

Automation in Ansys APDL Simulating Square Plate with a Circular Hole

Afreen, Master of Engineering, Sankethika Vidhya Parishad Engineering College, Vishakapatnam & India, afreen9901@gmail.com

Ch. Bhanu Sree, Assistant Professor, Sankethika Vidhya Parishad Engineering College, Vishakapatnam & India, bhanusri2712@gmail.com

Abstract: Now-a-days, AUTOMATION is improving at faster rates in our nation. Advantages commonly attributed to automation include higher production rates and increased productivity, more efficient use of materials, better product quality, improved safety, shorter work weeks for labor, and reduced factory lead times. Hence, the present work is introducing automation (programming in ANSYS APDL) in finding out stress concentration factors for rectangular or square plate with a circular hole at ANY LOCATION, also tested the same specimen with two materials – Structural Steel and Aluminum 6061-T6. The plate mentioned is subjected to uniform tensile load. The stress concentration factor has been calculated by using both analytical and finite element methods. By using this automation tool, a researcher can directly get SCF (stress concentration factor) for different materials in a very less time and can reduced R&D lead times.

Keywords — Ansys, APDL, Automation, Stress concentration factor, Finite Element Analysis

I. INTRODUCTION

Structural integrity is very crucial in all mechanical industries like Aerospace, Automobiles, Oil & Gas, and Power Sectors. As per codes and standards all the designing and execution works will carry out, for safety and quality. But the common structure that comes across in all types of industries is a plate with hole appears as a single part or in assemblies. Evaluating stress concentrations at the location where material has been removed like holes or notches is a standard practice followed before selection of material of construction (MOC).

Hence in this project a Square plate with a circular hole in it is considered with different materials, which is subjected to normal tensile load (bi-axially). Manually if done this project in ANSYS – Pre-processing, Solve and Post-processing will take atleast 30 to 50 minutes from an expert level to beginner level. Instead of that, this project has contributed its time in develop a program/code to complete this task in milliseconds, with Graphic User Interface and User Customization for geometry changes has also been added in this MACRO, which saves not only time, but also money in research point of view.

II. ANALYSIS OF SPWCH (SQUARE PLATE WITH A CIRCULAR HOLE) – MANUALLY

Following are the detailed step-by-step procedure for modeling a square plate with circular hole at center.

Since this part is symmetric in both 'X' and 'Y' axes, we can model quarter of the part and give symmetric boundary

conditions, so that it will save time and also reduces number of finite element equations to solve.

Step-1: Defining preferences as “Structural”

Step-2: Choose “8 Node 183” for Element Type

Step-3: After selecting the element type, click on “Options” button and select Element Behavior (K3) as “Plane str w/thk”, which says that the thickness can be added for element directly.

Step-4: Now go with the Real Constants and define Thickness (THK) = 2

Step-5: As shown go to Material Model and enter the mechanical properties of the material being considered for analysis.

Step-6: Geometry Modeling starts with defining Key points. The Square plate which is considered has dimensions as, $L \times B \times T = 100 \times 100 \times 2$ (mm). As shown in the figures below, define key points accordingly.

Step-7: Now creating lines and arc for the geometry. As shown in below figure, select “By End KPs & Radius” option. In the first go, select key points – ‘2’ and ‘6’, then click on “Apply” button, again it prompts to pick the center of the curvature, then select key point ‘1’, now a window will appear and prompts for “RAD” value, enter 5 and “OK”.

Step-8: Now an arc has created between key points ‘2’ and ‘6’, with 5mm radius.

Step-9: Create lines using “In Active Coord” option and select each point.

Step-10: “Mesh Tool” has used to mesh lines for manually setting the Number of Element Divisions (NDIV) and Spacing Ratio (SPACE) between the elements (a.k.a Biasing)

For Line 'L1' NDIV = 30; For Line 'L2' and 'L5' NDIV = 15 also SPACE = 8

For 'L3' and 'L4' NDIV = 15

Step-11: Create area using the lines (L1 to L6) including the arc.

Step-12: Concatenate lines 'L3' and 'L4', so that mapped meshing can be possible

Step-13: Again use Mesh Tool to mesh the area created. Make sure that "Mapped" option was selected and choose the area.

Step-14: Applying boundary conditions. First, define SYMMETRY B.C. as shown.

Step-15: Plot only lines to see the symmetry symbol "s" on the lines considered.

