Universal Journal of Mechanical Engineering 7(6): 367-379, 2019 http://www.hrpub.org
DOI: 10.13189/ujme.2019.070608

Assessment of Stress Raiser Factor Using Finite Element Solvers

A. A Fondjo[1], T. C Dzogbewu[2,*]

[1]Department of Civil Engineering, Central University of Technology, Free State, South Africa
[2]Department of Mechanical Engineering, Central University of Technology, Free State, South Africa

Received August 22, 2019; Revised September 23, 2019; Accepted October 1, 2019

Abstract The stress raisers factor around circular holes in a plate exposed to uniform tensile load at the edges has been studied using Finite Element Analysis solvers. The effect of mesh quality on stress raisers factor, the maximum Von Mises stresses, the computing time, and the percentage error has been examined. 4 Node Quadrilateral Element and 8 Node Quadrilateral Element were utilized respectively as first-order component (4NQE) and higher-order component (8NQE) to assess the maximum Von Mises stress and the numerical stress raiser factor (Kn) at various mesh sizes. The maximum Von Mises stress and the stress raiser factor were determined using the following finite element solvers: ABAQUS, ANSYS, CATIA, STRAND 7, ALGOR, COSMOS/M, and FEMAP. The estimations of the numerical stress raiser factor (Kn) were compared with the theoretical stress raiser factor (Kt). There were discrepancies observed between the maximum Von Mises stresses of the FEA solvers.

Keywords Finite Element Analysis, Stress Raiser Factors, Von Mises Stress, Mesh Size, Geometric Discontinuities

1. Introduction

Stress analyses were performed analytically and experimentally, which could be both troublesome and time-consuming, particularly when dealing with geometric discontinuities (notch, groove, hole, etc.). Researchers have analysed various stress raising geometries such as circular, elliptical openings, U and V notches in a plate with various loading conditions and materials (Howland, 1930; Pilkey and Pilkey, 2008; Dzogbewu et al., 2017). The utilization of the Finite Element Analysis (FEA) to investigate irregular structural systems has received noteworthy enthusiasm.

FEA has been developed into an indispensable tool in the modeling and simulation for decades (Mauk, 2010; Dzogbewu, 2017). This method is cost-effective and reliable to guarantee the functionality of the finished item (Lui and Quek, 2013). Hutton (2004) stated that the mathematical foundations of finite element analysis date back at least half of a century. Clough (1960) was the first to use the term finite element. The analysis of stress in a plane had been common usage since that time (Rayleigh, 1870; Hutton, 2004). During the period between 1960 and 1970, FEA was developed to the application in pressure vessels, shell bending, plate bending and general three-dimensional issue in the linear elastic analysis (Melosh, 1961; Melosh, 1963). During the year 1977, the finite element software CATIA was developed by Dassault Systèmes (Systemes, 2016). After many other finite element solvers were introduced such as ABAQUS (Hibbit and Inc, 2004), ANSYS (Thompson and Thompson, 2017), STRAND 7 (Strand7, 2010), ALGOR (Spyrakos, 1995), COSMOS/M (Lashkari, 1988), FEMAP (FEMAP, 2007).

The stress raiser factor is a dimensionless factor that is used to gauge how concentrated the stress is in a material. It is characterized as the proportion of the maximum stress in the component to the reference stress (Howland, 1930; Pilkey and Pilkey, 2008). The theoretical stress raiser factor (Kt) is obtained by analytical methods (charts and formulae), whereas the numerical stress raiser factor (Kn) is obtained from FEA (Pilkey and Pilkey, 2008; Mauk, 2010; Lui and Quek, 2013).

Howland (1930) was the first to investigate the stress raiser of a long isotropic rectangular plate with a central opening subject to a tensile force. Heywood (1952) investigated different isotropic shapes with an extensive range of holes and developed different equations of finite width plate with various opening shapes and long length. Ukadgaonker and Patil (1993) studied the stress raiser in a rectangular plate with two elliptical holes and proposed formulae. Another study by Young, Budynas and Sadegh

(2002) compiled analytical formulae for determining the theoretical stress raiser factor (Kt) on a plate under various loading conditions Pilkey and Pilkey (2008) Studied the stress raiser factor for a rectangular plate with holes and notches under bending, torsion and axial loads by aggregating previous research works and developed great theories, analytical formulae, and charts utilizing experimental works and mathematical analysis. These theories exhibit excellent methodology in a graphical form to assess the theoretical stress raiser factor (Kt) in an isotropic plate. Patle and Bhope (2012) proposed analytical formulae to determine the stress raiser factor in a plate with the diagonal opening. Pan, Cheng and Liu (2013) investigated a stress raiser in a finite plate with rectangular opening subjected to uniaxial loading condition and proposed formulae to be used for evaluating the theoretical stress raiser factor (Kt).

Authors like Mittal and Jain (2007), Li et al. (2008), Growney, (2008) and Snowberger (2008) investigated the stress concentration in a plate placed under various loading conditions utilizing FEA. It was observed that the selection of the size of the mesh is critical. Generally, models with small element size (fine mesh) yield exceptionally more accurate results. However, fine mech may take longer processing time. Models with large element size (coarse mesh) give less precise results with shorter computing time. Furthermore, More and Bindu (2015) and Dutt (2015) pointed out that the computing time and the accuracy of the results of FEA depend significantly on mesh density. Due to the rapid technology advancement, FEA solvers became capable to generate automatic meshes (Lui and Quek, 2013). Nevertheless, according to Lewis, Nithiarasu and Seetharamu (2004) and Anderson (2005) the craft of designing an appropriate mesh required human intervention, particularly for complex problems such as discontinuities, cracks, and openings. Anderson (2005) stated that determining the optimum finite element mesh during modeling was one of the difficult tasks to achieve by the pioneers and even until today.

The investigation of a specific problem using different FEA solvers with identical design parameters could yield results with differences. The current research seeks to use seven FEA solves to assess the stress raiser factor and highlight the eventual discrepancies among their results and also to investigate the correlations between the numerical stress raiser factor values (Kn) and the theoretical stress raiser factor (Kt).

2. Materials and Methods

2.1. Materials Properties, Loading Condition, and Meshing

The study was conducted in a rectangular steel plate (Fig.1). The material properties were adopted from (MatWeb, 2019) material property datasheet as follows:

AISI-1025 cold rolled steel strip,
Young Modulus E: 203395.5 MPa,
Density $\gamma = 7.861.10^3$ Kg .m^{-3}
Poisson ratio υ: 0.32,
Yield Criteria: Von Mises

To create the finite element model, the steel plate size was reduced using the symmetry conditions. The geometry and loading have a horizontal and vertical line of symmetry for the specimen (Fig.3). The strain, stress and displacement field in each quarter is the equivalent to the entire steel plate. Therefore, only one-quarter of the steel plate is required for modeling. The boundary conditions are applied at the edges (Fig.2). The symmetry conditions make the quarter plate (Fig.4) demonstrate the characteristics of a complete plate.

