Asian Journal of Research № 1-3, 2020
IMPACT FACTOR
SJIF 6,1
IFS 2,7

ISSN 2433-202x
JOURNAL DOI 10.26739/2433-202x
www.journalofresearch.asia
info@journalofresearch.asia

NUMERICAL ANALYSIS OF AN UNGRADED MILD STEEL STRUCTURAL PLATE USING FINITE ELEMENT ANALYSIS METHOD

Khalid Abdel Naser Abdel Rahim
MSc, BEng (Hons)
Khalid.ar@outlook.com
Department of Civil Engineering, Faculty of Science and Technology, University of Coimbra, Portugal

http://dx.doi.org/10.37057/2433-202x

Issue DOI http://dx.doi.org/10.37057/2433-202x-209-2020-1-6

Article DOI http://dx.doi.org/10.37057/2433-202x-2020-1-6-22

ACKNOWLEDGEMENT: I hereby Khalid Abdel Naser Abdel Rahim acknowledge that this research manuscript is my own work. Firstly, I would like to thank my family for their moral support. In addition, many thanks to Dr. Ian Mackie a senior lecturer at the University of Dundee. My greetings are also extended to all the staff of the Department of Civil Engineering in the University of Dundee for their technical support.
KEYWORDS: FEA, structural design, structural analysis, stresses, mesh improvement, LUSAS software.

ABSTRACT: This research focuses on solving a finite element analysis problem using LUSAS software. The main objective of the research was to find the most suitable position simulation of the hole in an ungraded mild steel structural plate in terms minimum maximum stress and to demonstrate how such problems are analyzed using LUSAS software. This has been carried out by introducing a two dimensional structure with a 20mm hole inside it. Besides, the 2D structure has been modelled and analyzed using computational modelling software (LUSAS) to obtain plane stresses. Furthermore, the 20mm hole has been positioned in the structure in order to minimize the maximum stress in the structure. The problem was tackled and analyzed with the aid of LUSAS software. Generally, LUSAS works by analyzing equations using finite element method and is consider one of the best structural analytical tools worldwide. Moreover, this research has concluded that (1) the position of the hole has a direct influence on the maximum stress in plate, (2) the more and smaller the element in terms of size, the more accurate results, (3) the elements in regions of high stresses should be smaller than those in regions of low stress, (4) the maximum stress in the plate is at the curve side of the structure and around the hole, (4) quadratic mesh provides better results than linear and was used due to the curves sides of the structure, (5) applying the appropriate number of divisions on the plate maintains better and sensible mesh and (6) dividing the mesh improves the results and gives mesh better appearance. Additionally, fur-

ther investigations on finite element analysis should be conducted in the future to validate structures with different geometries, boundary conditions and material properties.

Introduction

Overview

This report is about finite element analysis of a problem using LUSAS. The finite element analysis is a process of obtaining a numerical solution for a certain analytical problem. This method works by dividing the plate or structure in to a number of elements and illustrates each of the elements performance in an easy manner. The elements are connected with each other by nodes (Cook, 2002). In the first section of this report, the analysis specification will be presented with a description of the structure and analyses carried out such as, explanation of the analysis stages and work done by the author from co-ordinates of the structure, position of hole, mesh design and associated stresses. Moreover, the reasons behind carrying out the analysis for the structure will be explained. The results obtained for the structure will be presented, along with a brief discussion on what they indicate and mean. Furthermore, the results will be discussed in terms of plane stress and maximum stress. Finally, the conclusions will be stated clearly with an answer to the client question and reliability of results obtained.

Main aims of the paper

The main objectives of the paper were:

To analyze the structure using finite element in an effective manner.

To calculate the maximum stresses at different positions of the hole and find out simple.

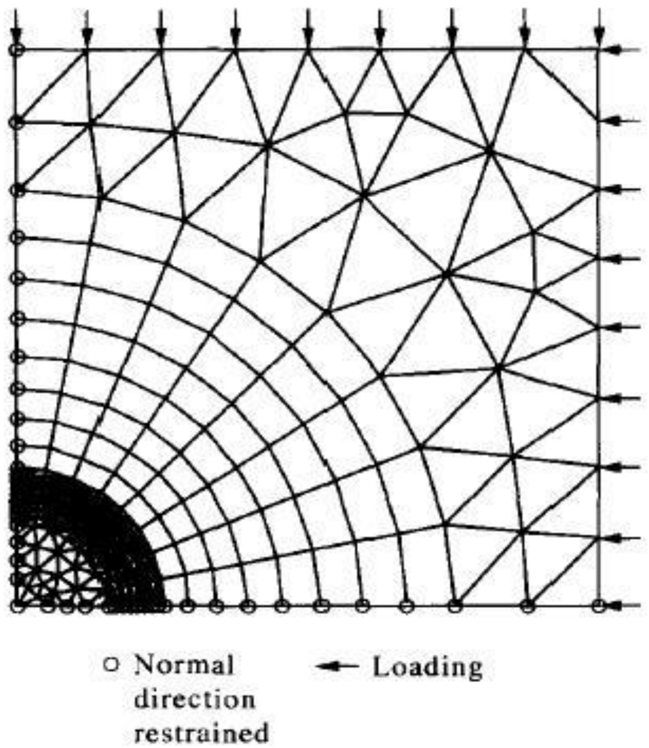
To find a suitable position of the hole to minimize the maximum stresses in the plate and determining the associated stresses.

To design appropriate mesh that looks sensible using finite element.

To present and discuss the results obtained for the analyzed plate.

Literature review

Jusheng and Nan (1993) presented a brief review on the techniques used by the finite element analysis softwares.



Zou et al. (1996) has conducted numerical analysis on different breakouts in a barehole. They did a stress distribution analysis on a numerical squared shape model with a borehole in the center of the object. Moreover, the dimensions of the analysised square were 540cm by 540cm with a radious of 30cm for the center barehole. In addition, their analysis focused on determining the breakouts alignment and boreholes constancy under the influence of different strees angles. Considering the symetry in the analyised structure, the analysis was carried out on quarter of the square as shown in figure 1 which illustrates a quarter section of the analyised model, along with the distribution of the finite element mesh and boundary conditions. Furthermore, their study concluded that stress angle has a direct influence on the breakouts alignments of the boreholes.

ISSN 2433-202x
JOURNAL DOI 10.26739/2433-202x
www.journalofresearch.asia
info@journalofresearch.asia

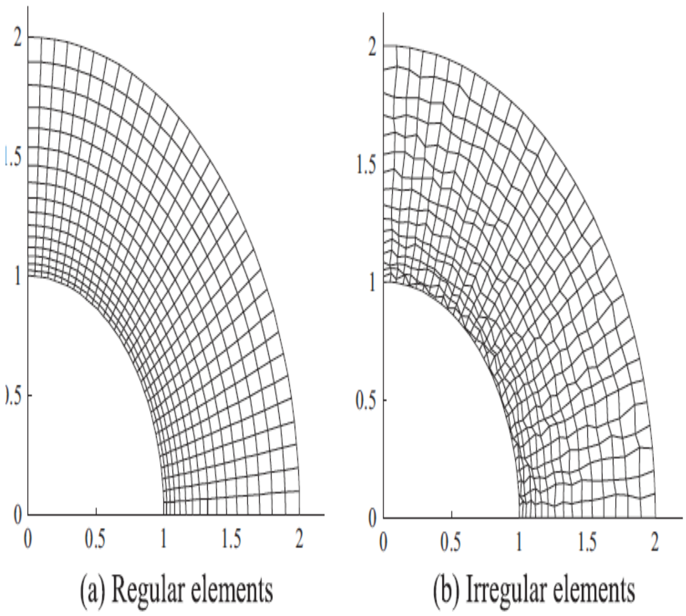
Figure 1. The arrangement of the finite element mesh by Zou et al. (1996).