Step-16: The tensile force which has to be applied on the plate must be distributed on all nodes existing on line 'L3'. For this to happen, choose "Coupling/Ceqn" and select all nodes using BOX option.

Step-17: After selection of all nodes a window appears and prompts to set reference number (NSET) – any number can be entered; choose "UX" for "Degree-of-freedom label" as the tensile force/load is going to be in X-direction only.

Step-18: Applying force. Select any node on the line 'L3', the force will distribute with equal share on each node.

Direction of force/mom = "FX"

VALUE = 500

As this is a symmetry design, the total force applied is 1000N

III. SOLUTION

Step-1: Go to plot results and select "Deformed Shape" = 'Def + undef shape'

Maximum Displacement obtained in X-direction, DMX = 0.127e-08 mm

Step-2: Plot Results > Contour Plot > Nodal Solu > Stress > Von Mises Stress

Maximum Stress obtained in X-direction, SMX = 14.74 MPa

Step-3: To see full portion of the entire plate.

Go to, PlotCtrls > Style > Symmetry Expansion > Periodic/Cyclic Symmetry > select type of cyclic symmetry as "1/4 Dihedral Sym"

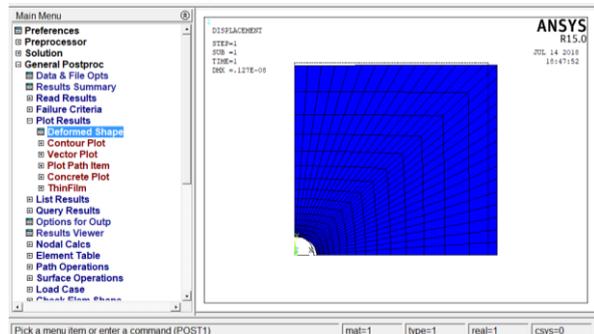


Fig:3.1 Deformed Shape of SPWCH – Quarter part

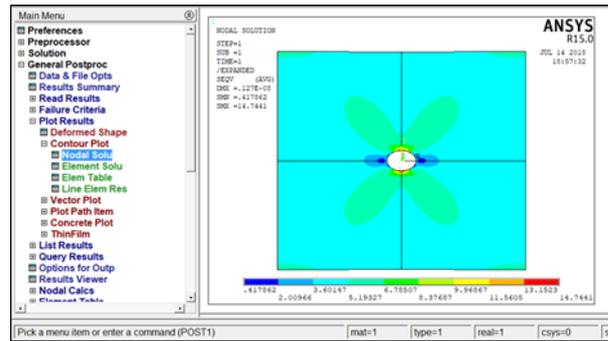


Fig: 3.2 Von-mises Stress for the entire plate

From ANSYS APDL main window access File > Read Input From... > Choose the TXT file which was saved in the previous procedure. A window will appear, prompting the required input to do the analysis.

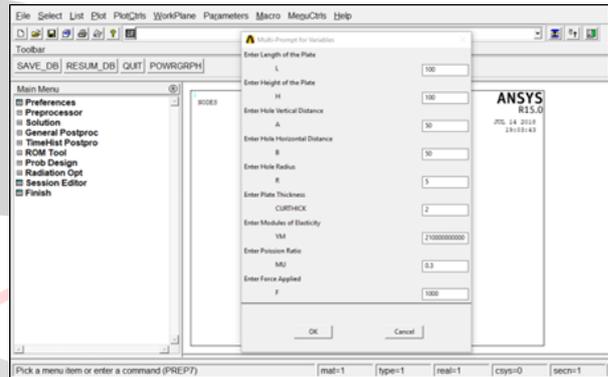


Fig:3.3 Graphic-User-Interface of the Developed Macro Inputs

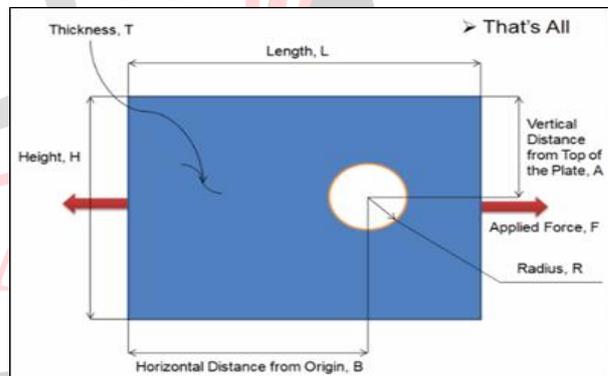


Fig:3.4 Actual meaning of the variables used in Macro Units

Inputs in the macro are

Length of the plate, L = 100 mm

Height of the plate, H = 100 mm

Vertical distance from top of the plate to hole center, A = 50 mm

Horizontal distance from edge of the plate to hole center, B = 50 mm

Hole radius, R = 5 mm

Plate thickness, CURTHICK = 2 mm

Modulus of Elasticity, YM = 210 GPa (210e9)

Poisson's Ratio, MU = 0.3

Force Applied, F = 1000 N

Enter the above values and click "OK"

"Solution is done!" - Exactly within 54 seconds.