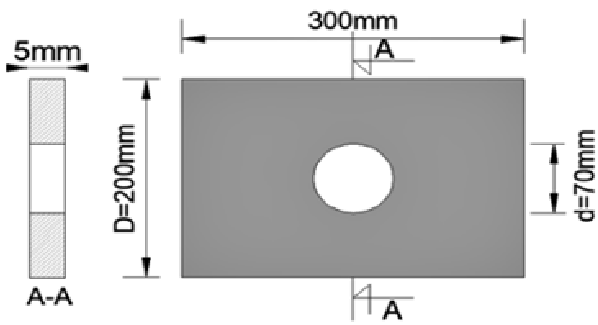


Figure 1. Specimen geometry

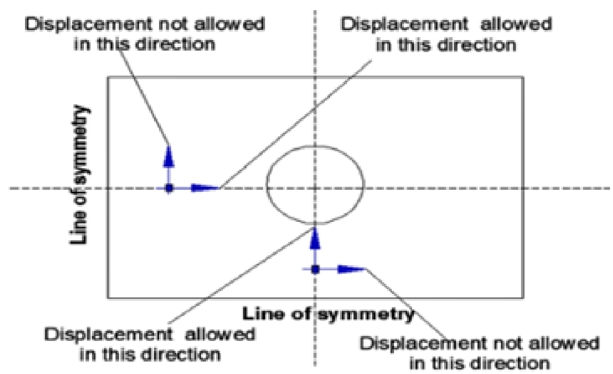


Figure 2. Boundary conditions

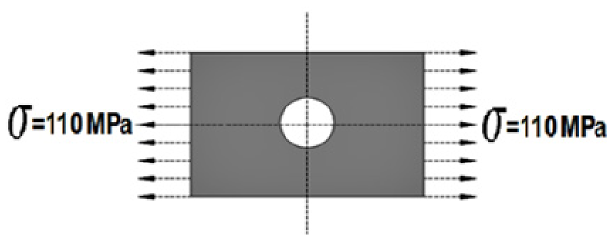


Figure 3. Specimen loading diagram

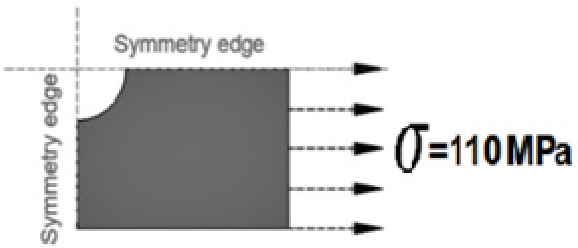
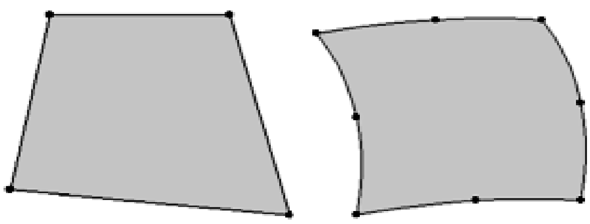


Figure 4. 1/4 Specimen loading diagram

2.2. Meshing

Quadrilateral element (Fig.5) is constantly exact contrasted with a triangular element since displacements are interpolated to a higher-degree in quadrilateral element than first-degree in the triangular element (Mauk, 2010; Lui and Quek, 2013). A three noded triangle is known as Constant Strain Triangle (CST) because the strain in those elements is constant, while in case of a quadrilateral element it is not the case. The quadrilateral element is a two-dimensional finite element with both global and local coordinates. It is described by a quadratic shape function in each of the x and y bearings (Clough, 1960; Spyrakos, 1995). 4 Node Quadrilateral Element (4NQE) (Fig.5a) and 8 Node Quadrilateral Element (8NQE) (Fig 5b) were used to construct the models.



a) Quad 4 b) Quad 8

Figure 5. 4 and 8 quadrilateral elements

2.3. Stress Raiser Factors Kt & Kn

The theoretical stress raiser factor (Kt) of the rectangular steel plate with a central circular opening subjected to tensile loading was determined using Equation 1 as proposed by Pilkey and Pilkey (2008). Different values of the numerical stress raiser factor (Kn) obtained from FEA were compare to the theoretical stress raiser factor (Kt).

$$\mathrm{Kt} = 3.00 - 3.140 \times \left(\frac{\mathrm{d}}{\mathrm{D}}\right) + 3.667 \times \left(\frac{\mathrm{d}}{\mathrm{D}}\right)^2 - 1.52 \times \left(\frac{\mathrm{d}}{\mathrm{D}}\right)^3 \quad (1)$$

For $0 \leq \frac{\mathrm{d}}{\mathrm{D}} \leq 1$

Where:

Kt = theoretical stress raiser factor

d = diameter of the single center circular hole

D = width of the plate

The theoretical stress raiser factor (Kt) is calculated using the information in Fig. 1: Specimen geometry.

$$\frac{\mathrm{d}}{\mathrm{D}} = \frac{70}{200} = 0.35 < 1$$

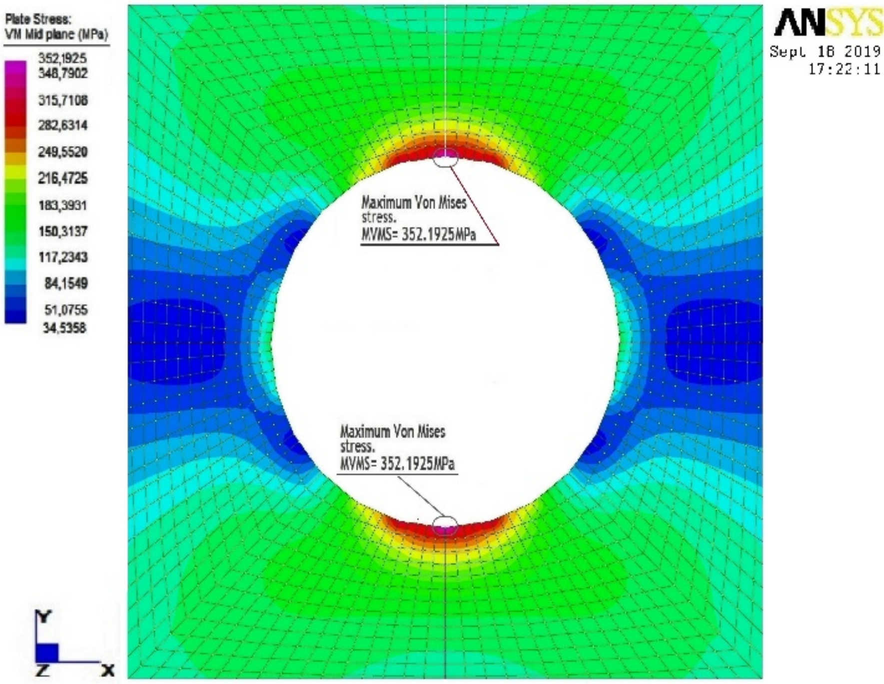
$$\mathrm{Kt} = 3.00 - 3.140 \times \left(\frac{70}{200}\right) + 3.667 \times \left(\frac{70}{200}\right)^2 - 1.527 \times \left(\frac{70}{200}\right)^3$$