Hamouda et al. (2001) did finite element analysis using LUSAS on the geometrical behavior on cooling steel material class ST50. Moreover, the analysis was carried out by modeling three different mesh elements; these are cylinder, cone and pyramid. However, the research focused on determining the temperature and residual stress of steel material.

Mackerle (2002) has presented the review of using finite element analysis and boundary element methods and the difference in application between them. Mackerle (2002) has mentioned different finite element analysis programs which are used such as ANSYS, SAP, DIANA, ABAQUS and LUSAS. However, his review did not present any finite element results obtained using these software's.

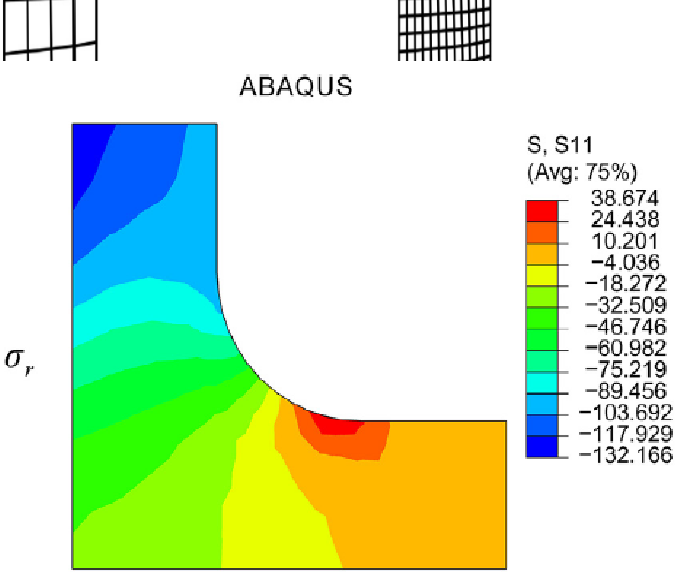
On the other hand, Wan et al. (2016) investigated the axisymmetric problems using refined finite element method. Furthermore, the finite element analysis was carried out on a thick walled hollow cylinder by refining and smoothening the mesh elements of the model as demonstrated in figure 2. The study concluded that the best method to obtain high accuracy in results is by refining and smoothening the elements mesh of the finite element models. In addition, irregular mesh elements could cause errors and unrealistic results during analysis. Additionally, the research recommended further investigations in the future on refining meshing elements for 3D models.

Figure 2. The arrangement of the finite element mesh where (a) regular and (b) irregular elements by Wan et al. (2016).



Rao (2018) has presented the use of ABAQUS, ANSYS and MATLAB software's to solve finite element analysis problems. Moreover, Rao (2018) has discussed the stages and steps to be carried out to conduct finite element analysis using these programs, including the general use. In addition, examples were given on the use of each software in terms of modeling and analysis. However, the examples did not focus on a specific problem and were general. Such examples given by Rao (2018) included the analysis of structural members under loads. Besides, the analyzed examples were majorly on the heat and temperature transfer. Finally but never the least, the presented studies by Rao (2018) seemed to be more like educational material and tutorials for using software's to solve finite element analysis problems.
Zhou et al. (2019) proposed a new finite element approach to analysis axisymmetric problems. The proposed approach was majorly based on a Hybrid Fundamental Solution (HFS). Furthermore, the analysis was validated on a thick-walled hollow cylinder model using various simulations on mesh elements arrangements and number. Moreover, Zhou et al. (2019) has used ABAQUS finite element analysis software to investigate and validate his proposed approach. In addition, fundamental solutions approaches were conducted as part of the study. Figure 3 demonstrates a cross section of the thick-walled hollow cylinder with 48 and 432 elements mesh. While figure 4 shows the contour map of the distribution of stresses in the analyses cross section model. Finally, a comparison between the distribution of stresses in the cross sectional model

ISSN 2433-202x
JOURNAL DOI 10.26739/2433-202x
www.journalofresearch.asia
info@journalofresearch.asia

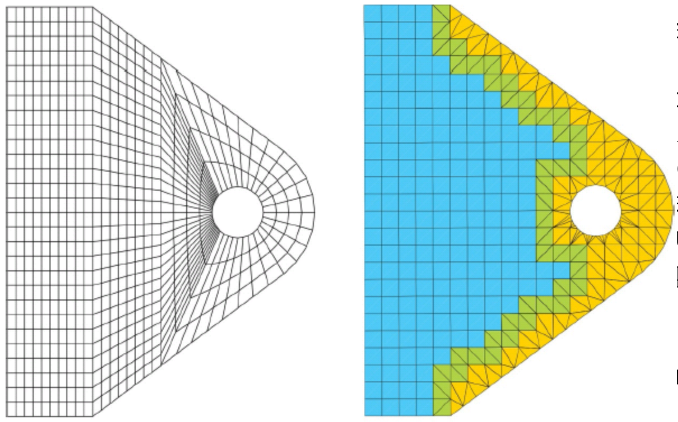


2) 432 elements ABAQUS and 48 elements
l approach.

sh of the thick-walled hollow cylinder with

in the analyses cross section model by Zhou

esearches conducted on Automatic Meshing pattern for finite element analysis. In addi- of AMORE in finite element analysis soft- elements arrangement. Including compari- own in figure 5. The review concluded that ern for all aspects of finite element analysis. uture studies has to be done to take the full pping and Regular Elements (AMORE) pat-



(a) Traditional mesh (b) AMORE mesh

nd (b) Automatic Meshing with Overlapping
19).