Values entered at 19:03:43 and Solution done at 19:04:47,

that too with fine Mapped Meshing.

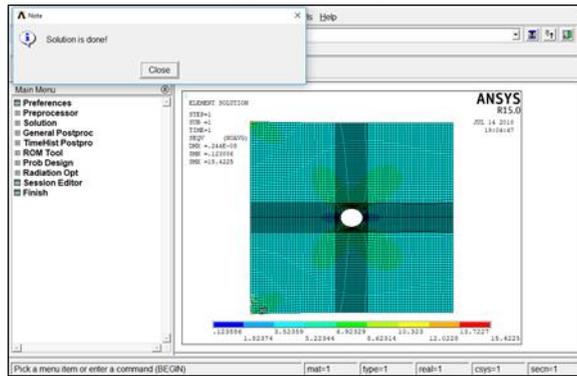


Fig:3.5 Mapped Meshing done by Macro

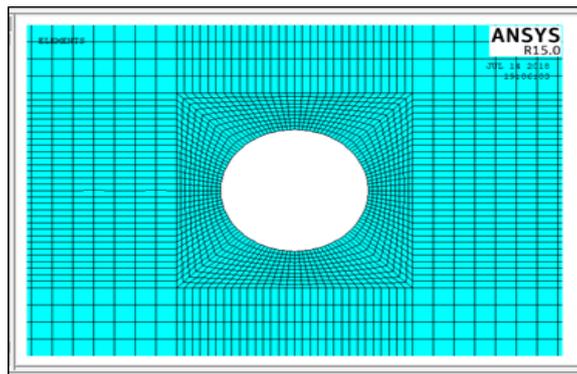


Fig:3.6 Accuracy of Mapping

This paper considering two ductile materials for comparison:

- Structural Steel (Young's Modulus, $E = 210\text{GPa}$, $\mu = 0.3$)
- Aluminum 6061-T6 (Young's Modulus, $E = 68.9\text{GPa}$, $\mu = 0.33$)

Also with four "Hole Diameter to Width" ratios has selected to compare the program.

Problem description:

- a. Square Plate dimensions (Length X Width X Thick) = 100 X 100 X 2 (mm)
- b. Tensile Force applied, $F = 1000\text{ N}$
- c. Four Hole Diameter (d) / Plate Width (D) ratios are 0.10, 0.15, 0.20 and 0.25

Following are the results obtained: (Figures below are listed in detailed form top to bottom)

- Hole Diameter = 10 mm & 15 mm (Structural Steel) (Done Manually)
- Hole Diameter = 20 mm & 25 mm (Structural Steel) (Done Manually)
- Hole Diameter = 10 mm & 15 mm (Al 6061-T6) (Done Manually)
- Hole Diameter = 20 mm & 25 mm (Al 6061-T6) (Done Manually)
- Hole Diameter = 10 mm & 15 mm (Structural Steel) (Done Using MACRO)
- Hole Diameter = 20 mm & 25 mm (Structural Steel) (Done Using MACRO)
- Hole Diameter = 10 mm & 15 mm (Al 6061-T6) (Done Using MACRO)
- Hole Diameter = 20 mm & 25 mm (Al 6061-T6) (Done Using MACRO)

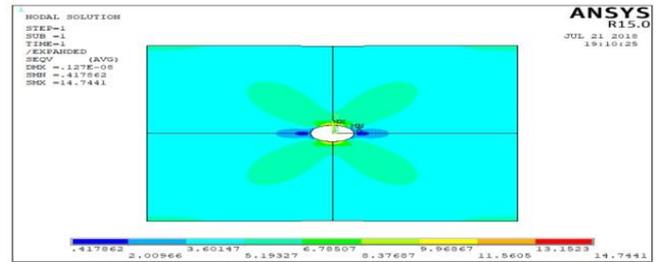


Fig:3.6(a)