$\mathrm{Kt} = 2.285.$

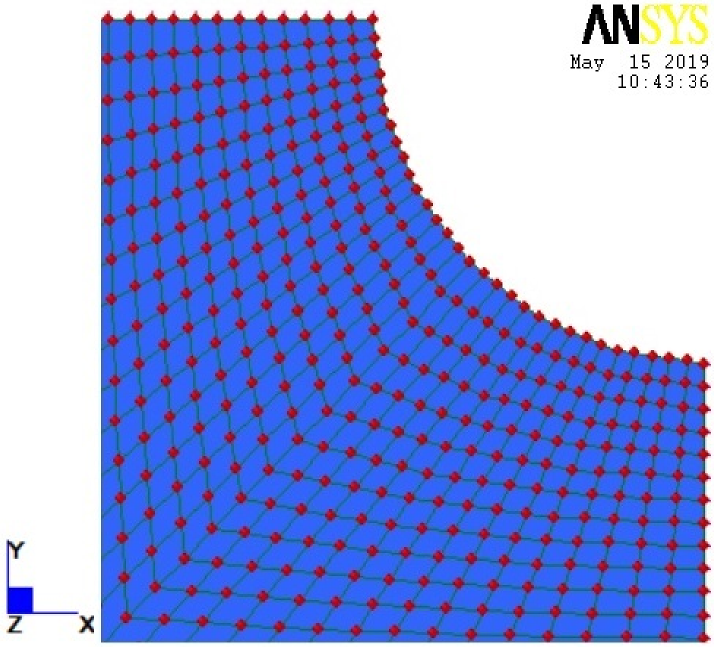
3. Results Analysis and Discussion

3.1. Analysis of Maximum Von Mises Stress

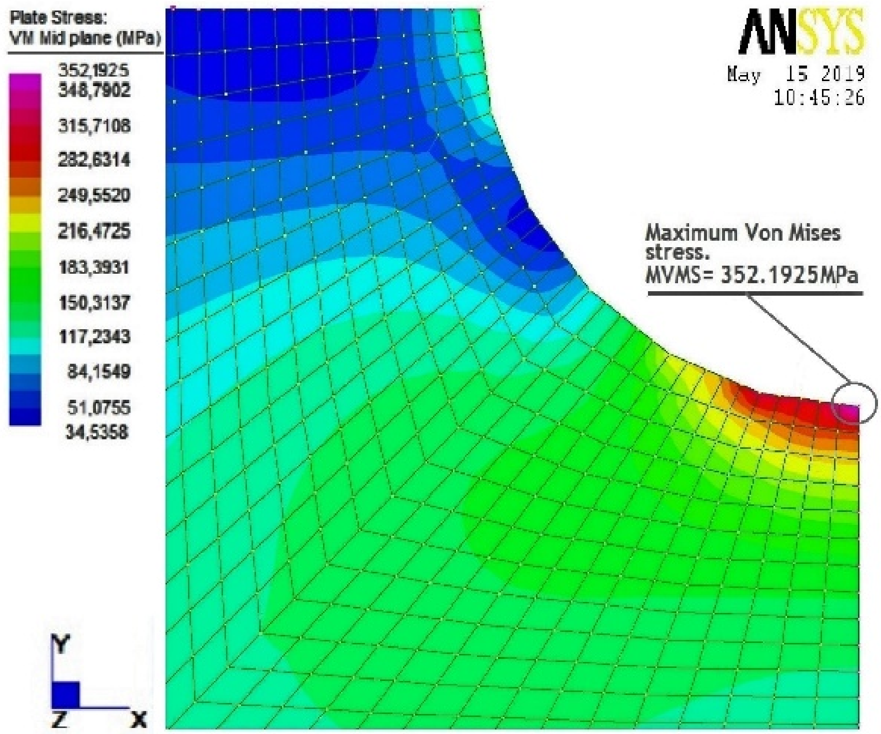
Finite element analysis was performed utilizing finite element solvers: ABAQUS, ANSYS, CATIA, STRAND 7, ALGOR, COSMOS/M, and FEMAP to determine the Maximum Von Mises stress (MVMS) and the numerical stress raiser factor (Kn). The analysis was done using a model of the first-order element (4NQE) and a higher-order element (8NQE). Nevertheless, only ANSYS results are presented because the results are very close to the Mean Maximum Von Mises stress (MMVMS). The MVMS distribution and the type of mesh are shown in Figs. 6 and 7. The maximum Von Mises stress occurs at the corner of the opening, and their values are 352.1925 MPa and 355.0830 MPa respectively for 4NQE and 8NQE.



a) Stress distribution of 4NQE (Full model)

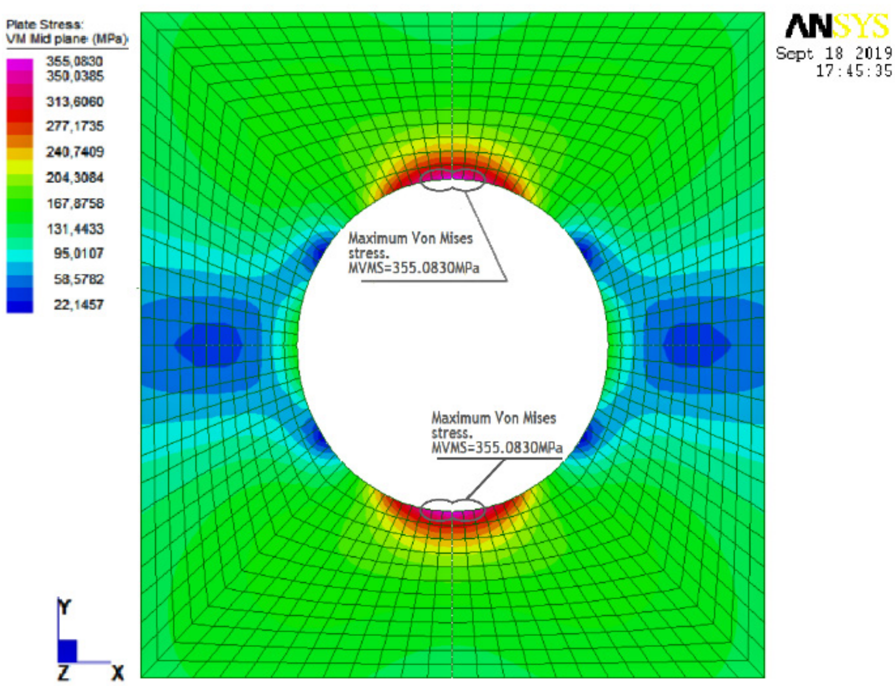


b) Mesh of 4NQE (Quarter model)

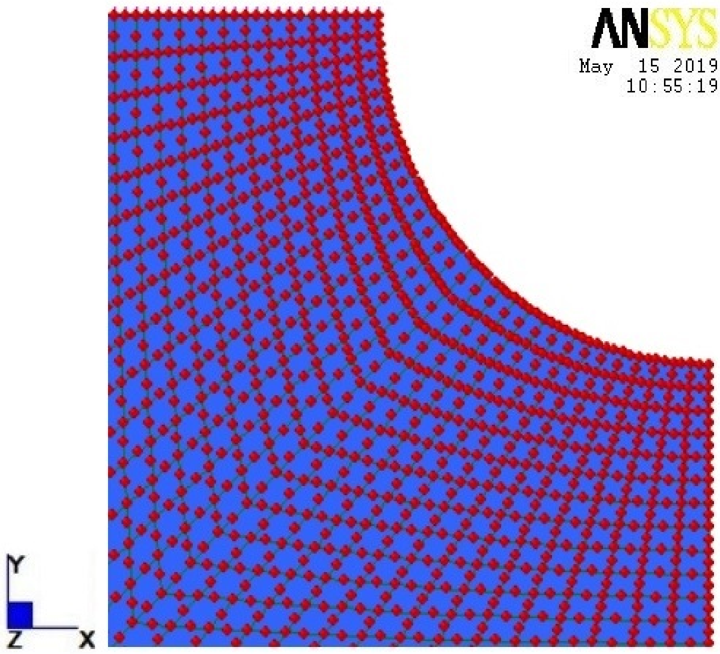


c) Stress distribution of 4NQE (Quarter model)

