e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
Volume:02/Issue:05/May-2020 www.irjmets.com

# STRESS ANALYSIS OF COMPOSITE PLATE WITH ELLIPTICAL AND RECTANGULAR CUTTOUTS BY FINITE ELEMENT METHOD

**Laxman V Bharambe[*1], Prof.Hemant R Nehete[*2]**

[*1]M.Tech Scholar, Dept. of Mechanical Engineering, J.T. Mahajan College of Engg, Faizpur, India

[*2]Assistant Professor, Dept. of Mechanical Engineering, KCES'S COEIT, Jalgaon, India.

## ABSTRACT

To study the stress analysis of plates with different central cut-outs first an experimental stress analysis is carried out. For designing of structures, understanding of the effects of cut-out on the load bearing capacity and stress concentration of such plates or shells is very important. Particular emphasis is placed on flat in finite plates subjected to a uni-axial tension load. The results based on analytical solution are compared with the results obtained using finite element methods. The objective of this study is to demonstrate the accuracy and simplicity of presented analytical solution for stress analysis of plates with special shaped central cut-out. The parameters like cut-out shape, load direction or cut-out orientations, which affect the stress distributions and stress concentration factor in the metallic plates, are taken into account. The results conclude that the stress concentration factor of metallic plate significantly changed by using special shape cut-out, bluntness and orientation. ANSYS software is used for stress analysis by finite element method.

**KEYWORDS:** Stress analysis, cut-out, metallic plates, load direction.

## I. INTRODUCTION

Prevalent mechanical properties of composite materials, for example, high firmness and solidarity to weight proportions made it be utilized progressively in numerous zones of innovation including marine, aviation, car and others. As of late, scientists have invested colossal amounts of energy so as to examine the pressure dispersion around circular and elliptical openings. Various specialists have endeavored to figure the SCF for various types of geometric discontinuity under various types of loads by various methods.

Circular, elliptical, and rectangular cutouts are two-dimensional gaps as a rule. For circular hole issues in limited field, there are some exploration results utilizing analytical methods.

The main purpose of the paper is that finding the stress and deformations in elliptical and rectangular plate with and without cutout is by using finite element analysis.

## II. FINITE ELEMENT METHOD

Finite Element Analysis (FEA) as applied to structures is a multidisciplinary technique, based on knowledge from three fields:

(1) Structural Mechanics, encompassing elasticity, strength of materials, dynamics, plasticity, etc.

(2) Numerical Analysis, involving approximation methods, solving linear sets of equations.

(3) Applied Computer Science, dealing with the development and maintenance of large computer codes.

FEA is used to solve large-scale analytical problems. Its task is to model and describe the mechanical behavior of geometrically complex structures. The procedure is a discretized approach: the geometric shape or the internal stress strain-displacement field are described by a series of discrete quantities (like coordinates) distributed through the structure. This requires a matrix notation. The tools are the computers, able to store long lists of numbers and manipulate them.

## III. OBJECTIVES OF FEA

The objective of FEA is to replace the infinite degree of freedom system in continuum applications by a finite system exhibiting the same basis as discrete analysis. The aim is finding an approximate solution to a boundary and initial value problem by dividing the domain of the system into a set of interconnected finite-sized sub domains of different size and shape, and defining the unknown state variable approximately, within each element, by means of a linear combination of trial functions. The sub domains are called finite elements, the set of finite elements is known as the mesh and the trial functions are referred to as interpolation functions. With the individually defined functions matching each other at certain points called nodes, the unknown function is approximated over the entire domain. The primary difference between the FEA and other approximate methods for the solution of boundary-value problems (finite-difference, weighted-residual Rayleigh-Ritz, Galerkin) is that in the FEA the approximation is confined to relatively small sub domains.