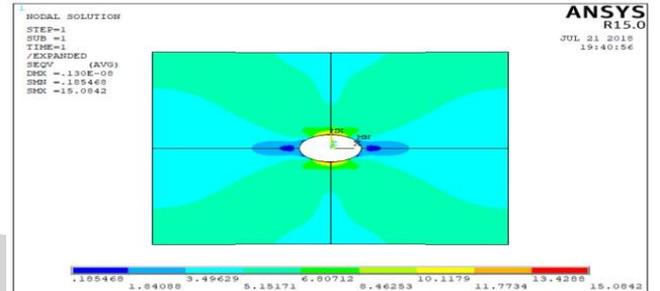


Fig:3.6(b)

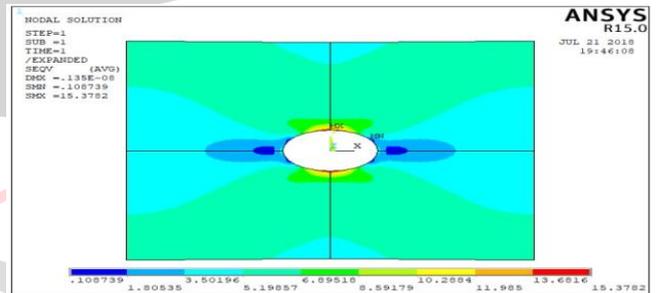


Fig:3.6(c)

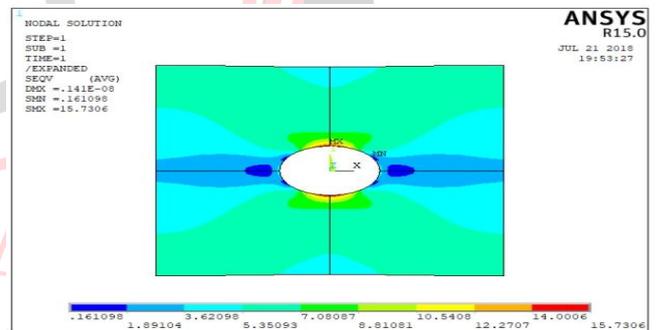


Fig:3.6(d)

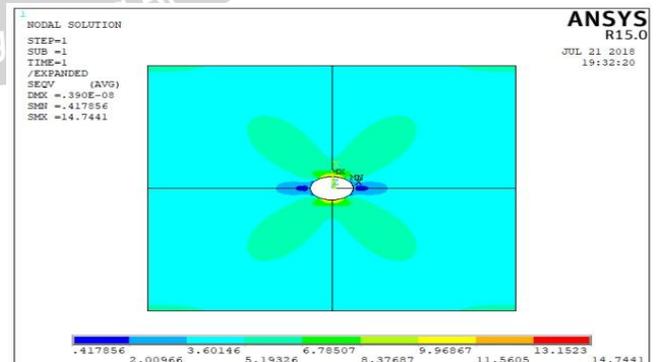


Fig:3.6(e)

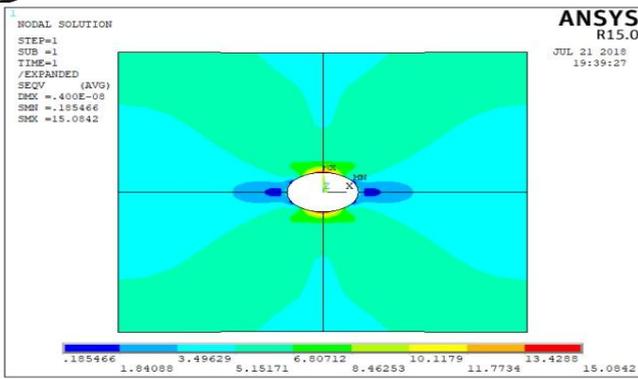


Fig:3.6(f)

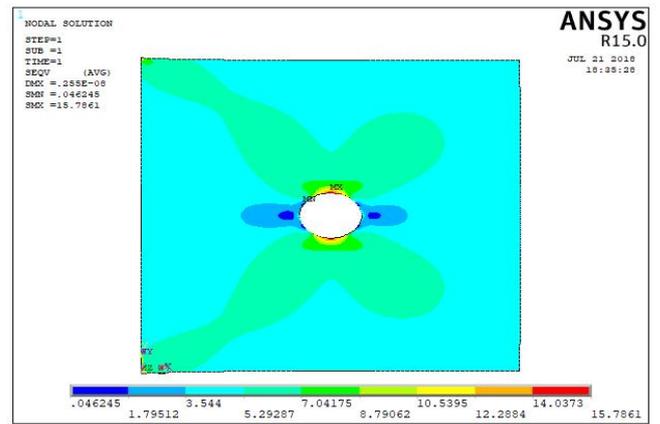


Fig:3.6(j)