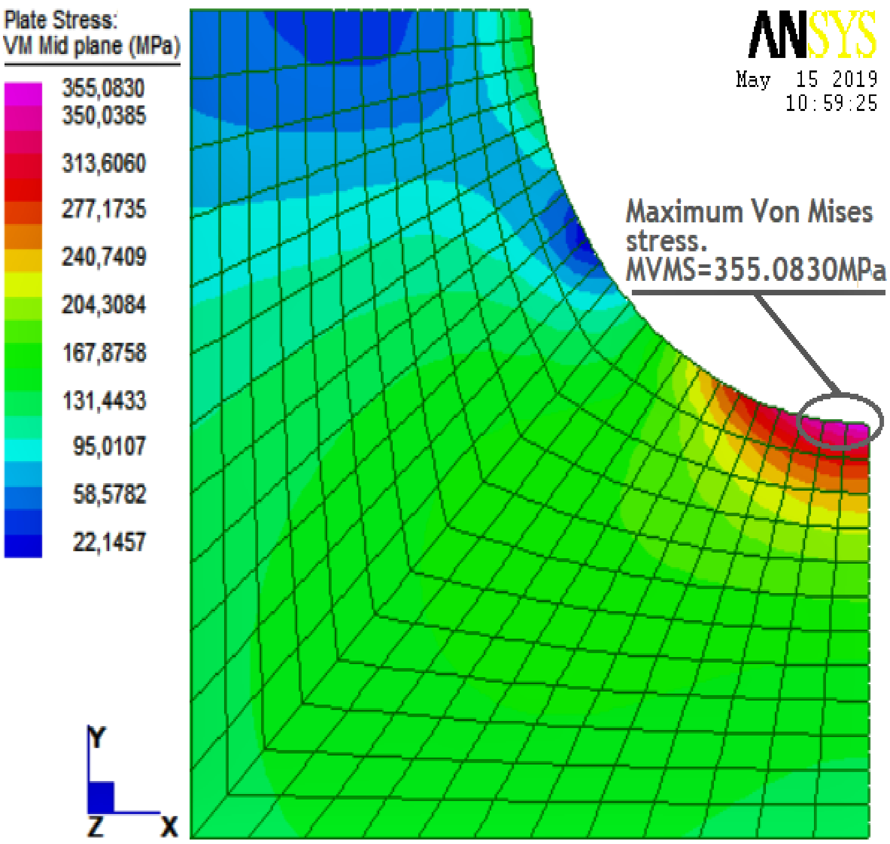
Figure 6. Mesh & stress distribution (4NQE)



a) Stress distribution of 8NQE (Full model)



b) Mesh of 8NQE (Quarter model)



b) Stress distribution of 8NQE (Quarter model)

Figure 7. Mesh & stress distribution (8NQE)

The Mean Maximum Von Mises stress (MMVMS) is defined as the average values of the maximum Von Mises stress (MVMS) obtained from FEA solvers: ABAQUS, ANSYS, CATIA, STRAND 7, ALGOR, COSMOS, and FEMAP.

Fig.8 shows the maximum Von Mises stress obtained at various mesh sizes within the range of 0.4 mm to 3.75 mm, utilizing First-order element (4NQE). The MVMS values obtained from the FEA solvers were different from one FEA solver to another. However, all the solvers demonstrated a decrease in the MVMS value with an increase in mesh size. The MMVMS in the graph was estimated as 350.160 MPa at 0.4 mm mesh size. It is apparent that there is a good correlation between the (MMVMS) values since the coefficient of correlation R^2=0.9956.

Fig.9 shows the maximum Von Mises stress obtained at various mesh sizes within the range of 0.4 mm to 3.75 mm, utilizing higher-order element (8NQE). The MVMS values obtained from the FEA solvers demonstrated some degree of discrepancies similar to the previous result of the lower-order element (4NQE) (Fig 8). Although the MMVMS value obtained for 8NQE at 0.4 mm mesh size (353.030 MPa) is only slightly higher than that of 4NQE (350.106 MPa), the slope of 8NQE (Fig. 9) is less steep than that of 4NQE (Fig.8). It could be speculated that with finer mesh size FEA solvers will demonstrate less discrepancies in their value with identical design parameters. The coefficient of correlation R^2=0.9408 proved that there is a good correlation between the MMVMS of the FEA solvers for 8NQE.

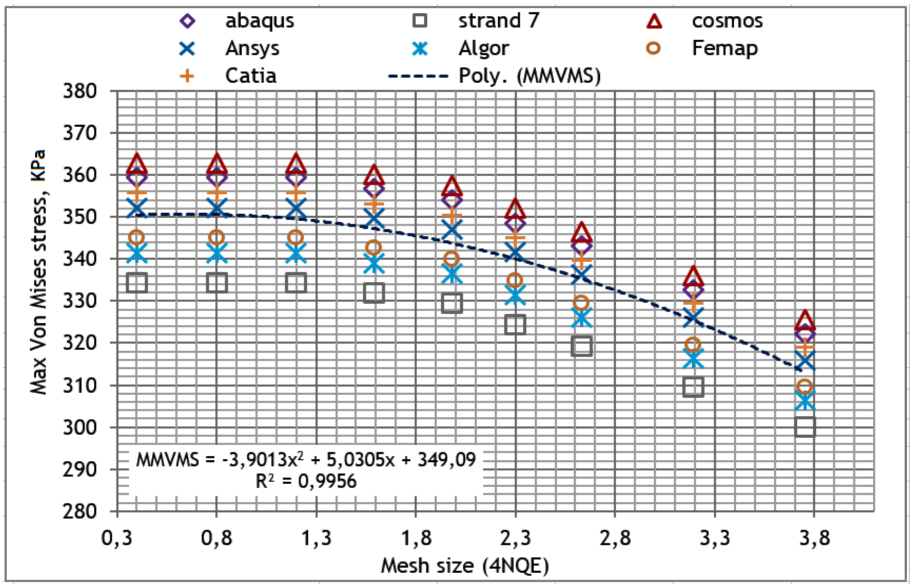


Figure 8. Maximum Von Mises stress versus Mesh size (4NQE)

