Design and Analysis of Plate with Holes using Finite Element Analysis

Sunilakumar Biradar

BLDEA's V.P Dr.PGH College of Engg & Tech, Vijayapur

Abstract— It is very Often that analysis of stresses in complex structures is Very tedious job for mechanical engineers. So various methods have been introduced like analytical method, by using software's, And by FEM approach. Among all the methods software based Analysis is easy because it encourages any types of problems and the results obtained are error free. MSC NASTRAN PATRAN is one among that software that has helped from aeronautical to design engineers in their work. Due to its well-structured program and user friendly interface Analysis of structure is very easy. Our vision is to understand the software by solving some complex Structures using this software and validating it with the tradition Methods like analytical method or by fem approach.

Key words: Plate, converge, Fos, MSC NASTRAN/PATRAN

I. INTRODUCTION

In day today life we come across of various many structures like loads, bridges, towers, plate with hole etc. but as an engineers can we think are these structures safe, if safe what is FOS? Or what is stress distribution, stress concentration etc.

These are the basic things that we need to know as a mechanical engineer. If those structures are not safe what is the modification on substitution for those. Our Project deals with such an effort or a small step to understand such concepts. We choose some day today examples of plate with hole in different structures that are in existence to our study. Before moving deep further let's know what the basic to understand our report are.

A. Concept of Plate with Hole:

It is seen in day to day engineering that the plates with holes or Semicircular holes are seen. So they are necessary for some design and unavoidable. But, whenever testing of such material fails at the hole due to stress Concentration, so as a result the analysis becomes important and remedy is also important for such cases.

Different Structures or Plate with hole:

1) Circular Plate with circular hole used in dumbels

Fig. 1.2(a):

2) Hexagonal plate with circular holes used in Rivetts and joint

Fig. 1.2(b):

3) Circular plate with circular hole used in disc brakes

Fig. 1.2(c):

4) Rectangular plate with circular hole used in Rivets and joints

Fig. 1.2(d):

Advantages:

1) The plate with hole has many applications so study and analysis is necessary before it is used directly. So in order to attain the safe design the analysis is very advantageous
2) It saves the time as well as cost
3) Easily recyclable
4) High durability

Applications:

1) Plate with hole used in joints
2) Used in the different machine components
3) Used in window panes
4) Used in the vehicle components like disc brakes , airoplane frames , bus roofs ,etc

B. Introduction to MD Nastran- Patran Software & FEM Approach:

Nastran: Nastran is FEA program that was developed for NASA in 1960's in USA. It was initially used for stress analysis in space vehicles. It name taken by NASA Structure Analysis. It was released for public in late 1971 that helped to study the elastic structure.

Patran: Patran is the world's most widely used pre/post processing software for FEA that provides solid modeling, meshing, analysis setup & post-processing for multiple solvers.

C. Fem Steps:

The solution of a general continuum problem by the finite element method always follows an orderly step by step process. The step-by-step procedure with the reference the static structural problem can be stated as follows:

1) Discretisation of the continuum: The First step in the finite element method is to divide the given continuum into smaller regions of finite dimensions called as "Finite elements". The original Continuum or Structure is then considered as an assemblage of these elements connected at the finite number of the joints called as " Nodes" or " Nodal Points ".
2) Selection of approximating functions: Approximating functions are also known as the displacement function or interpolation model. Displacement function is the starting point of the mathematical analysis. The displacement function may be approximated in the form a linear function or a higher-order function. A convenient way to express it is by polynomial expression.
3) Formation of Element stiffness matrix: After continuum is discrete with desired element shapes, the individual element stiffness matrix is formulated. Basically it is a minimization procedure whatever may be approach adopted.
4) Formation of Overall Stiffness matrix: After the element stiffness matrices in global coordinates are formed, they are assembled to form overall stiffness matrix. The overall stiffness is symmetric and banded.
5) Formation of element loading matrix: The loading forms an essential parameter in any structural engineering problem. The Loading inside an element is transferred at the nodal points and consistent element matrix is formed.
6) Formation of Overall Loading Matrix: The element loading matrices are assembled to form the overall loading matrix. This matrix has one column per loading case.
7) Formation of Overall Equilibrium equation: Overall equilibrium equation is the systematic arrangement of overall stiffness matrix, overall load vector and overall displacement vector to get set of simultaneous equations. Overall equilibrium equation can be expressed as shown Below
8) Incorporation of boundary conditions: The boundary restraint conditions are to be imposed in the stiffness matrix to avoid the condition of singularity. The solution cannot be obtained unless support conditions are included in the stiffness matrix.
   [K]{Q} = {F}
   Where, [K] is a overall or global stiffness matrix (square matrix)
   {Q} is a overall or global displacement vector (Column matrix)
   {F} is a overall or global force vector (Column matrix)
9) Calculation of unknown nodal displacements: After incorporation of boundary conditions elimination method or penalty methods of handling boundary condition are used to calculate unknown nodal displacements from equilibrium equation or simultaneous equations.
10) Calculation of strain and stresses: Nodal displacements are utilized for calculation of strain and stresses using suitable equations.

II. ANALYSIS OF PLATE WITH HOLE

Determine the maximum Stress and Displacement in rectangular plate with hole is loaded axially.

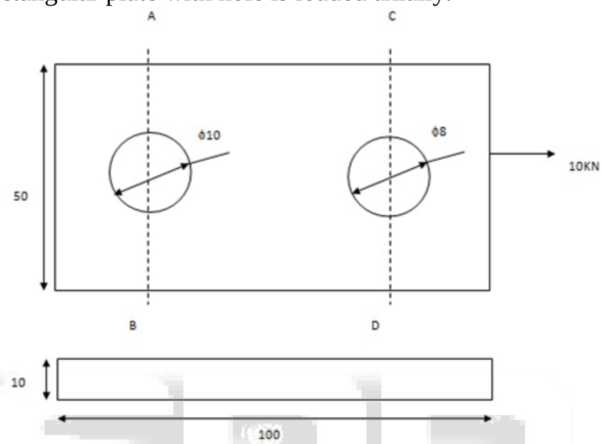

Fig. 2.1:

The above figure shows the rectangular plate of 50*100 having two holes of diameters 10 mm and 8 mm and axial load of 10KN is applied.

III. SOLUTION BY SOM APPROACH

Consider the section A-B

Normal stress= Force/Area = F/ (w-a) t
= (10*1000)/(50-10)10
=25 N/mm2

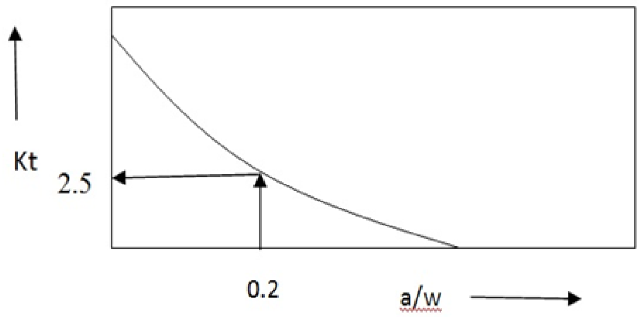

Fig. 3.1:

From the graph at a/w = 10/50=0.2

Kt=2.5

Therefore the Maximum stress

□max = Kt * Normal stress
= 2.5*25
= 62.5 N/mm2

Now considering the section C-D

Normal stress = Force/Area = F/ (w-a) t
= (10*1000) / (50-8)10

= 23.81 N/mm2

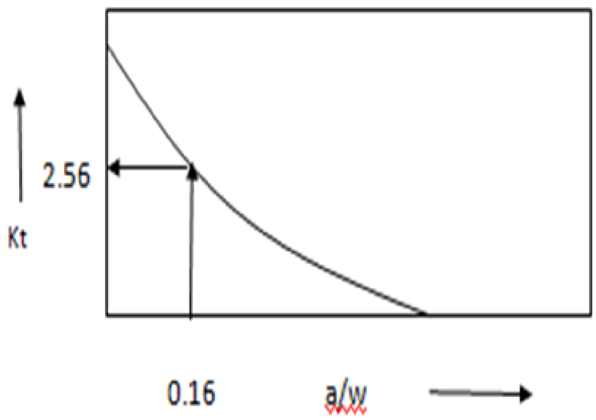

Fig. 3.2:

From the graph at a/w=10/50=0.16

Kt=2.56

Therefore, The Maximum stress

σ_{max} = Kt * Normal stress

= 2.56*23.81

= 60.59N/mm2

IV. SOLUTION USING NASTRAN & PATRAN SOFTWARE

Model: It is the model taken for analysis .It has the rectangular plate with two circular holes

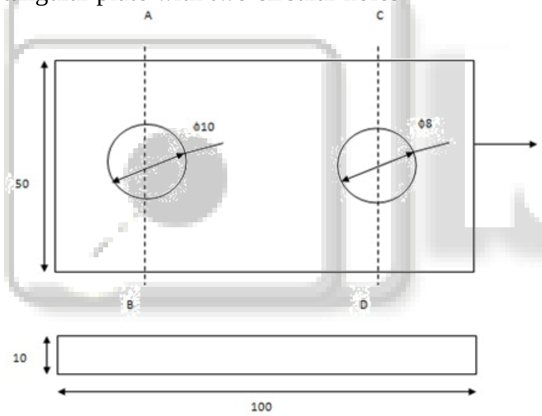

Fig. 4.1: Model

Material Data : Here we consider size as 2 and discritization is done as shown in figure

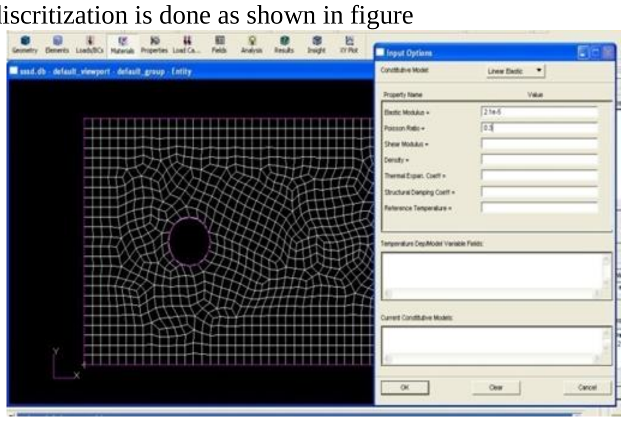

Fig. 4.2: Material Data

Properties: Here we given the properties like material of element as MS and the area for the all elements as 1 mm2

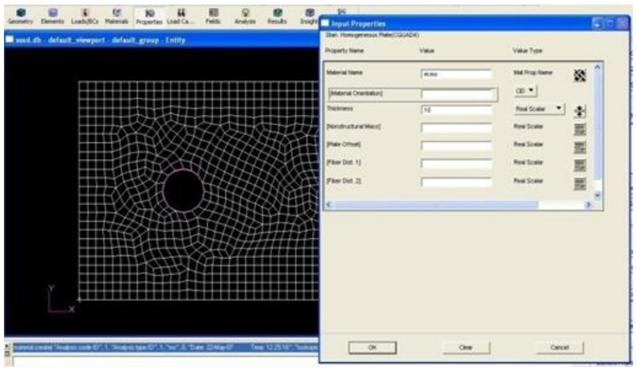

Fig. 4.3: Properties

Loading conditions: In this step we given the boundry conditions. The node 1 is fixed so here we restrict all degrees of freedom. and point load of 384.6 N is applied at node 2