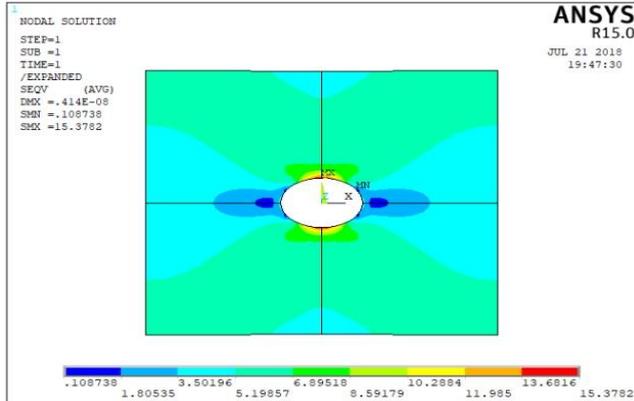


Fig:3.6(g)

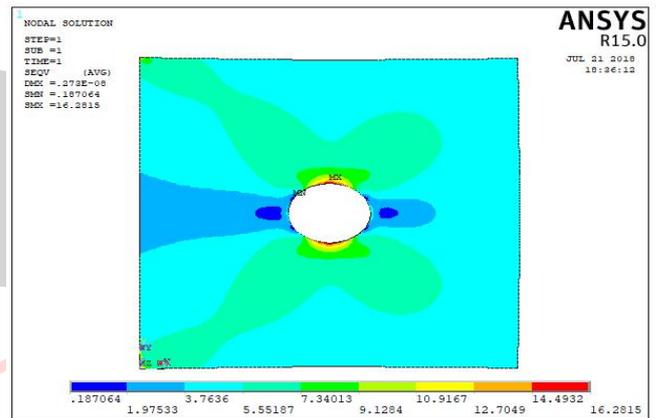


Fig:3.6(k)

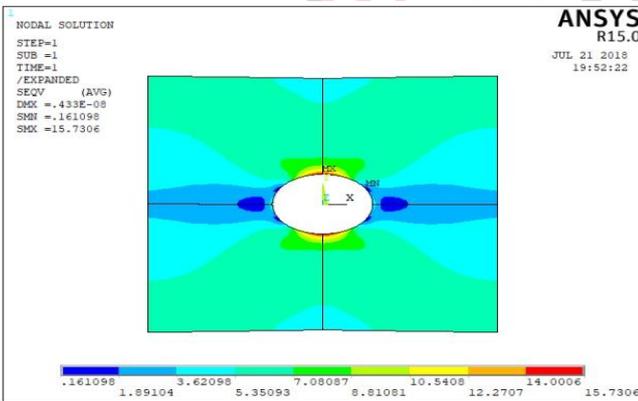


Fig:3.6(h)

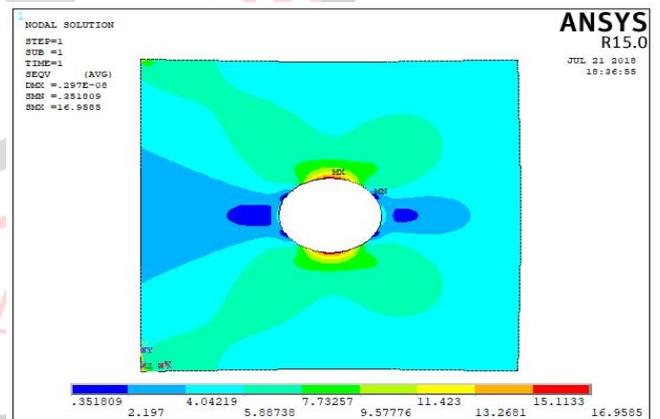


Fig:3.6(l)