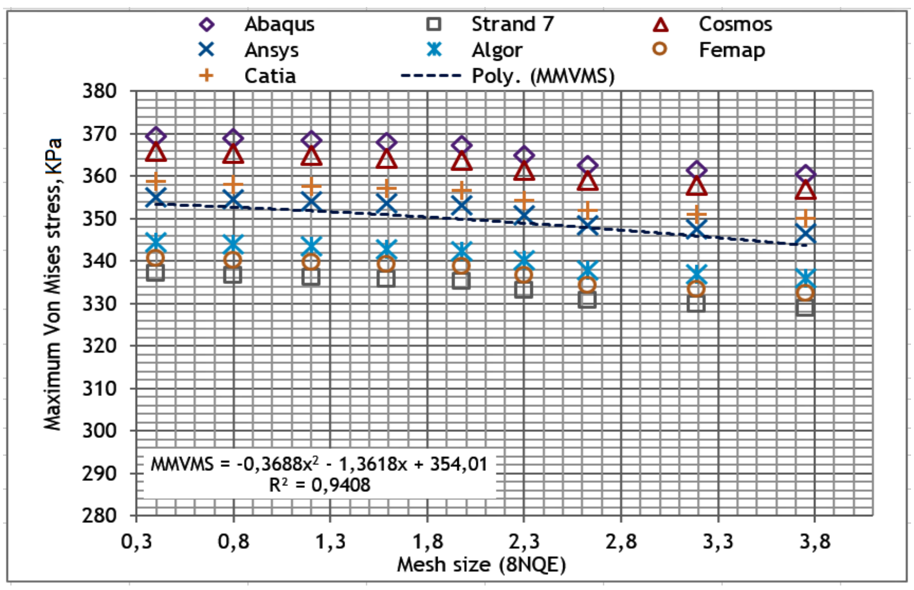


Figure 9. Maximum Von Mises stress versus Mesh size (8NQE)

Fig.10 compares the discrepancies between the higher-order element (8NQE) and the first-order element (4NQE) at various mesh sizes within the range of 0.4 mm to 3.75 mm. The value obtained using higher-order element (8NQE) is greater than the values obtained from the first-order element (4NQE). However, within the range of 0.4 mm to 1.2 mm, the differences were very small about 0.68 %. This shows that as the mesh gets finer both the 8NQE and 4NQE exhibited almost the same MVMS value. This reinforces the previous point that the degree of variance in the MVMS of different FEA solvers would reduce with a reduction in mesh size.

Fig.11 illustrates the percentage of discrepancies displays by each FEA solvers based on higher-order element (8NQE). The discrepancies values of ANSYS, CATIA, and FEMAP are respectively 0.581%, 1.597 %, and 1,451%. Whereas the discrepancies values of ABAQUS, STRAND 7, ALGOR, and COSMOS/M are respectively 2.612%; 4.499%; 2.467%; and 3, 628%. Furthermore, ANSYS, CATIA and FEMAP results are closer to the mean maximum Von Mises stress (MMVMS) than the results obtained from ABAQUS, STRAND 7, ALGOR, COSMOS/M.

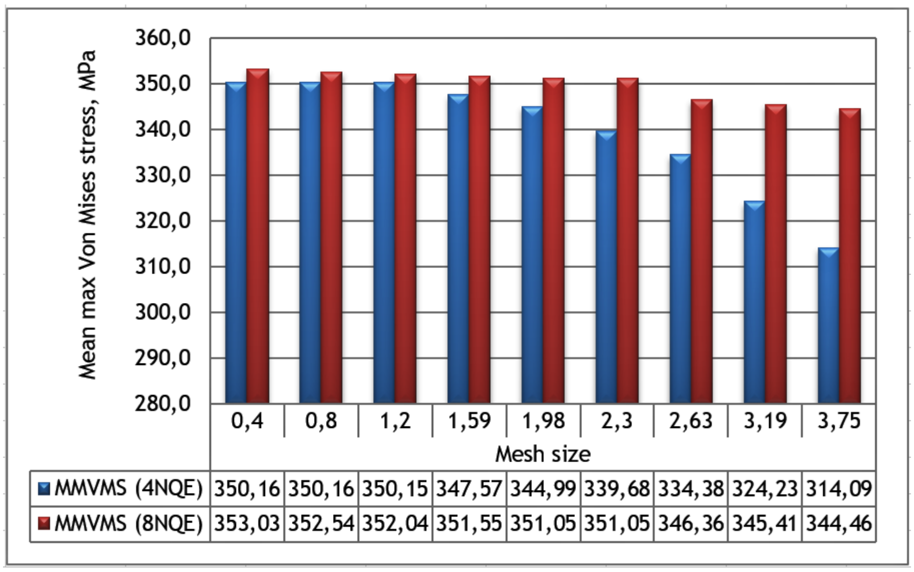


Figure 10. Mean max Von Mises stress versus Mesh size

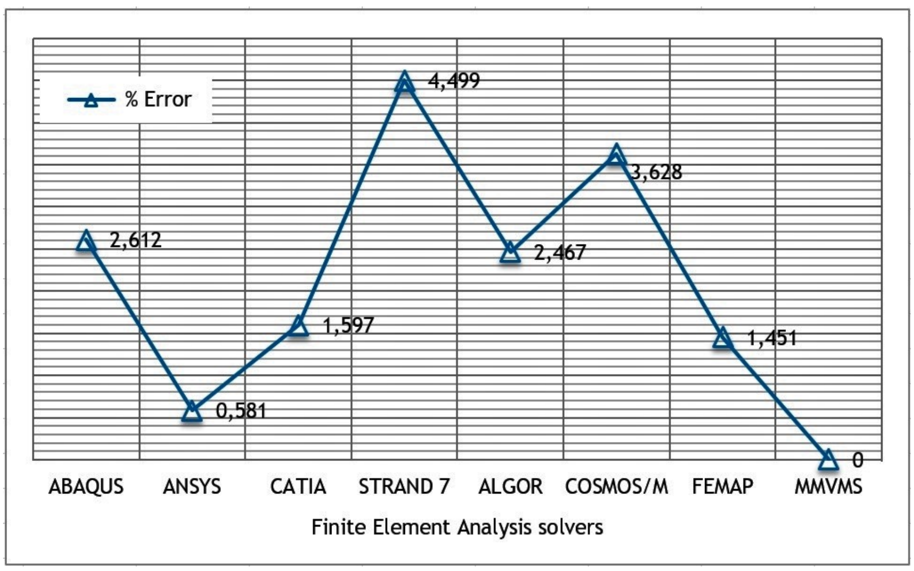


Figure 11. FEA Solvers MVMS percentage error (8NQE)

3.2. Convergence Analysis

The convergence study originates from reducing element size without changing the element order, first-order element (4NQE), and a higher-order element (8NQE). Several increasingly refined models (mesh sizes) were utilized to assess convergence error and convergence analysis. Fig.12 shows the convergence analysis of the maximum Von Mises stresses obtained from several mesh sizes within the range of 0.4 mm to 3.75 mm using 4NQE and 8NQE. It can be observed that, as the mesh sizes reduce, the mean maximum Von Mises stresses values increases. Nevertheless, for the mesh sizes within the range of 0.4mm to 1.2 mm, the values of MMVMS were almost constant. The convergence is attained at 0.8 mm mesh size for 4NQE and the MMVMS = 350.160 MPa. On the other hand, the convergence is attained at 0.4 mm mesh size for 8NQE and the MMVMS = 353.034 MPa.

Fig.13 exhibits an increase in convergence error with mesh size increment. The slope shows converging values with mesh size. The convergence error of 4NQE is higher than the convergence error of 8NQE. At the initial mesh size 3.75 mm the convergence errors were 6.46 % and 1.5 % for 4NQE and 8NQE respectively. As the mesh size is refined, the convergence error reduces. At the smallest mesh size 0.4 mm, the converge errors are estimated at 0.4% and 0.1% for 4NQE and 8NQE respectively. The convergence relied upon how well the genuine stress distribution is represented with the given mesh.