FEA is a localized version of the Rayleigh-Ritz method. Instead of finding an admissible function satisfying the boundary conditions for the entire domain, which is often difficult, in the FEA the admissible functions are defined over element domains with simple geometry and pay no attention to complications at the boundaries. This was possible only at the time the computers became available. The outstanding success of the finite element method can be attributed to a large extent to timing. While the finite element method was being developed, so were increasingly powerful digital computers, which led to automation. The computer is not only able to solve the discretized equations of equilibrium, but also to carry out such diverse tasks as the formulation of equations, by making decisions concerning the finite element mesh and the assembly of stiffness matrices.

Perhaps more important is the fact that the finite element method can accommodate systems with complicated geometries and parameter distributions. The wide use of the classical Rayleigh-Ritz method has been limited by the in ability to generate suitable admissible functions for a large number of practical problems. Indeed, systems with complex boundary conditions or complex geometry cannot be described easily by global admissible functions, which tend to have complicated expressions, difficult to handle on a routine basis. In turn, in the FEA an approximate solution is constructed using local admissible functions, defined over small sub domains of the structure. In order to match a given irregular boundary, or to handle parameter non-uniformities, the FEA can change not only the size of the finite elements but also their shape. This extreme versatility, coupled with the development of powerful computer codes based on the method, some of them made available as open source free software, has made the FEA the method of choice for the analysis of structures.

FEA, the equations of equilibrium are obtained from variational principles implying the stationery of the functional defined by the total potential energy. While solving differential equations with complicated boundary conditions may be difficult, integrating known polynomial functions, even approximately, should be easier.

Mathematically, solving

$[A]\{x\} = \{b\}$ is equivalent to minimizing

$P(x) = 1/2\{x\}^T[A]\{x\} - \{x\}^T\{b\}$

This is the heart of the FEA when applied to structures.

## IV. FINITE ELEMENT ANALYSIS STAGES

The finite element analysis is divided into three different stages:

a. Pre-Processing (build the FE model, loads and constraints)

b. Solution Processing (assemble and solve the system of equation)

c. Post Processing (sort and display the results)