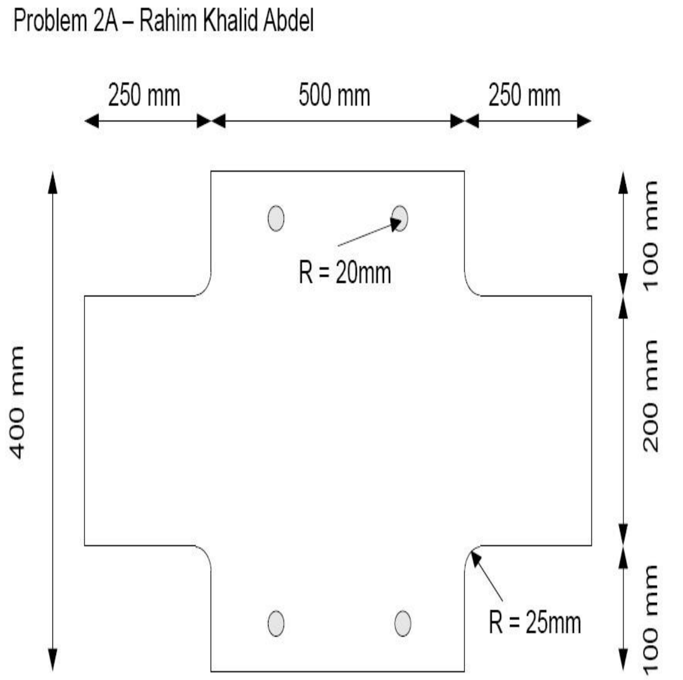
It was not possible to put this research into the scientific context, neither it was possible to compare the results obtained in this research with the other recent investigations due to massive variation between the studies as below:

- Different types of finite element analysis.
- Different boundary conditions.
- Different structural geometries.
- Different material properties.

Description of the structure

The structure is made from ungraded mild steel (KN, m, t, s, c) with a thickness of 10mm (0.01m). As shown in figure 6 drawing of structure (Problem 2A) the geometry of structure has some curves of a 25mm radius and four shaded holes of 20mm radius. Also the structure is 400mm in height and 1000mm in width and has four cut-outs at the corners. Each of the cut-outs has a dimension of 100mm in height and 250mm in width. Furthermore, the plate is subjected to a uniformly distributed load applied at each end. According to the author, the applied loading was assumed to be 100KN/m. Mackie (2010) has stated that "…element type needed depends on the geometry of structure and applied loading". Therefore, the element type that has been used was quadratic. This was chosen because of the curve side which can not be analysed using linear analysis. Additionally, the quadratic element maintains better modelling and more accuracy.

Figure 6. Shows the geometry of structure to be analyzed.



Analysis specification

Purpose of carrying out the analysis

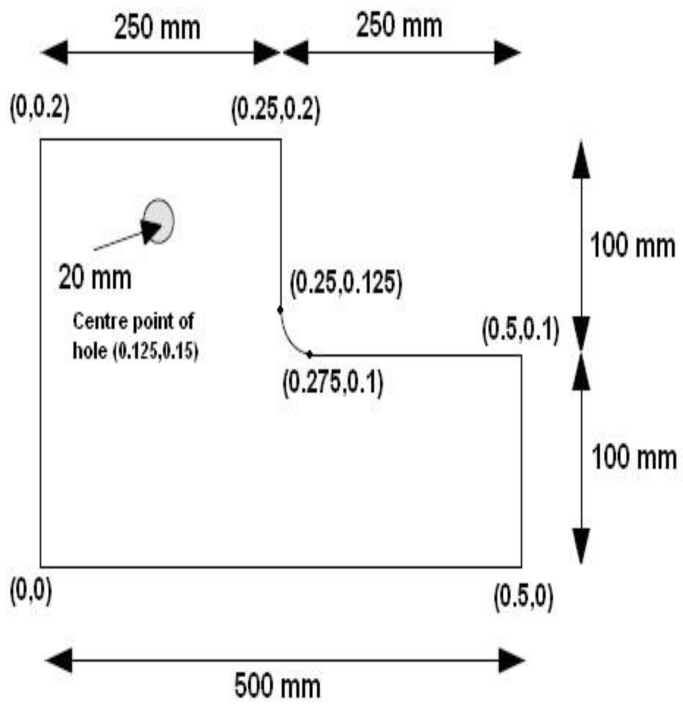
The main purpose of the analysis is to calculate the maximum stresses of the plate at different positions of the hole and to find the appropriate position of the hole with the minimum maximum stress. Another important aspect of the analysis is to produce a sensible and good looking mesh free of dodgy elements.

Procedures of the analysis carried out

Description of the stages and work done

As can be seen in figure 7 the structure can be divided into four equal quarters. Since each quarter of the plate has the same dimensions and shape, the author has decided to model and analysis one quarter of the plate. The analysis of one quarter of the structure was decided because of symmetry it is assumed that each quarter will have the same type of elements, mesh design and stresses.

Figure 7: Shows the co-ordinates of one quarter of the structure.

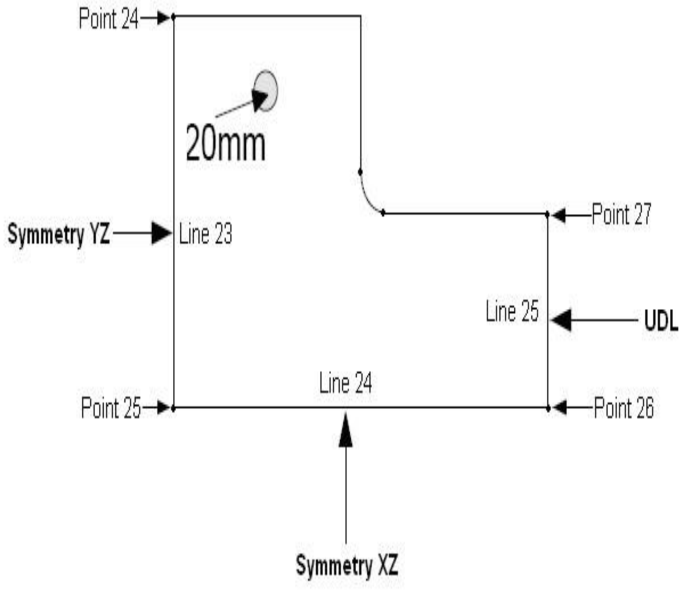


Initially, the co-ordinates of structure have been identified and points were drawn as shown in figure 7. Then the points were joined together to form lines as part of modelling and the plate surface was defined. The next step was identifying material and thickness of the plate as indicated in coursework sheet. The material was recognized as mild steel ungraded (KN, m, t, s, c) and thickness was 10mm (0.01m). After that they were both applied on the plate.

Finite elements analysis

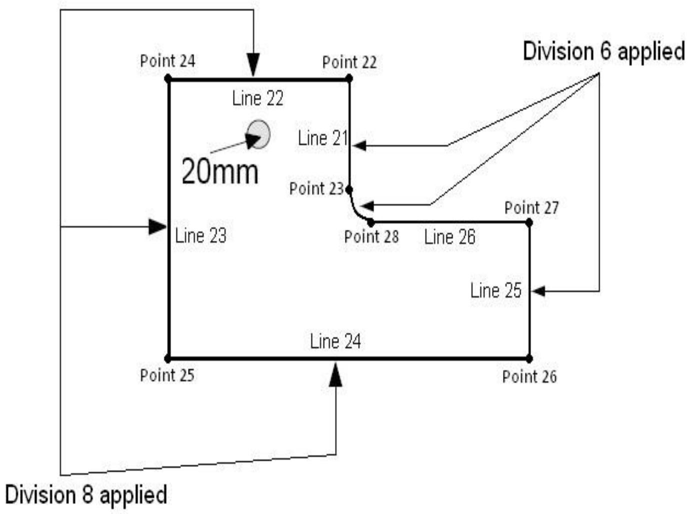
The linearity of loads with respect to displacement in the plate clarifies that the stiffness matrix is symmetric. The symmetry was used as indicated in the coursework sheet. In addition, all the data and boundary conditions such as, restraints and loads were applied correctly in the finite element model to satisfy equilibrium and compatibility conditions over the whole plate (Dawe, 1984). This was achieved by creating a uniform distributed load with an applied load of 100KN/m. The uniform distributed load was applied on line 25 (points 26, 27) as shown in figure 3. In addition, symmetry XZ was applied on x – axis line 24 (points 25, 26) and symmetry YZ was applied on y – axes line 23 (points 24, 25) with respect to figure 8. The next step that has been carried out was producing and applying mesh on the whole plate. Due to the curve side in the plate and to achieve better results, the mesh was identified as quadratic at interpolation order and element shape was quadrilateral. Additionally, the type of mesh chosen was irregular and plane stress for structural element type.

