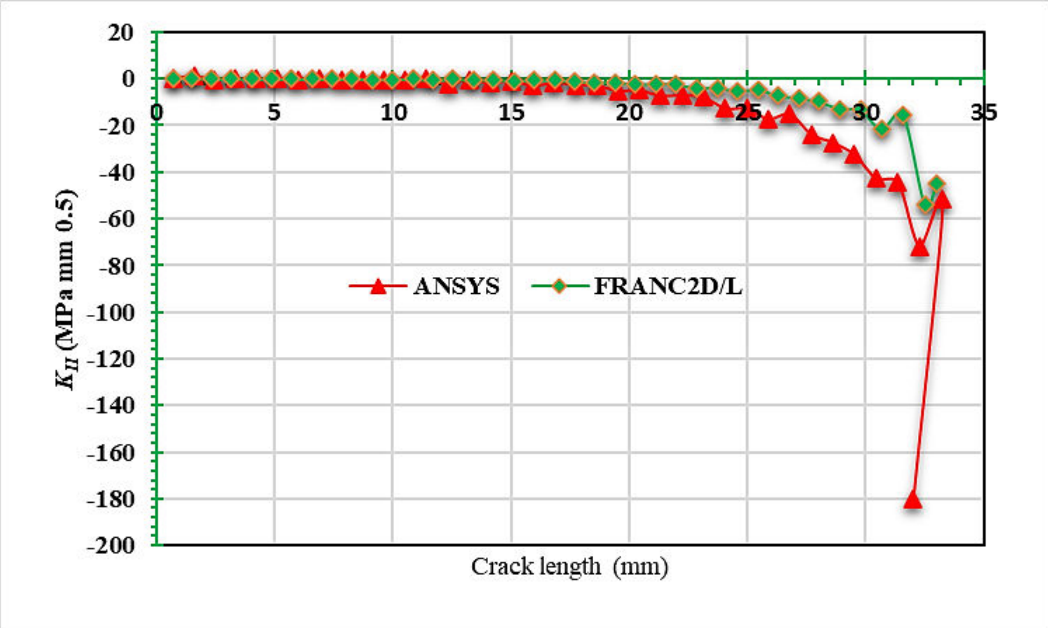


Figure 12. Predicted values of K_{II} by ANSYS and FRANC2D/L.

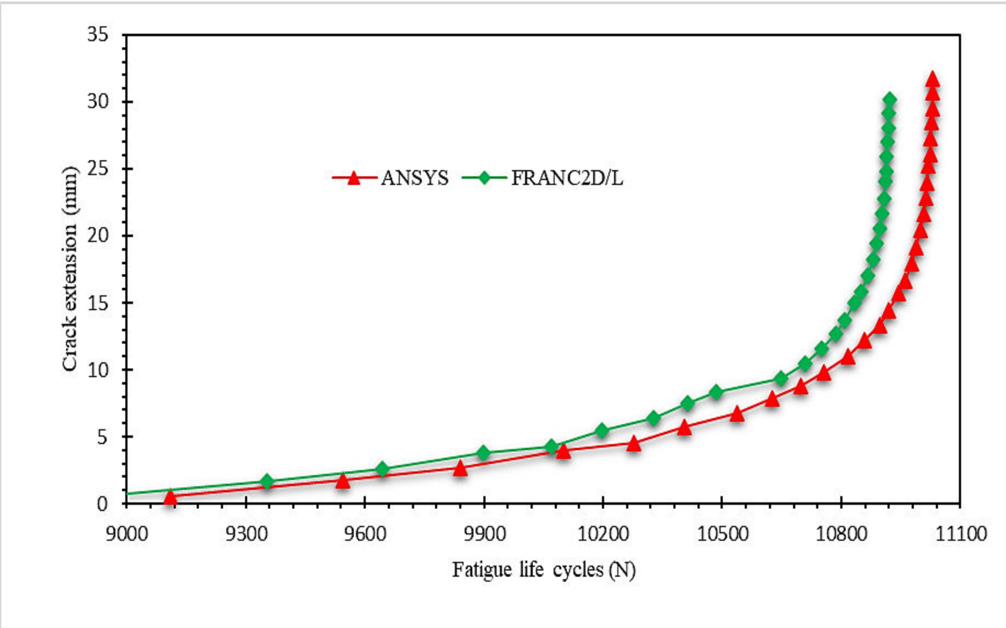


Figure 13. Predicted Fatigue life cycles from ANSYS and FRANC2D/L.