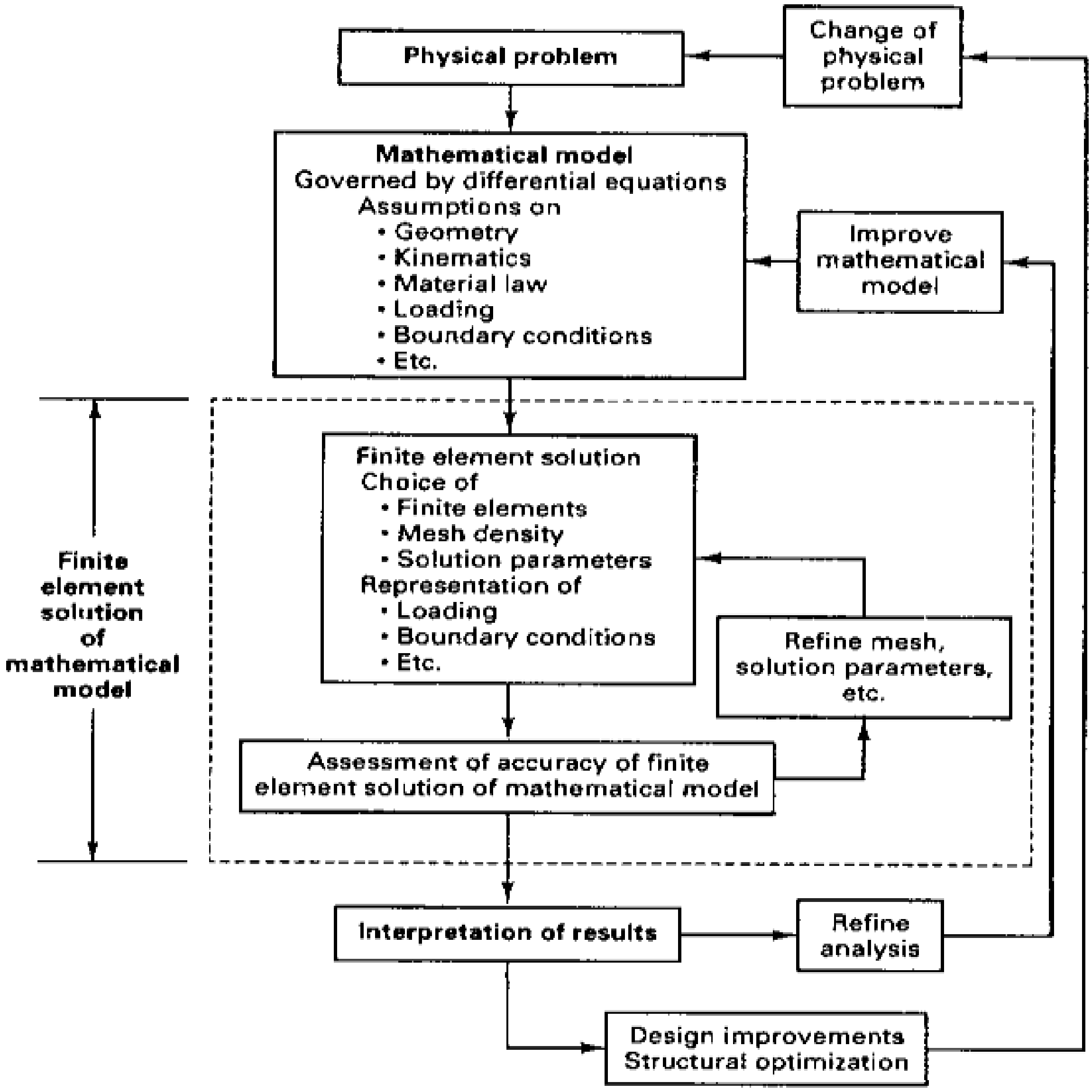



**Fig-1:** Finite element analysis stage

### a) Pre-processing

The user constructs a model of the part to be analyzed in which the geometry is divided into a number of discrete sub regions, or "elements," connected at discrete points called "nodes." Certain of these nodes will have fixed displacements, and others will have prescribed loads. Some of these preprocessors can overlay a mesh on a preexisting CAD file, so that finite element analysis can be done conveniently as part of the computerized drafting-and-design process. In the pre-processing discrete the region known as discretization. The various steps in the Pre- Processor, these are as follows:

In short, steps are covered in Pre- Processing are:

a) Create model and specify the material properties, element type.

b) Discretize the model (meshing).

c) Applying pressure and loads where it required.

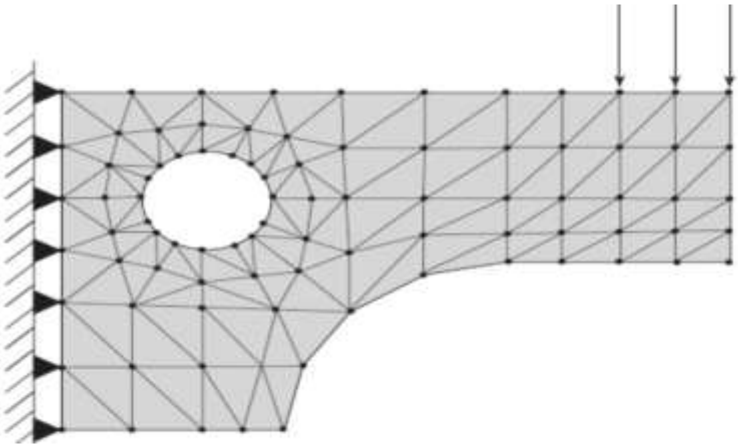


**Fig-2:** Finite element analysis stage-preprocssing

**b) Solution**

Solution processing is done automatically in which the pre-solver read the model created in the preprocessor and formulates the mathematical representation of the model upon the clicks of several button. When the model defined is correct, the solver proceeds to form the element stiffness matrix for the model problem and simply calculates the results. All these results will then be read during the post processor.

In this stage, after applying the pressures and loads on the geometry solving it for the final solution of required results or output. This is important stage which provides output results in case of proper geometry and applying of the loads and pressures. Otherwise it cannot give the results like statement "Solution is Done".