Figure 8: Shows the applied symmetry XZ, YZ and UDL.



The two main aspects of solving this problem were obtaining a sensible and good looking mesh with less maximum stress. These aspects were fulfilled by applying the appropriate number of divisions in zones with the maximum stresses. At the same time consideration was taken in to account to the limited number of division that can be applied, due to the student software version which only allows up to a total number of 500 divisions. Generally, the number of divisions that has to be applied depends on the geometry and size of the plate. Dawe (1984) has stated that "… the elements in regions of high stresses should be smaller than those in regions of low stress…" Accordingly, it was found that the maximum stress in the plate is at the curve side of the structure and around the hole. Thus, the problem was tackled by applying more divisions on the curve side of the structure (point 23 - 28) and around the hole. In accordance to figure 9, six divisions were applied on lines 21, 25 and 26 and eight divisions were applied on lines 22, 23 and 24. In addition, twelve divisions were applied around the hole due to the high maximum stress at that region.

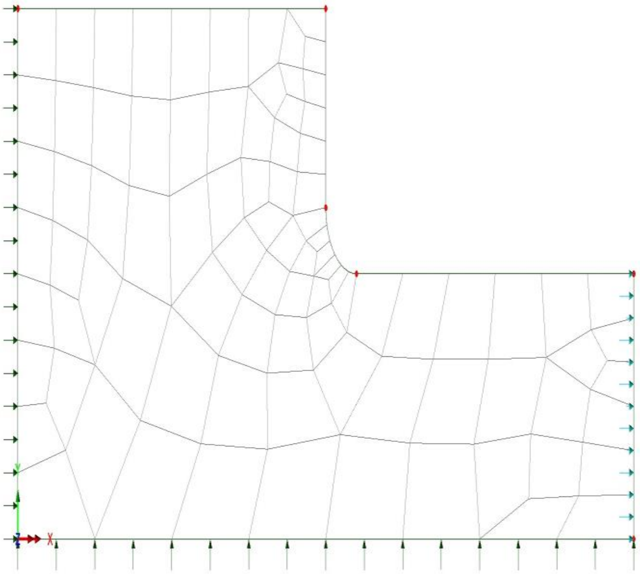
Figure 9: Shows the applied divisions.



After applying the appropriate divisions and before creating the hole, the plate was analyzed to find the zones with the maximum stress as revealed in figure 10. The results of the analysed plate before creating the hole were divided in to 4 zones. As illustrated in figure 11, zone 1 had the lowest maximum stress and zone 3 showed the highest maximum stress in the plate especially in the curve (points 23 and 28). Thus, it was assumed that the appropriate position of the hole will be within zone 1. Afterwards, the hole was created with a start position coordinates of (0.225, 0.195, 0), centre point (0.225, 0.175, 0) and end point (0.225, 0.195, 0). Subsequently, a new surface of the whole structure including the hole was created. Thus, the hole is part of structure after creating the new surface.

Figure 10: Shows the mesh and associated stresses diagram of the plate with out the hole.

ISSN 2433-202x
JOURNAL DOI 10.26739/2433-202x
www.journalofresearch.asia
info@journalofresearch.asia



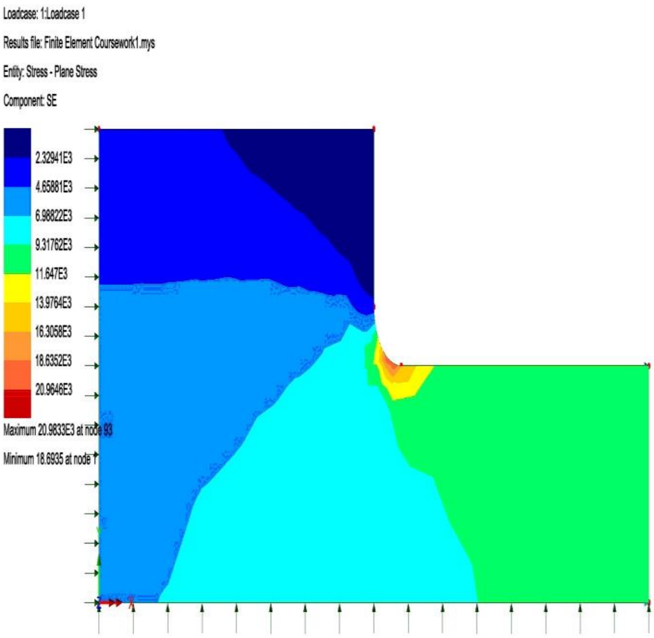
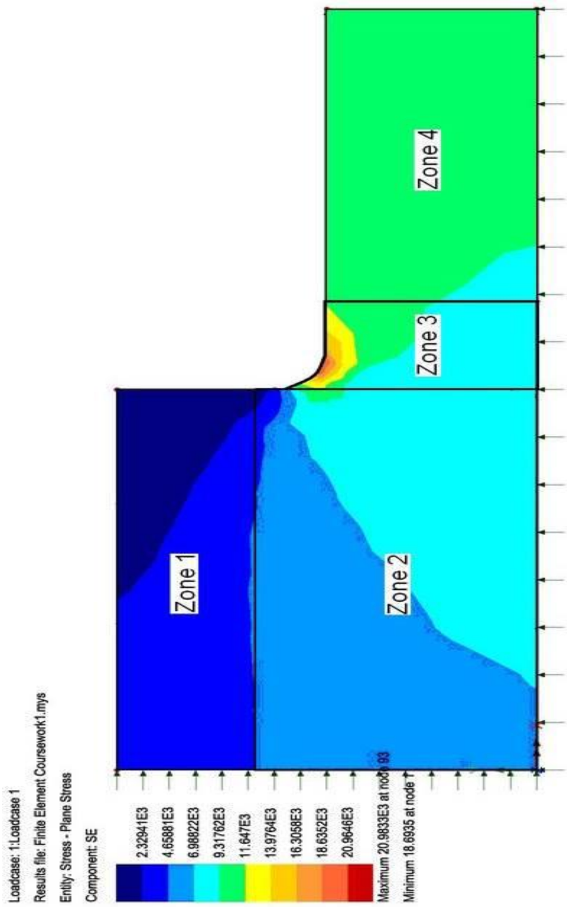


Figure 11: Shows the zones 1 to 4 according to the maximum stress.