CONCLUSION:

This study demonstrates that FRANC2D/L software can serve as a viable alternative to ANSYS Workbench for analyzing crack propagation problems using LEFM. Though both software yielded comparable results for parameters like stress intensity factors, fatigue life, and crack propagation path, FRANC2D/L offered faster computation times and greater flexibility in mesh generation. While ANSYS Workbench may still be preferable for complex 3D visualizations. By benchmarking against reference solutions, this work validates the use of FRANC2D/L as a substitute for ANSYS Workbench, especially when 3D modeling is not critical. The study provides evidence that FRANC2D/L is a useful computational tool for LEFM analysis across multiple applications. The hole's presence not only acts as a crack stopper but also attracts the crack path towards it, resulting in growth towards the hole. The extent of crack growth towards the hole is contingent upon the distance between the hole and the crack location. In some cases, the crack trajectory deviates from the hole, resulting in the missed hole phenomenon, in some instances, cracks can grow towards and sink into the hole, known as the sinking in hole phenomenon. These phenomena have significant implications for engineering design, as the presence of holes can either promote or inhibit crack growth, depending on their location and size. It is therefore important to carefully consider the position and size of holes and other structural features when designing materials and components to ensure they can withstand the stresses and loads they will be exposed to during their operational lifetime. Based on the simulation analysis performed in both software, it appears that FRANC2D/L has a significantly shorter execution time compared to ANSYS when selecting the same mesh size. According to the analysis, FRANC2D/L takes no more than 1 minute to complete the simulation, while ANSYS requires approximately 30 minutes for the same task. Through the comparison of different software packages, valuable insights can be gained regarding areas for improvement and the strengths and weaknesses of each tool. This analysis not only aids users in selecting the most suitable software for their specific requirements but also promotes competition among developers, driving enhancements in software offerings. As a result, our efforts contribute to the continuous development and refinement of software tools, which plays a pivotal role in supporting the scientific and engineering community. By identifying and addressing the limitations and potentials of these tools, we actively contribute to advancing the capabilities and usability of software, benefiting researchers, practitioners, and the broader scientific community.

References

Alizadeh, H., D. Hills, P. De Matos, D. Nowell, M. Pavier, R. Paynter, D. Smith and S. Simandjuntak (2007). "A comparison of two and three-dimensional analyses of fatigue crack closure." International journal of fatigue **29**(2): 222-231.

Alshoaibi, A. M. (2010). "Finite element procedures for the numerical simulation of fatigue crack propagation under mixed mode loading." Structural Engineering and Mechanics **35**(3): 283-299.

Alshoaibi, A. M. (2019). "Comprehensive Comparisons of Two and Three Dimensional Numerical Estimation of Stress Intensity Factors and Crack propagation in Linear Elastic Analysis." International Journal of Integrated Engineering **11**(6): 45-52.

Alshoaibi, A. M. and Y. A. Fageehi (2020). "2D finite element simulation of mixed mode fatigue crack propagation for CTS specimen." Journal of Materials Research and Technology **9**(4): 7850-7861.

Alshoaibi, A. M. and O. Yasin (2019). "Finite element simulation of crack growth path and stress intensity factors evaluation in linear elastic materials." Journal of Computational & Applied Research in Mechanical Engineering (JCARME).

Anderson, T. L. (2017). Fracture mechanics: fundamentals and applications, CRC press.

ANSYS (2020). "Academic Research Mechanical, Release 19.2, Help System." Coupled Field Analysis Guide **ANSYS, Inc.**: (ONLINE).

Bang, D., A. Ince and M. Noban (2019). "Modeling approach for a unified crack growth model in short and long fatigue crack regimes." International Journal of Fatigue **128**: 105182.