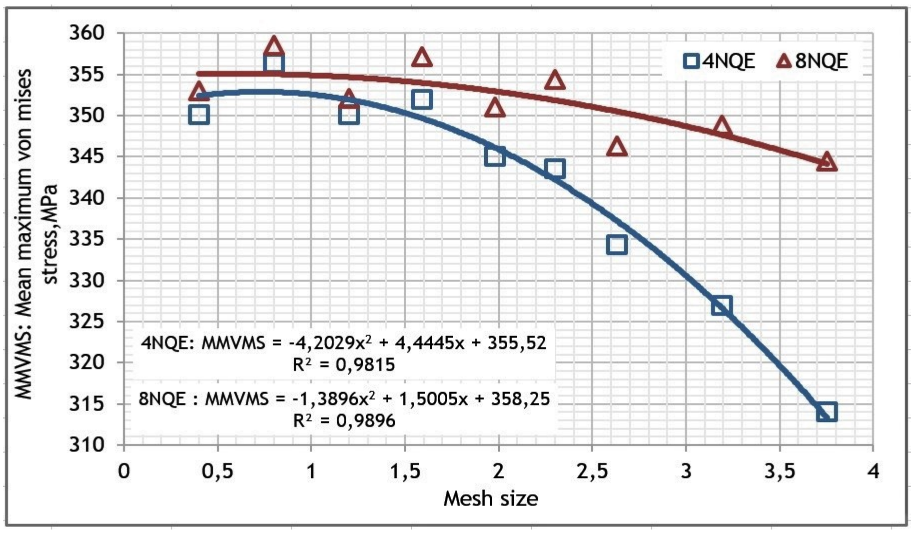


Figure 12. Convergence

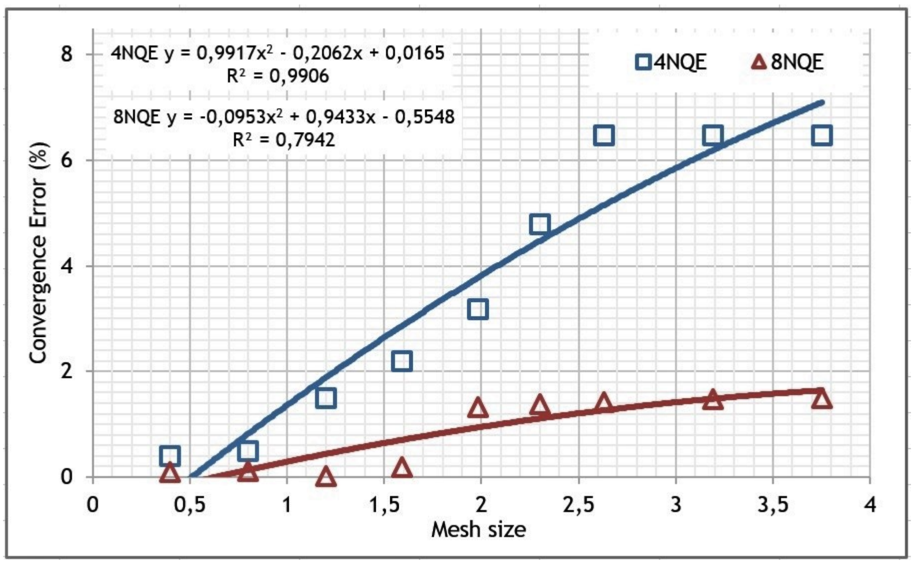


Figure 13. Convergence error

3.3. Relationship between Kn and Mesh Size

The numerical stress raiser factor (Kn) is a dimensionless factor that is utilized to evaluate how concentrated stress is in a material. The knowledge of the stress raiser factor forms the basis for the selection of material. The relationship between the mesh size versus the numerical stress raisers factor for 4NQE and 8NQE are illustrated in Figs 14 and 15.

In Fig.14, it was observed that the mesh size and the type of element significantly influence the stress raiser factor values. The relationship between mesh size and Kn exhibits good correlation with a coefficient of correlation estimated at R^2= 0.995 and R^2=0.941 respectively for 4NQE and 8NQE. Kn increases as the mesh size reduce and vice versa. On the other hand, Fig.15 shows the numerical stress raiser factor (Kn) obtained from the higher-order element (8NQE) and the first-order element (4NQE) at various mesh sizes within the range of 0.4 mm to 3.75 mm. The value of Kn obtained using higher-order element (8NQE) is greater than the values obtained from the first-order element (4NQE). However, within the range of 0.4 mm to 1.2 mm, the differences are very small about 0.68 %. The values of Kn are 2.072 and 2.089 for 4NQE and 8NQE respectively. The mesh quality influences the stress raiser factor value. These outcomes are in harmony with the study conducted by Thohura and Islam (2014) on the impact of finite element mesh quality on stress raiser factor of plates with openings which revealed that fine mesh is increasingly exact contrasted with coarse mesh.