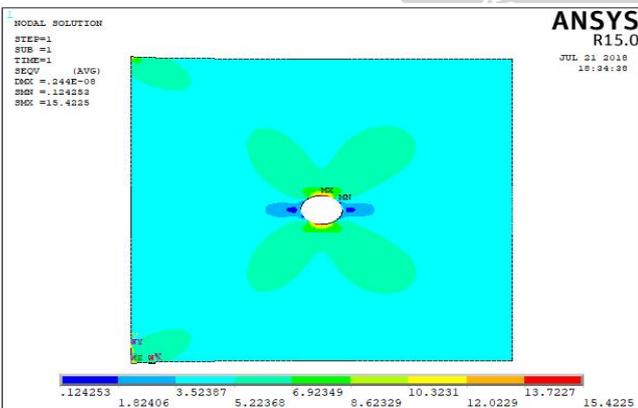


Fig:3.6(i)

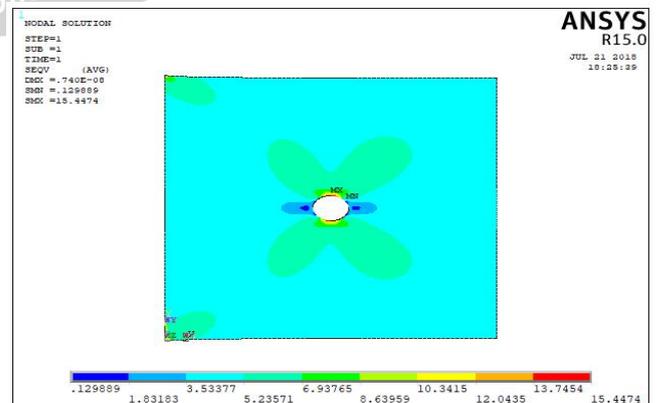


Fig:3.6(m)

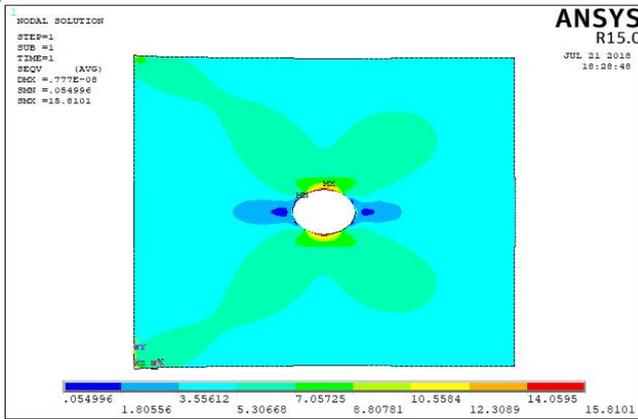


Fig:3.6(n)

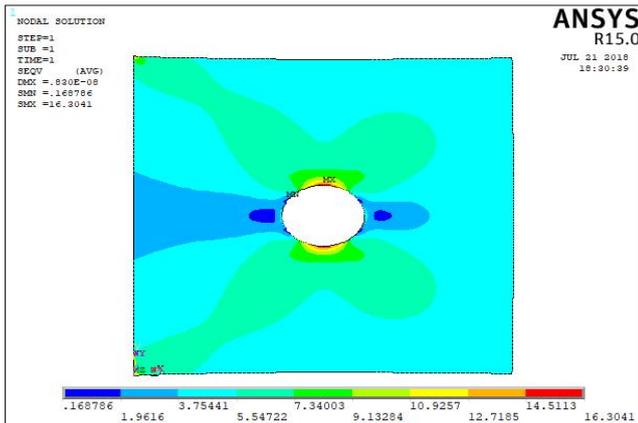


Fig:3.6(o)

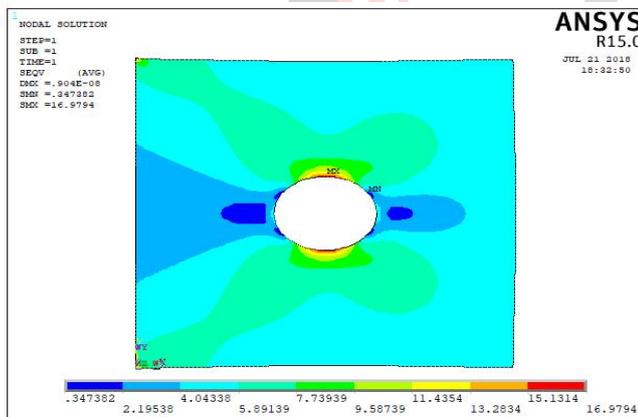


Fig:3.6(p)