Position of the hole

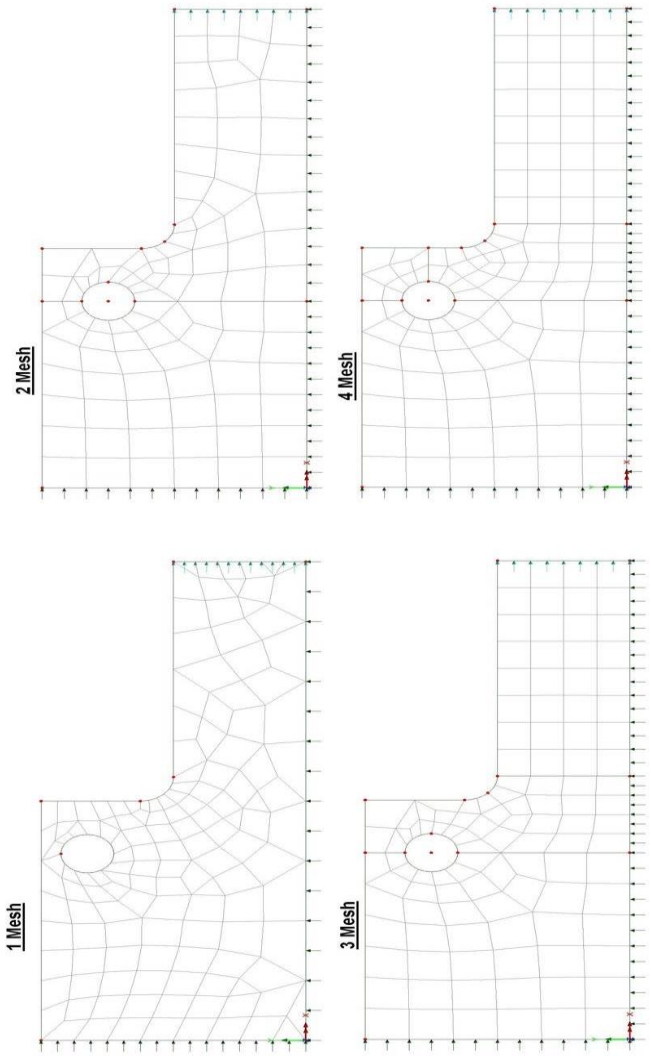
The next step after creating the hole was moving it within the plate. This was done using trail and error method to determine the best location of the hole. The hole was moved from a start position of (0.228, 0.198, 0.0), centre point of (0.228, 0.178, 0.0) and end point of (0.228, 0.198, 0.0). Initially, the hole was moved in 36 different positions within zone 1 as shown in figure 11. In order to be more confident the hole was also moved in 66 different positions within the other zones (zone 2 to 4) to make sure that the appropriate position with the less minimum maximum stress within the plate is at zone 1. This was done by moving the hole in both x and y directions. At each position the hole was moved 20mm either in x direction, y direction or both. For each position the analysis was carried out to determine the maximum stress in the plate at a specific position of the hole. Firstly, the problem was solved by running the program after moving the hole. Secondly, the contour results were identified as stress – plane stress for entity and component of SE. Subsequently, the range of results and maximum stress were presented by the program as a contour diagram. The results of the 102 stages were then compared in terms of the maximum stress. Afterwards, the 36 stages moved in zone 1 were evaluated with the 66 stages moved in zone 2 to 4. Furthermore, the results will be discussed more in the results section of this report.

Asian Journal of Research № 1-3, 2020
IMPACT FACTOR
SJIF 6,1
IFS 2,7
ISSN 2433-202x
JOURNAL DOI 10.26739/2433-202x
www.journalofresearch.asia
info@journalofresearch.asia

Mesh design

Producing a fine mesh was one of the most important factors of solving the problem. The problem of refining the mesh has been resolved by dividing the mesh. It is known that the smaller the element in terms of size, the more accurate results. Thus, dividing the mesh was decided to obtain smaller elements in size which will aid in both improving the look of the mesh and in the accuracy of the results. The mesh was divided after identifying the suitable position of the hole. This was carried out in four phases. As shown in figure 12, the first phase was the fine mesh design which was refined after determining the appropriate position of the hole. The second phase was dividing the mesh in to two individual surfaces. Hence, the mesh looked better after the division. The third phase was established by creating an additional mesh in the lower hand side of the plate as illustrated in figure 13. This was decided due to the rectangular shape of this part of the plate. The final phase consisted of four individual surfaces. This phase was carried out for a better looking mesh with smaller element which enhanced the accuracy of the results. Generally speaking, the fourth mesh phase diagram looks sensible and free of dodgy elements.

Figure 12: Shows the mesh for the four phases.