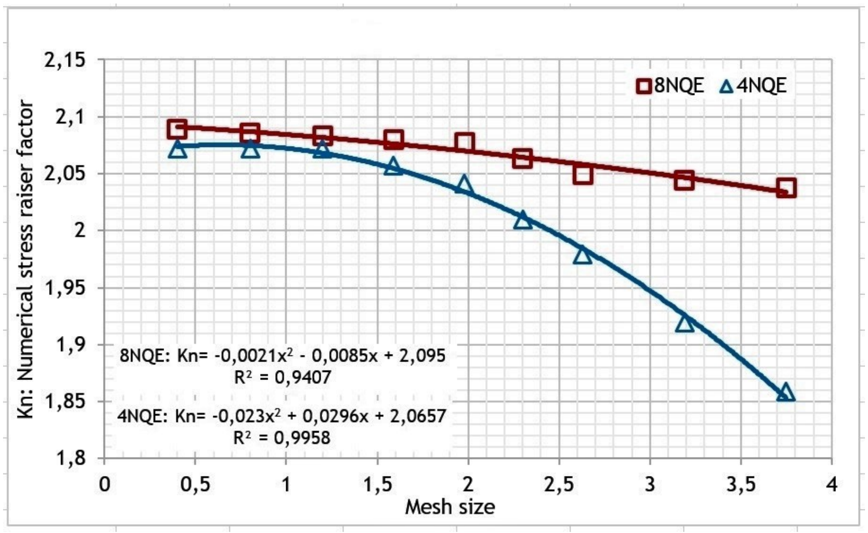


Figure 14. Mesh size Versus Kn

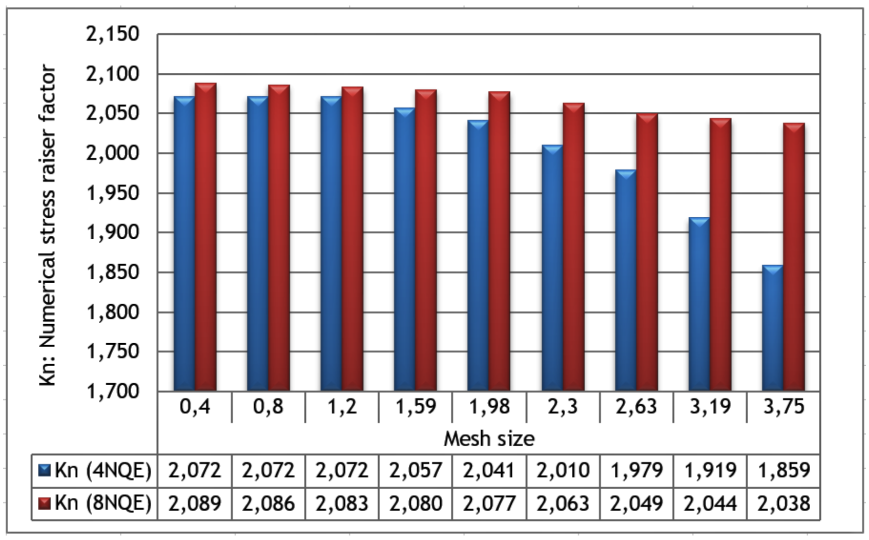


Figure 15. Mesh size versus Kn

3.4. Relationship between Kt & Kn

The theoretical stress raiser factor (Kt) was calculated from the mathematical and analytical equation as establish by Pilkey and Pilkey (2008) and the values of the numerical stress raiser factor (Kn) were obtained from the FEA solvers. Fig 16 and 17 shows the correlation between the theoretical stress raiser factor (Kt) and the numerical stress raiser factor (Kn). The values of the numerical stress raiser factors obtained from the FEA solvers were Kn= 2.072 and Kn=2.089 for 4NQE and 8NQE respectively (Fig.14). These values are smaller than the theoretical stress raiser factor (Kt=2.285) (Equation 1) obtained using the analytical method as proposed by Pilkey & Pilkey [2]. The differences between Kn and Kt were calculated as 10.28 % and 9.382 % for 4NQE and 8NQE respectively. These results are in line with the study conducted by Thohura and Islam (2014) investigating the stress concentration and the displacement of a rectangular plate with a central elliptical hole using the analytical method and ANSYS (FEA). The stress raiser factors obtained from the FEA were smaller compared to the analytical method. These discrepancies between Kn and Kt may be due to the inherent approximated approach use in FEA. Nevertheless, FEA is an efficient and robust tool to model and assess stress raisers' factor with acceptable accuracy.

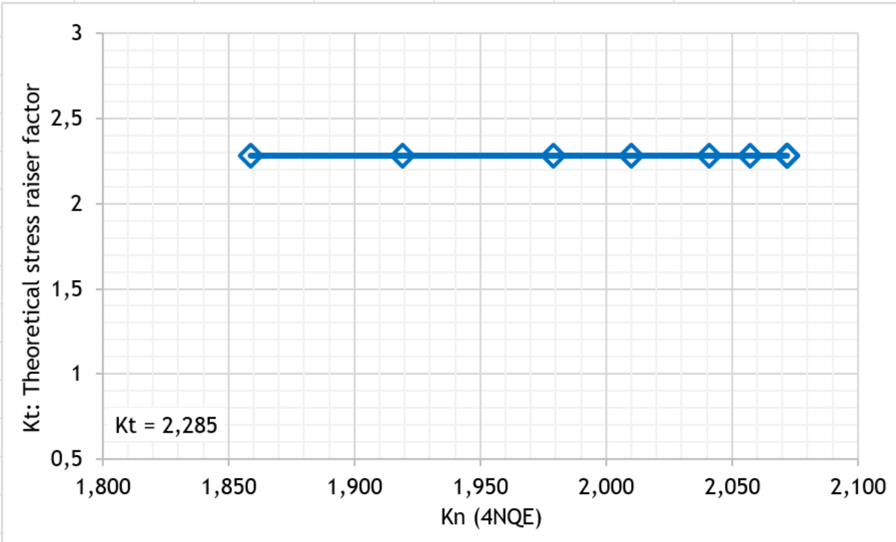


Figure 16. Kn (4NQE) versus Kt

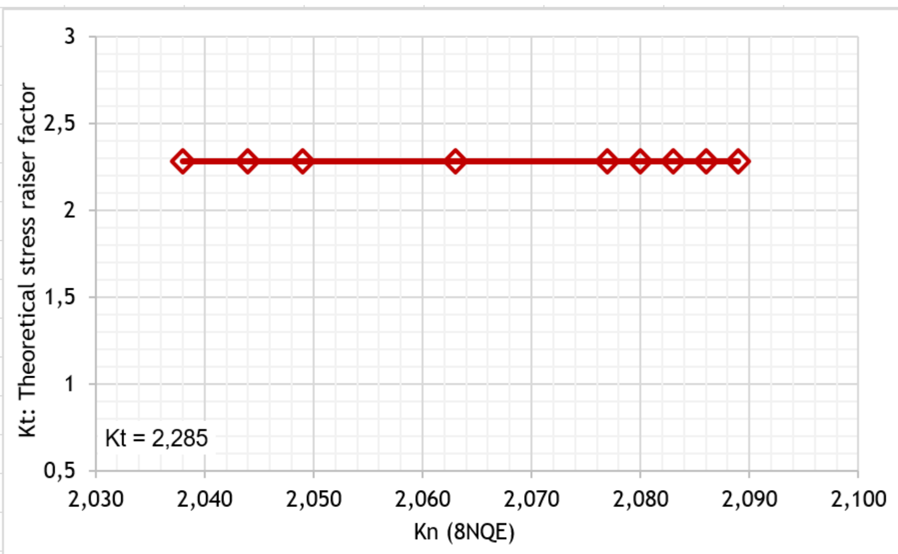


Figure 17. Kn (8NQE) Versus Kt

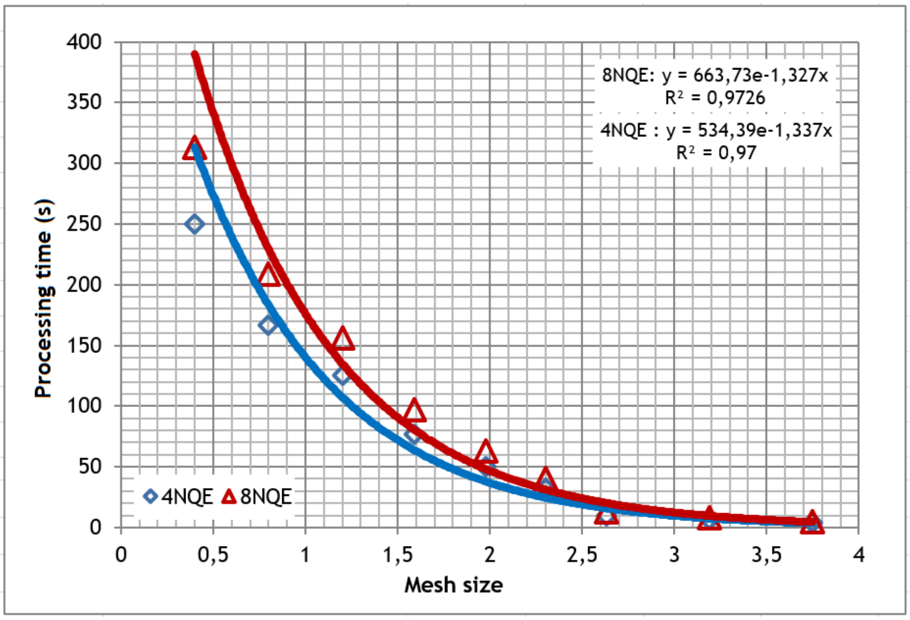


Figure 18. Mesh size versus Processing time

3.5. Correlation between Mesh Size and Computing Time

When performing FEA, the accuracy of the results and required processing time are influenced by mesh size. Finite element models with a fine mesh yield accurate outcomes yet may take longer processing time. Likewise, those finite element models with coarse mesh may lead to less accurate outcomes, yet smaller processing time. As a result, in producing finite element models, the preeminent issue is to pick suitable components size so that the models will yield precise FEA results while saving as much processing time as possible. Fig.18 shows a good correlation between mesh size and processing time. As the meshing size increases the processing time decreases as expected. The coefficient of correlation for 8NQE and 4NQE are respectively R^2=0.973 and R^2=0.970. These observations are in concurrence with the research conducted by More and Bindu (2015) on the impact of stress mesh density on finite element analysis of plate structure, which reveals that small element size yields very precise results, however, may take longer processing time. Likewise, large element size may lead to less precise results but smaller processing time.