Stress Concentration Factor from Stresses,

$$K_t = \sigma_{max} / \sigma_{nom}$$

Stress Concentration Factor from Diameter to Width Ratios,

$$K_t = 3 - 3.13 \left(\frac{d}{D}\right) + 3.66 \left(\frac{d}{D}\right)^2 - 1.53 \left(\frac{d}{D}\right)^3$$

$$K_t = 3 - 3.13 \left(\frac{10}{100}\right) + 3.66 \left(\frac{10}{100}\right)^2 - 1.53 \left(\frac{10}{100}\right)^3 = 2.72$$

$$K_t = 3 - 3.13 \left(\frac{15}{100}\right) + 3.66 \left(\frac{15}{100}\right)^2 - 1.53 \left(\frac{15}{100}\right)^3 = 2.61$$

$$K_t = 3 - 3.13 \left(\frac{20}{100}\right) + 3.66 \left(\frac{20}{100}\right)^2 - 1.53 \left(\frac{20}{100}\right)^3 = 2.51$$

$$K_t = 3 - 3.13 \left(\frac{25}{100}\right) + 3.66 \left(\frac{25}{100}\right)^2 - 1.53 \left(\frac{25}{100}\right)^3 = 2.42$$

Percentage Error,

$$\% \text{ Error} = \frac{\text{Theoretical Value} - \text{Experimental Value}}{\text{Theoretical Value}} \times 100$$

d/D RATIO	Kt (Formula)
0.10	2.72
0.15	2.61
0.20	2.51
0.25	2.42

Table:4.1 Respective Values of d/D Ratio and K_t

STRUCTURAL STEEL						ALUMINUM 6061-T6					
FEA			MACRO			FEA			MACRO		
MAX STRESS	NOM. STRESS	Kt	MAX STRESS	NOM. STRESS	Kt	MAX STRESS	NOM. STRESS	Kt	MAX STRESS	NOM. STRESS	Kt
14.74	5.56	2.65	15.42	5.56	2.78	14.74	5.56	2.65	15.44	5.56	2.78
15.08	5.88	2.56	15.78	5.88	2.68	15.08	5.88	2.56	15.81	5.88	2.69
15.37	6.25	2.46	16.28	6.25	2.60	15.37	6.25	2.46	16.30	6.25	2.61
15.73	6.67	2.36	16.95	6.67	2.54	15.73	6.67	2.36	16.97	6.67	2.55

Table:4.2 Nominal, Max. - Stresses & K_t Obtained in ANSYS using MANUAL & MACRO usage

IV. RESULTS COMPARISON, VALIDATION AND PERCENTAGE ERROR CHECKING

Before that all the theoretical formulae which has been used to calculate and compare are shown below:

Stress concentration factor, K_t for a circular hole = 3, generally

Nominal Stress,

$$\sigma_{nom} = \frac{\text{Force Applied}}{\text{Area}} = \frac{F}{(\text{Width} - \text{Diameter of Hole}) * \text{Thickness}}$$

Nominal Stress,

$$\sigma_{max} = \text{Values obtained from Numerical Analysis (ANSYS)}$$

STRESS CONCENTRATION FACTOR, K _t - % ERROR CHECK											
STRUCTURAL STEEL						ALUMINUM 6061-T6					
ERROR B/T THEORETICAL & FEA			ERROR B/T THEORETICAL & MACRO			ERROR B/T THEORETICAL & FEA			ERROR B/T THEORETICAL & MACRO		
THEORETICAL	FEA	%	THEORETICAL	MACRO	%	THEORETICAL	FEA	%	THEORETICAL	MACRO	%
2.72	2.65	2.50	2.72	2.78	2.00	2.72	2.65	2.50	2.72	2.78	2.13
2.61	2.56	1.64	2.61	2.68	2.93	2.61	2.56	1.64	2.61	2.69	3.12
2.51	2.46	1.89	2.51	2.60	3.92	2.51	2.46	1.89	2.51	2.61	4.05
2.42	2.36	2.51	2.42	2.54	5.05	2.42	2.36	2.51	2.42	2.55	5.17

Table:4.3 Stress Concentration Factor, K_t % Error Check

THEORETICAL AND MACRO - MAX STRESS - % ERROR CHECK					
STRUCTURAL STEEL			ALUMINUM 6061-T6		
THEORETICAL	MACRO	%	THEORETICAL	MACRO	%
14.74	15.42	4.61	14.74	15.44	4.75
15.08	15.78	4.64	15.08	15.81	4.84
15.37	16.28	5.92	15.37	16.30	6.05
15.73	16.95	7.76	15.73	16.97	7.88

Table:4.4 Max. Stress % Error Check

Since this macro is developed with a motivation that the circular hole can be anywhere inside the plate, not only at the center. If an engineer has to do analysis for an unsymmetrical location of hole in a square plate, he/she has to model and analyze in full portion and there is no chance to consider quarter portion. Thus at those situations this MACRO can help them.