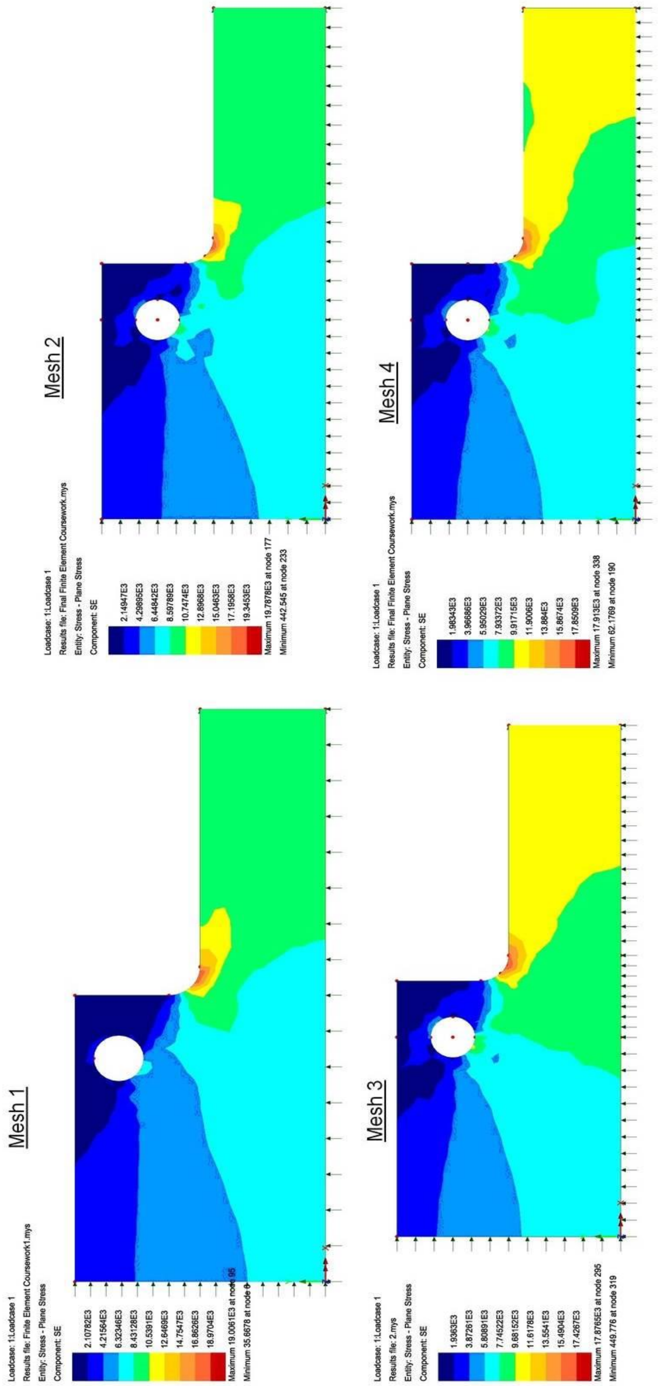
Associated stresses of the four phases

The maximum stress for the one mesh phase was 19.0061E3 kN/m2 at node 95. On the other hand, the maximum stress for two mesh phase was 19.7878E3 kN/m2 at node 177. Due to the increase in maximum stress after dividing the mesh in two parts it was decided to divide the mesh in to a third mesh phase. In addition, the maximum stress for three mesh phase was 17.8765E3 kN/m2 at node 295. This indicates that the stress is minimized after dividing the mesh in to three meshes. Afterwards, the mesh was again divided in to a four mesh phase. This was done in order to obtain smaller element sizes, thus, more accurate results. Additionally, the maximum stress for four mesh phase was 17.913E3 kN/m2.

Asian Journal of Research № 1-3, 2020
IMPACT FACTOR
SJIF 6,1
IFS 2,7
ISSN 2433-202x
JOURNAL DOI 10.26739/2433-202x
www.journalofresearch.asia
info@journalofresearch.asia

Figure 13: Shows the associated stresses diagram for the four phases.

Presentation of results



Results

Table 1. Illustrates the movement of the hole in x and y directions at each of the 36 positions for zone 1 along with the maximum stress for each stage.

ISSN 2433-202x
JOURNAL DOI 10.26739/2433-202x
www.journalofresearch.asia
info@journalofresearch.asia

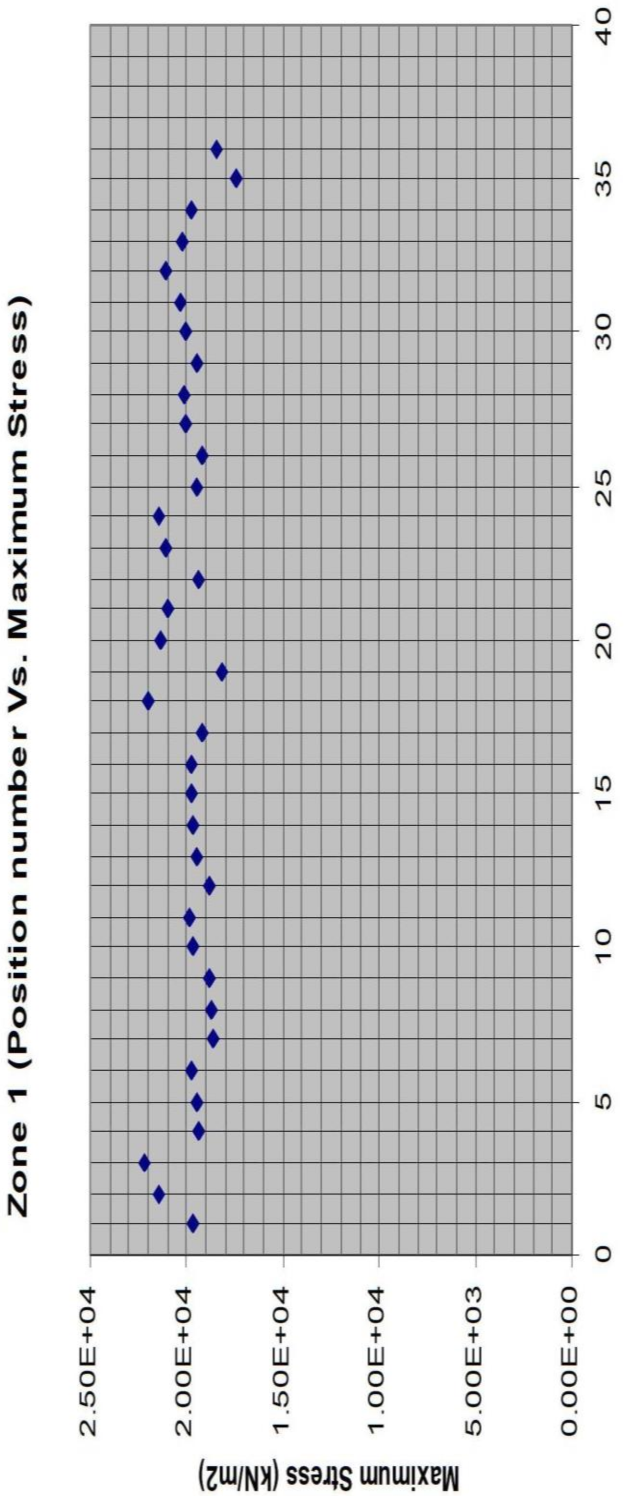
Position	x-direction (m)	y-direction (m)	Max. Stress (kN/m2)
1	Start point	Start point	19.628 E3
2	-0.02	-	21.481 E3
3	-0.04	-	22.180 E3
4	-0.06	-	19.357 E3
5	-0.08	-	19.454 E3
6	-0.10	-	19.760 E3
7	-0.14	-	19.635 E3
8	-0.16	-	19.680 E3
9	-0.18	-	19.827 E3
10	-0.20	-	19.645 E3
11	-0.20	-0.02	19.840 E3
12	-0.18	-0.02	19.832 E3
13	-0.12	-0.02	19.448 E3
14	-0.08	-0.02	19.650 E3
15	-0.06	-0.02	19.740 E3
16	-0.04	-0.02	19.726 E3
17	-0.02	-0.02	19.157 E3
18	-	-0.02	21.991 E3
19	-	-0.04	19.207 E3
20	-0.02	-0.04	21.370 E3
21	-0.04	-0.04	20.956 E3
22	-0.08	-0.04	19.380 E3
23	-0.10	-0.04	21.024 E3
24	-0.14	-0.04	21.422 E3
25	-0.16	-0.04	19.522 E3
26	-0.18	-0.04	19.237 E3
27	-0.20	-0.04	20.038 E3
28	-0.20	-0.06	20.087 E3
29	-0.18	-0.06	19.445 E3
30	-0.16	-0.06	20.056 E3
31	-0.12	-0.06	20.349 E3
32	-0.10	-0.06	21.096 E3
33	-0.08	-0.06	20.243 E3
34	-0.06	-0.06	19.750 E3
35	-0.04	-0.06	19.006 E3
36	-0.02	-0.06	19.464 E3