**c) Post-processing**

In the earlier days of finite element analysis, the user would pore through reams of numbers generated by the code, listing displacements and stresses at discrete positions within the model. It is easy to miss important trends and hot spots this way, and modern codes use graphical displays to assist in visualizing the results By using the post processor, the analysis results can be presented in the form of contour plot, deformed shape as well as the table list of displacement.

In this stage, after solving the solution various results can be shown like Deformation, Stress, Strain etc. This result shown in layered colour format which shows the intensities from minimum to maximum.

## V. ANALYSIS PROCEDURE

The geometry of the model was created using CAD program based on the original shape of the model. The model is then imported to FEA program to perform the modal analysis. In the pre-processor stage, the FFE (fast finite element) thinker used topological space methodology to calculate twenty modes additionally to any rigid body modes obtainable within the model.

In the answer process stage, the program runs a linear static analysis to calculate the misshapen form so calculates the frequencies and mode shapes. Throughout the post process stage, the corresponding frequency of the mode form additionally because the displacement of the plate plot on misshapen or under formed shapes. The mesh was generated by ANSA with components.

The operation of a particular code is typically elaborate within the documentation concomitant the computer code, and vendors of the dearer codes can typically supply workshops or coaching sessions additionally to assist users learn the intricacies of code operation.

The tabular format shows the Ansys result of deformation (displacement) shown below:

**Table-1:** Displacement for with and without cut out composite Plate

| Sr.no | Load (KN) | Displacement | |
|---|---|---|---|
| | | Simple plate | Elliptical cutout |
| 1 | 50 | 0.23818 | 0.38152 |
| 2 | 60 | 0.28583 | 0.469 |
| 3 | 70 | 0.33347 | 0.54979 |
| 4 | 80 | 0.38311 | 0.63132 |
| 5 | 90 | 0.42875 | 0.71359 |

The tabular format shows the Ansys result of stress shown below

**Table-2:** Stress for with and without cut out composite Plate

| Sr.no | Load (KN) | Stress | |
|---|---|---|---|
| | | Simple plate | Elliptical cutout |
| 1 | 50 | 75.023 | 191.01 |
| 2 | 60 | 90.028 | 229.21 |
| 3 | 70 | 105.03 | 267.41 |
| 4 | 80 | 120.04 | 305.61 |
| 5 | 90 | 135.04 | 343.82 |

The actual model of plate created in the ANSYS 16.0 Software which shown in Figure No. 3. Then applying loads in the ANSYS 16.0 Software and taking the various results plotted by applying the material properties, boundary conditions.

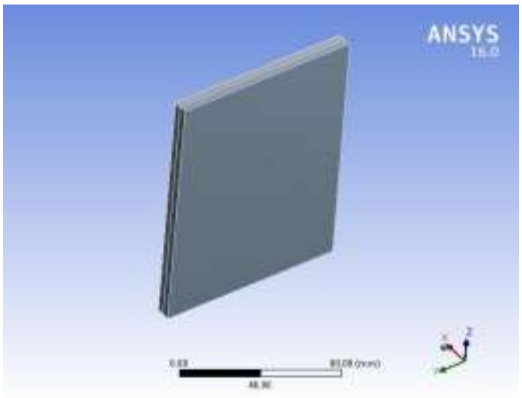



**Fig-3:** Model of composite plate

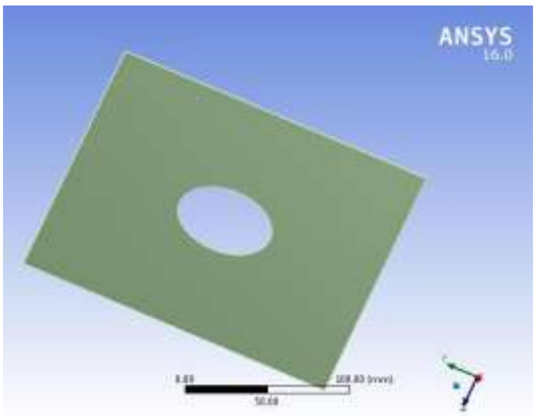



**Fig-4:** Model of elliptical cuout plate

Now we have to take result regarding both cross-sections composite plates. Now according to Pre-processing.