Barsom, J. and S. Rolfe (1999). "Fracture and Fatigue in Structure: Application of fracture Mechanics." American Society for Testing and Materials Philadelphia, PA.

Belytschko, T. and T. Black (1999). "Elastic crack growth in finite elements with minimal remeshing." International journal for numerical methods in engineering **45**(5): 601-620.

Bjørheim, F. (2019). Practical comparison of crack meshing in ANSYS mechanical APDL 19.2. Master's thesis in structural engineering, University of Stavanger, Norway.

Bouchard, P.-O., F. Bay, Y. Chastel and I. Tovena (2000). "Crack propagation modelling using an advanced remeshing technique." Computer methods in applied mechanics and engineering **189**(3): 723-742.

Broek, D. (2012). Elementary engineering fracture mechanics, Springer Science & Business Media.

Chen, H., Q. Wang, W. Zeng, G. Liu, J. Sun, L. He and T. Q. Bui (2019). "Dynamic brittle crack propagation modeling using singular edge-based smoothed finite element method with local mesh rezoning." European Journal of Mechanics-A/Solids **76**: 208-223.

Cheng, Z. and H. Wang (2018). "How to control the crack to propagate along the specified path feasibly?" Computer Methods in Applied Mechanics and Engineering **336**: 554-577.

Cheng, Z. and H. Wang (2019). "An exact and efficient X-FEM-based reanalysis algorithm for quasi-static crack propagation." Applied Mathematical Modelling **72**: 601-622.

De Jesus, A. M., A. L. da Silva, M. V. Figueiredo, J. A. Correia, A. S. Ribeiro and A. A. Fernandes (2011). "Strain-life and crack propagation fatigue data from several Portuguese old metallic riveted bridges." Engineering Failure Analysis **18**(1): 148-163.

Dekker, R., F. van der Meer, J. Maljaars and L. Sluys (2019). "A cohesive XFEM model for simulating fatigue crack growth under mixed-mode loading and overloading." International Journal for Numerical Methods in Engineering **118**(10): 561-577.

Demir, O., A. O. Ayhan and S. İriç (2017). "A new specimen for mixed mode-I/II fracture tests: Modeling, experiments and criteria development." Engineering Fracture Mechanics **178**: 457-476.

Dirik, H. and T. Yalçinkaya (2018). "Crack path and life prediction under mixed mode cyclic variable amplitude loading through XFEM." International Journal of Fatigue **114**: 34-50.

Duchêne, L., F. El Houdaigui and A. M. Habraken (2007). "Length changes and texture prediction during free end torsion test of copper bars with FEM and remeshing techniques." International Journal of Plasticity **23**(8): 1417-1438.

Erdogan, F. and G. Sih (1963). "On the crack extension in plates under plane loading and transverse shear." Journal of basic engineering **85**(4): 519-525.

Fageehi, Y. A. and A. M. Alshoaibi (2020). "Nonplanar Crack Growth Simulation of Multiple Cracks Using Finite Element Method." Advances in Materials Science and Engineering **2020**.

Fageehi, Y. A. and A. M. Alshoaibi (2020). "Numerical Simulation of Mixed-Mode Fatigue Crack Growth for Compact Tension Shear Specimen." Advances in Materials Science and Engineering **2020**.

Giner, E., N. Sukumar, J. E. Tarancón and F. J. Fuenmayor (2009). "An Abaqus implementation of the extended finite element method." Engineering fracture mechanics **76**(3): 347-368.

Gomes, G. and A. C. Miranda (2018). "Analysis of crack growth problems using the object-oriented program bemcracker2D." Frattura ed Integrità Strutturale **12**(45): 67-85.

Hellan, K. (1985). Introduction to fracture mechanics, McGraw-Hill.

Huynh, H. D., M. N. Nguyen, G. Cusatis, S. Tanaka and T. Q. Bui (2019). "A polygonal XFEM with new numerical integration for linear elastic fracture mechanics." Engineering Fracture Mechanics **213**: 241-263.