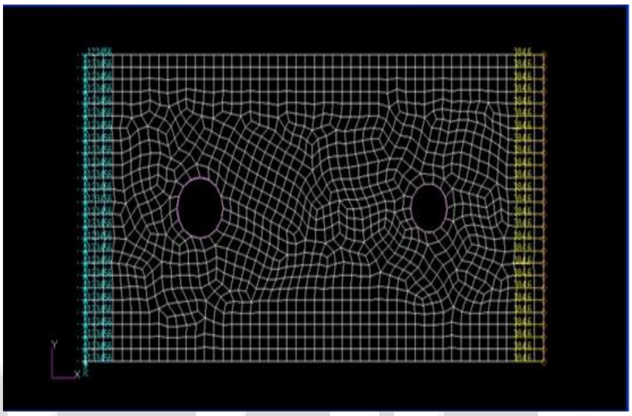

Fig. 4.3: Loading Conditions

Displacement Result: Finally we get theMaximum result as 1.07 e-3 mm and Minimum result as 0 mm

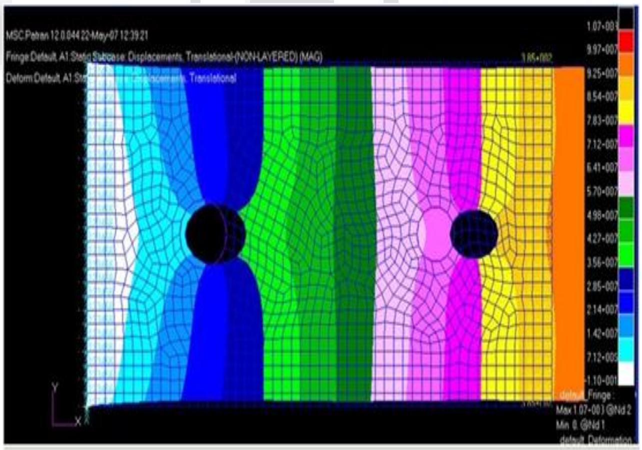

Fig. 4.4: Displacement Result

Stress Results: We get the Maximum stress as 54.8 N/mm2 and the Minimum stress as 6.14 N/mm2

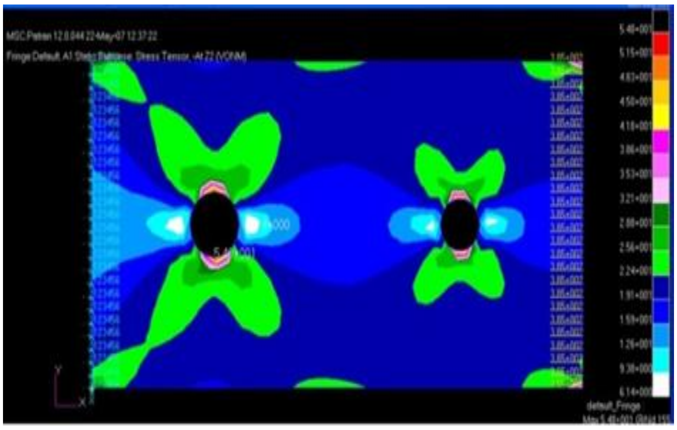
Fig. 4.5: Stress Result

A. A Software Results:

	Maximum	Minimum
Stress Results	54.8 Mpa	6.14 Mpa
Displacement Results	$1.07 * 10^{-3}$mm	0mm

Table 4: A Software Stress Strain Results

V. Results and Discussions

Method	MOM Approach	Nastran and Patran Software
Stress in N/mm^2	At section A-B 62.5	Maximum 54.8 Minimum 6.14
Strain	$2.976*10^{-4}$	$2.6*10^{-4}$

Table 5: A Stress Strain Results

VI. Conclusion

1) It is seen that the results obtained by FEA simulation is near to analytical solutions.
2) We can converge the simulation of FEA simulation by increasing that no of the elements.
3) Analytical method is observed to be tedious time consuming so the FEA software's are useful to save time.
4) By changing the material we can improve the stability of the structure.

References

[1] Wikipedia website www.wikipedia.com
[2] Engineers garage website www.engineersgarage.com
[3] Design of Machine Elements text Book by V.B.Bhandhari
[4] Finite Element Methods text Book by S B Halesh
[5] Finite Element Methods text Book by H K Manhohar
[6] M C Graw's Access science Reference book
[7] ASTM Standards & Engg Library
[8] Scholar website www.scholar.com