V. CONCLUSION

From the present investigation, it can be concluded that the maximum stress occurs at corners of holes in both materials. Also, for both materials, while an increase in T/D ratio resulted into a decrease in Von-Mises stress and as T/D ratio increases with increase in stress concentration factor. The stress concentration factor values obtained from finite element analysis shows a maximum error of 2.51% for Manual analysis and 5.05 for Macro analysis – Structural Steel and also shows a maximum error of 2.51% for Manual analysis and 5.17 for Macro analysis – Aluminum 6061-T6 which are acceptable. Thus, from the present study, it can be said that finite element analysis is a very effective tool to determine stresses induced in various materials.

Even though the experiment of tensile test on a plate with a hole in it is very simple one, but it has enormous technical challenges which have not been relieved yet. If for different materials this same experiment has to run, then the entire pre-processing process has to be changed and re-run the analysis. If in another case the hole which is at center has to be in another location, except center, then the entire modeling work and then the pre & post works has to be carried out newly. With this macro, the mentioned challenges can be solved very easily and at rocket speeds.

VI. FUTURE SCOPE

This macro developed for square plate alone. In future it can be upgraded to work for rectangular plates also with holes with different geometry like circular, box type, rhombus shape etc. Also same procedure can be followed to develop macros that can work for plates with notches. Forecasting also is in progress for composite materials.

ACKNOWLEDGMENT

The preferred spelling of the word “acknowledgment” in American English is without an “e” after the “g.” Use the singular heading even if you have many acknowledgments.

REFERENCES

- [1] Ulku Babuscu Yesil, “The effect of the initial stretching of the rectangular plate with a cylindrical hole on the stress and displacement distributions around the hole”, Yıldız Technical University, Faculty of Chemical and Metallurgical Engineering, Department of Mathematical Engineering, Davutpa, sa Campus, 34210, Esenler, Istanbul-TURKEY Received 19.03.2010.
- [2] Warren C. Young And Richard G. Budynas, “Roark’s Formulas for Stress and Strain”, McGraw-Hill, New York Chicago San Francisco Lisbon London, Madrid Mexico City Milan New Delhi San Juan Seoul, Singapore Sydney Toronto.
- [3] Konish, H.J. and Whitney, J.M., “Approximate Stresses in an Orthotropic Plate Containing a Circular Hole”, Journal of Composite Materials, Vol. 9, pp. 157-166, 1975.
- [4] Jinho Woo, and Won-Bae Na, “Effect of Cutout Orientation on Stress Concentration of Perforated plates with Various Cutout and Bluntness”, International Journal of Ocean system engineering Vol. 1, pp. 95-101, 2011.
- [5] Pilkey, W. D. and R. E. Peterson, “Peterson’s Stress Concentration Factors”, 2nd. New York, Wiley 1997.
- [6] S. Dhanjal and R. Arora, “Stress Analysis of a Rectangular Plate with Circular Hole Using Three Dimensional Finite Element Model” Int. J. Engg., Business and Enterprise Applications (IJEBA), 1(12), 77-80 (2015).
- [7] B. Mallikarjun, P. Dinesh and K. I. Parashivamurthy, “Finite Element Analysis of Elastic Stresses Around Holes in Plate Subjected to Uniform Tensile Loading” Bonfring Int. J. Indust. Engg. Manage. Sci., 2(4), 136-142 (2012).
- [8] Y. Nakasone and S. Yoshimoto, T. A. Stolarski, "Engineering Analysis with ANSYS Software" Elsevier Publications.
- [9] Troyani, N. Homes C, Sterlacci G., "Theoretical stress concentration factors for short rectangular plates with centred circular holes", Journal of Mechanical Design, pp.126-128,2002.
- [10] Stanley, P.Day, B.V. "Stress concentrations at an oblique hole in a thick flat plate under an arbitrary in-plane biaxial loading", Journal of strain analysis for engineering design, Vol.28, pp. 223-235, 1993.