4. Conclusions

The values of the numerical stress raiser factors obtained from FEA were Kn=2.072 and Kn=2.089 for 4NQE and 8NQE respectively. The theoretical stress raiser factor Kt=2.285 and the differences between Kn and Kt were estimated as 10.28 % and 9.382 % for (4NQE) and 8NQE respectively. These discrepancies between Kn and Kt may be due to the fact that, FEA is a numerical and approximated approach. Analytical method results must always be compared to FEA for an efficient design. FEA Solver ANSYS results are closer to the MMVMS than the results obtained from other FEA solvers, it displays the smallest percentage error estimated at 0.581 %. The percentage of errors of CATIA and FEMAP are respectively 1.597% and 1.451 %, they are less than 2 %. However, the percentage of error of other software are: ABAQUS, 2.612; STRAND7, 4.499%; ALGOR, 2.467 % and COSMOS/M, 3.628% is greater than 2%. Not only the level of skill of the engineer is critical in FEA, but the type of FEA solver also has an impact on the results.

Acknowledgements

This work is based on the research supported by the South African Research Chairs Initiative of the Department of Science and Technology and National Research Foundation of South Africa (Grant №97994)

REFERENCES

[1] Anderson, T L. 2005. Fracture mechanics - fundamentals and application. 3rd. CRC press.

[2] Clough, R W. 1960. "The finite element method in plane stress analysis." In Proceedings of 2nd ASCE Conference on Electronic Computation, Sept. 8 and 9. Pittsburgh.

[3] Dutt, A. 2015. "Effect of mesh size on finite element analysis of beam." International Journal of Mechanical Engineering 2 (12): 8-10.

[4] Dzogbewu, T C. 2017. Direct metal laser sintering of

titanium alloys for biomedical applications. Doctoral dissertation, Bloemfontein: Central University of Technology, Free State.

[5] Dzogbewu, T C, L Monaheng, I Yadroitsava, W B du Preez, and I Yadroitsev. 2017. "Finite element analysis in design of DMLS mandible implants." Proceedings of the International Conference on Sustainable Smart Manufacturing (S2M 2016), October 20-22, 2016. Lisbon, Portugal.

[6] FEMAP. 2007. FEMAP User Guide. Version 9.3. UGS Corp.

[7] Growney, W D. 2008. "Using FEA Results to Determine Stress Concentration Factors." In 2008 International Ansys Conference. ANSYS, Inc.

[8] Heywood, R B. 1952. Designing by photoelasticity. Chapman & Hall.

[9] Hibbit, K, and S Inc. 2004. Abaqus User Guide V. 6.6. Pawtucket, RI, USA: HKS Inc.

[10] Howland, R C J. 1930. "On the stresses in the neighbourhood of a circular hole in a strip under tension." Philosophical Transactions of the Royal Society of London. Series A, Containing Papers of a Mathematical or Physical Character 229 (670-680): 49-86.

[11] Hutton, D. 2004. Fundamentals of finite element analysis. McGraw-Hill.

[12] Lashkari, M. 1988. Cosmos/m user guide. California, USA: Structural Research and Analysis Corporation.

[13] Lewis, R W, P Nithiarasu, and K N Seetharamu. 2004. Fundamentals of the finite element method for heat and fluid flow. John Wiley & Sons.

[14] Li, F, Y He, C Fan, H Li, and H Zhang. 2008. "Investigation on three-dimensional stress concentration of LY12-CZ plate with two equal circular holes under tension ." Materials Science and Engineering 483 (A): 474-476.

[15] Lui, GR, and SS Quek. 2013. In the finite element method: a practical course, Butterworth-Heinemann.

[16] MatWeb. 2019. "Material property data, AISI-1025 cold rolled steel strip." Available at: www.matweb.com [Accessed 19.05.2019].

[17] Mauk, ML. 2010. Review of the Accuracy of Stress Concentration Factors. Doctoral dissertation, Oklahoma State University.

[18] Melosh, R J. 1961. " A stiffness matrix for the analysis of thin plates in bending." Journal of the Aerospace Sciences 28 (1): 34-42.

[19] Melosh, R J. 1963. "Structural analysis of solids." Journal of the Structural Division 89 (4): 205-248.

[20] Mittal, N D, and N K Jain. 2007. "The optimize design of a square simply supported isotropic plate with central circular hole for reduction of stress concentration subjected to transverse static loading ." Proceedings of ICTACEM.

[21] More, S T, and R S Bindu. 2015. "Effect of mesh size on finite element analysis of plate structure." Int. J. Eng. Sci. Innovative Technol 4 (3): 181-185.

[22] Pan, Z, Y Cheng, and J Liu. 2013. "Stress analysis of a finite plate with a rectangular hole subjected to uniaxial tension using modified stress functions." International Journal of Mechanical Sciences 75: 265-277.

[23] Patle, B C, and D V Bhope. 2012. "Evaluation of stress concentration factors in plate with oblique hole." Journal of Mechanical and Civil Engineering 2 (2): 28-32.

[24] Pilkey, W D, and D F Pilkey. 2008. Peterson's stress concentration factors. John Wiley & Sons.

[25] Rayleigh, L. 1870. On theory of resonance. London: Transactions of the royal society.

[26] Snowberger, D. 2008. Stress concentration factor convergence study of a flat plate with an elliptic hole under elastic loading conditions. Doctoral dissertation, Master thesis, Rensselaer Polytechnic Institute. Connecticut, USA.

[27] Spyrakos, C. 1995. Finite element modeling in engineering practice. Pittsburgh: West University Virginia Press/Algor.

[28] Strand7. 2010. Introduction to the Strand7 Finite Element Analysis Systems. Sidney: Strand7 Pty Ltd.

[29] Systemes, D. 2016. Catia Systems Engineering Dymola. Vélizy-Villacoublay, France: Dassaults Systems.

[30] Thohura, S, and M S Islam. 2014. "Study of the effect of finite element mesh quality on stress concentration factor of plates with holes., 7(08), p.19." Proceedings of the 15th Annual Paper Meet 7 (8): 19-19.

[31] Thompson, M K, and J M Thompson. 2017. ANSYS Mechanical APDL for finite element analysis. Butterworth-Heinemann.

[32] Ukadgaonker, V G, and D B Patil. 1993. "Stress analysis of a plate containing two elliptical holes subjected to uniform pressures and tangential stresses on hole boundaries." Journal of engineering for industry 115 (1): 93-101.

[33] Young, W C, R G Budynas, and A M Sadegh. 2002. Roark's formulas for stress and strain. Vol. 7. New York: McGraw-Hill.