Kanth, S. A., G. Harmain and A. Jameel (2018). "Modeling of Nonlinear Crack Growth in Steel and Aluminum Alloys by the Element Free Galerkin Method." Materials Today: Proceedings **5**(9): 18805-18814.

Kim, Y.-S., S.-H. Yang, D. Shan, S.-O. Choi, S.-M. Lee and B.-S. You (2006). "Three-dimensional rigid-plastic FEM simulation of metal forming processes." Journal of Materials Engineering and Performance **15**(3): 275-279.

Leclerc, W., H. Haddad and M. Guessasma (2017). "On the suitability of a Discrete Element Method to simulate cracks initiation and propagation in heterogeneous media." International Journal of Solids and Structures **108**: 98-114.

Lee, Y.-L., J. Pan, R. Hathaway and M. Barkey (2005). Fatigue testing and analysis: theory and practice, Butterworth-Heinemann.

Li, X., H. Li, L. Liu, Y. Liu, M. Ju and J. Zhao (2020). "Investigating the crack initiation and propagation mechanism in brittle rocks using grain-based finite-discrete element method." International Journal of Rock Mechanics and Mining Sciences **127**: 104219.

Liu, Y., Y. Li and W. Xie (2017). "Modeling of multiple crack propagation in 2-D elastic solids by the fast multipole boundary element method." Engineering Fracture Mechanics **172**: 1-16.

Paris, P. C., H. Tada and J. K. Donald (1999). "Service load fatigue damage—a historical perspective." International Journal of fatigue **21**: S35-S46.

PC, P. (1961). "A rational analytic theory of fatigue." Trends Engin **13**: 9-14.

Pook, L. P. (2000). Linear elastic fracture mechanics for engineers: theory and applications, WIT press.

Rezaei, S., S. Wulfinghoff and S. Reese (2017). "Prediction of fracture and damage in micro/nano coating systems using cohesive zone elements." International Journal of Solids and Structures **121**: 62-74.

Rozumek, D. and E. Macha (2006). "Elastic–plastic fatigue crack growth in 18G2A steel under proportional bending with torsion loading." Fatigue & Fracture of Engineering Materials & Structures **29**(2): 135-144.

Rozumek, D., Z. Marciniak, G. Lesiuk and J. Correia (2017). "Mixed mode I/II/III fatigue crack growth in S355 steel." Procedia Structural Integrity **5**: 896-903.

Shao, Y., Q. Duan and S. Qiu (2019). "Adaptive consistent element-free Galerkin method for phase-field model of brittle fracture." Computational Mechanics **64**(3): 741-767.

Sih, G. and H. Liebowitz (1968). Mathematical Fundamentals. Fracture, Academic Press New York. **2:** 67-190.

Surendran, M., S. Natarajan, G. Palani and S. P. Bordas (2019). "Linear smoothed extended finite element method for fatigue crack growth simulations." Engineering Fracture Mechanics **206**: 551-564.

Suresh, S. (1998). Fatigue of materials, Cambridge university press.

Varfolomeev, I., M. Burdack, S. Moroz, D. Siegele and K. Kadau (2014). "Fatigue crack growth rates and paths in two planar specimens under mixed mode loading." International Journal of Fatigue **58**: 12-19.

Wiragunarsa, I. M., L. R. Zuhal, T. Dirgantara and I. S. Putra (2021). "A particle interaction-based crack model using an improved smoothed particle hydrodynamics for fatigue crack growth simulations." International Journal of Fracture: 1-16.

Zhan, Z., W. Hu, B. Li, Y. Zhang, Q. Meng and Z. Guan (2017). "Continuum damage mechanics combined with the extended finite element method for the total life prediction of a metallic component." International Journal of Mechanical Sciences **124**: 48-58.

Zhang, R. and R. Guo (2018). "Determination of crack tip stress intensity factors by singular Voronoi cell finite element model." Engineering Fracture Mechanics **197**: 206-216.

Zhang, W. and A. Tabiei (2020). "An Efficient Implementation of Phase Field Method with Explicit Time Integration." Journal of Applied and Computational Mechanics **6**(3): 373-382.