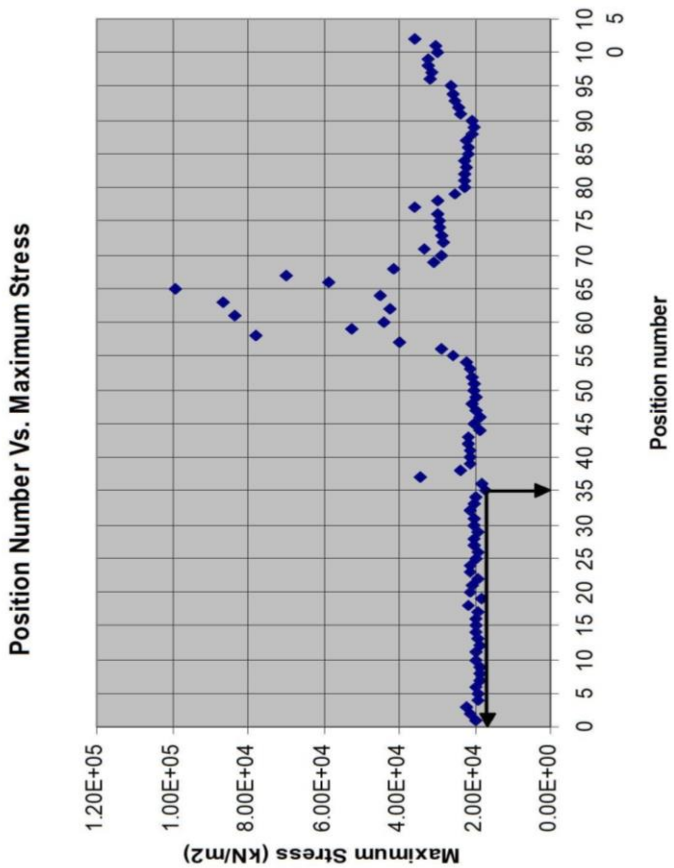
Discussion of the results

The results were obtained from LUSAS as a contour map which shows the distribution of the stresses in all regions of the plate. The analysis of the hole was carried out at 102 different positions within all the zones of plate. It was discovered that as the hole is moved towards the edges of the structure and curve the maximum stress increases and the opposite is true. The results illustrate the movement of the hole from a start point with a start position of (0.228, 0.198, 0.0), centre point (0.228, 0.178, 0.0) and end point (0.228, 0.198, 0.0). As shown in table 1, the hole was moved in 36 different positions within zone 1. The hole was moved 20mm each time either in x – direction, y – direction or both as revealed in figure 14. Each position had different value for maximum stress. For instance, position 2 was moved -0.02 meters (20mm) in x – direction and the maximum stress at this position was 21.4812E3 kN/m2. Accordingly, figure 16 illus-

ISSN 2433-202x
JOURNAL DOI 10.26739/2433-202x
www.journalofresearch.asia
info@journalofresearch.asia

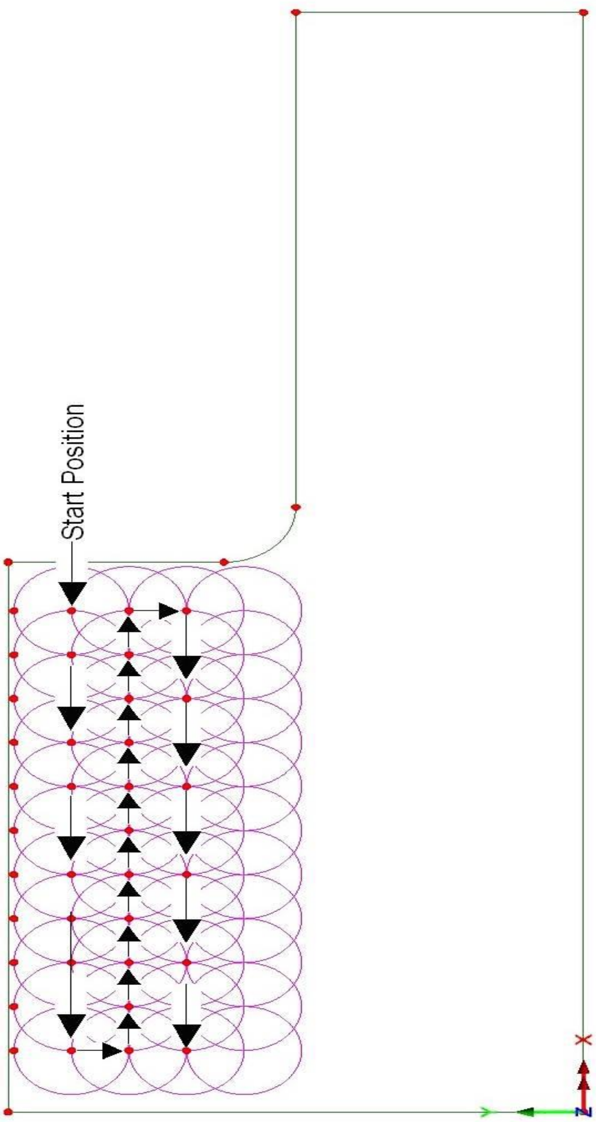
trates the positions moved in all zones of the plate. In addition, the lowest maximum stress in zone 2 was 20.0897E3 kN/m2 at position 46 and highest was 36.1768E3 kN/m2 at position 37. Conversely, the lowest maximum stress in zone 3 was 25.9085E3 kN/m2 at position 56 and highest was 100.0981E3 kN/m2 at position 65. Thus, zone 3 is the worst place for the hole to be position in. Moreover, the lowest maximum stress was 21.3645E3 at position 89 and maximum was 38.7124E3 at point 77. On the other hand, figure 15 shows the maximum stress values at different position within zone 1. The maximum stresses were slightly different and the minimum maximum stress in this zone and in the whole structure was at position 35 with a value of 19.0061E3 kN/m2.





ISSN 2433-202x
JOURNAL DOI 10.26739/2433-202x
www.journalofresearch.asia
info@journalofresearch.asia

Figure 14: The movement of the hole within the structure.



The author believes that the results are valid to some extend. This is due limitation usage of number of divisions applied and to some errors in the software which may lead to not very accurate results. After moving the hole in 102 different positions within the plate (zones 1 -2), carrying out the associated stresses analyses for each position and comparing the results, it was found that position 35 is the optimum position with the lowest maximum stress. Moreover, the contour map results in figure showed that the minimum stress is at zone 1 where the optimum position was found to be.

Figure 15: Position number vs. maximum stress at zone 1.

Figure 16: All the positions number moved vs. maximum stress within structure.

Figure 17: The final four phase mesh diagram.

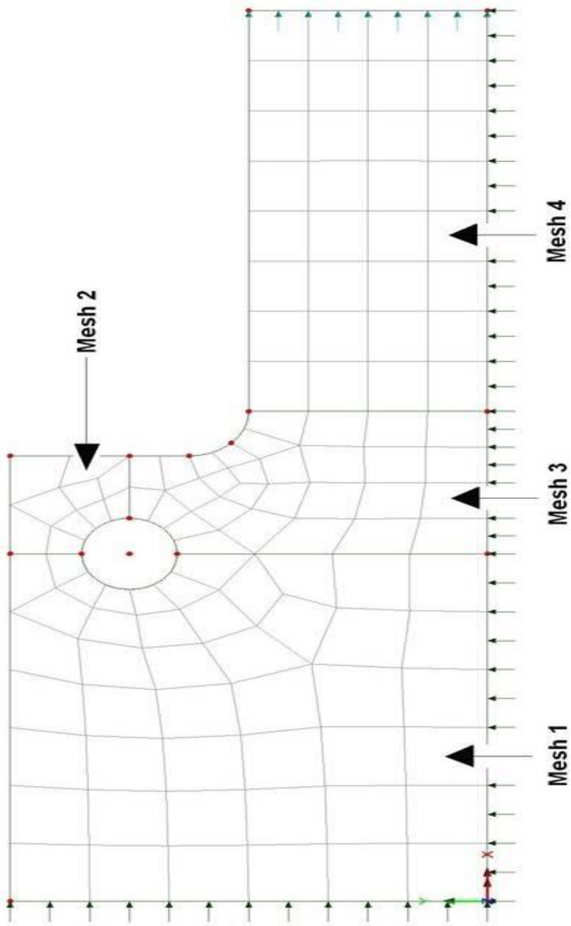
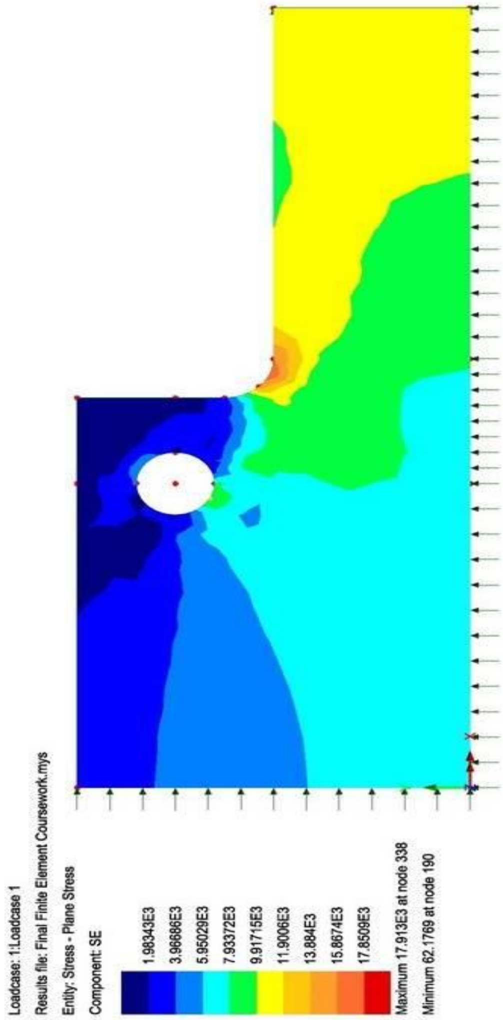


Figure 18: Shows the distribution of stresses contour map for final mesh.

Conclusion

The finite element analysis of a problem using LUSAS was set to find the most suitable position of the hole in terms minimum maximum stress. It was found that different positions of hole have different effects on the maximum stress value, for example, the maximum stress was increased significantly when the hole was moved in zone 3 and had the highest values at that zone. The mesh was divided in to four individual surfaces for the production of sensible mesh which is free of doggy elements and accurate results. In addition, the maxi-

Asian Journal of Research № 1-3, 2020
IMPACT FACTOR
SJIF 6,1
IFS 2,7
ISSN 2433-202x
JOURNAL DOI 10.26739/2433-202x
www.journalofresearch.asia
info@journalofresearch.asia



mum stress was moderated in value when the hole was moved in zone 2 and 4. Thus, the appropriate position was found to be in zone 1.

The results obtained revealed that position 35 is the optimum position in the structure. Furthermore, figures 17 and 18 shows the final mesh produce along with a contour map of stresses distribution in structure. This position was found after analyzing and comparing 102 different positions in the whole structure. Moreover, the maximum stress at this point was 19.0061E3 kN/m2. According to results the following has been concluded:

The position of the hole has a direct influence on the maximum stress in plate.

The more and smaller the element in terms of size, the more accurate results.

The elements in regions of high stresses should be smaller than those in regions of low stress.

The maximum stress in the plate is at the curve side of the structure and around the hole.

Quadratic mesh provides better results than linear and was used due to the curves sides of the structure.

Applying the appropriate number of divisions on the plate maintains better and sensible mesh.

Dividing the mesh improves the results and gives mesh better appearance.

The minimum maximum stress was at position 35 with a value of 19.0061E3 kN/m2.

The results could have been improved by using the full version of LUSAS software to apply more division on the structure. This could not be achieved using the student version which allows up to a total of 500 divisions. The usage of the full version permits more divisions to be added which aids in producing a finer mesh with more elements and smaller in size.

conflict of interest statement

The corresponding author states that there is no conflict of interest.

,

References

[1] Bathe, K., J., The AMORE paradigm for fnite element analysis, Advances in Engineering Software 130 (2019) 1–13.

[2] Cook, R. D. (2002) Finite element modeling for stress analysis, New York, Wiley.

[3] Dawe, D. J. (1984) Matrix and finite element displacement analysis of structures, New York, Oxford University Press.

[4] Hamouda, A. M. S., Sulaiman, S., Lau, C. K., Finite element analysis on the effect of workpiece geometry on the quenching of ST50 steel, Journal of Materials Processing Technology 119 (2001) 354-360.

ISSN 2433-202x
JOURNAL DOI 10.26739/2433-202x
www.journalofresearch.asia
info@journalofresearch.asia

[5] Jusheng, Y., Nan, Y., A brief review of FEM software technique, Advances in Engineering Software, Volume 17, Issue 3, 1993, Pages 195-200.

[6] Mackerle, J. Review Article FEM and BEM in the context of information retrieval, Computers and Structures 80 (2002) 1595–1604.

[7] Mackie, I. (2010) Mesh design. Lecture 2 Notes Advanced Structural Engineering. University of Dundee.

[8] Rao, S., S., Chapter 21 - Finite Element Analysis Using ABAQUS, The Finite Element Method in Engineering (Sixth Edition), 2018, Pages 671-701, Oxford, Butterworth Heinemann.

[9] Rao, S., S., Chapter 22 - Finite Element Analysis Using ANSYS, The Finite Element Method in Engineering (Sixth Edition), 2018, Pages 703-721, Oxford, Butterworth Heinemann.

[10] Rao, S., S., Chapter 23 - MATLAB Programs for Finite Element Analysis, The Finite Element Method in Engineering (Sixth Edition), 2018, Pages 723-743, Oxford, Butterworth Heinemann.

[11] Wan, D., Hu, D., Yang, G., Long, T., A fully smoothed finite element method for analysis of axisymmetric problems, Engineering Analysis with Boundary Elements 72 (2016) 78–88.

[12] Zhou, J., Wang, K., Li, P., A hybrid fundamental-solution-based 8-node element for axisymmetric elasticity problems, Engineering Analysis with Boundary Elements 101 (2019) 297–309.

[13] Zou, Y., Taylor, W. E. G., Heath, D. J., Technical Note A Numerical Model for Borehole Breakouts, International Journal of Rock Mech. Min. Sci. & Geomech. Abstr. Vol. 33, No. 1, pp. 103-109, 1996.