**Meshing of composite plates for elliptical cutout**

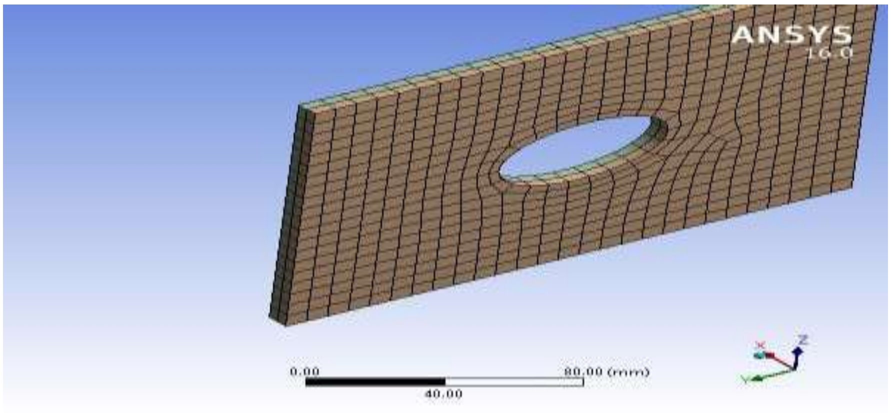



**Fig-5:** Meshing of elliptical cutout plate

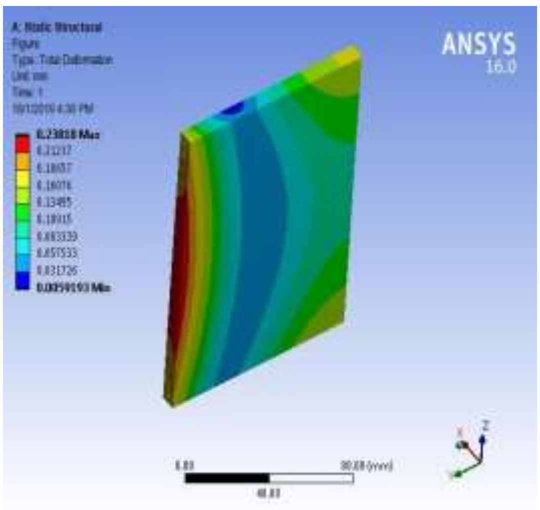
Now considering the loads applying on the with and without cutouts composite plate to take various results like deformation and stress. Forces applying both side of plate for simple composite plates and rectangular and elliptical composite plate with cut outs.

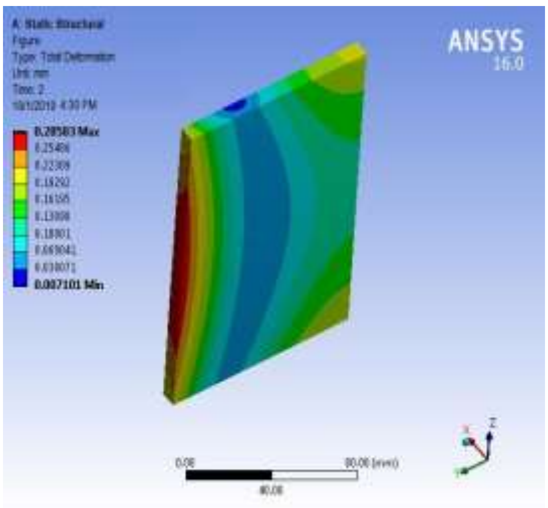
The constant forces are applying on the both plates on both sides and these are pulling the plates i.e. compressive force (50 KN to 90 KN) acting on the plate and finding the displacement in mm occurring in gauge length.

Results are taking after applying the forces on both sides after solving i.e. in solution processor, plots various results at general post processor as follows:

**Deformation of plate**

**Deformation in without cutout composite plates:**



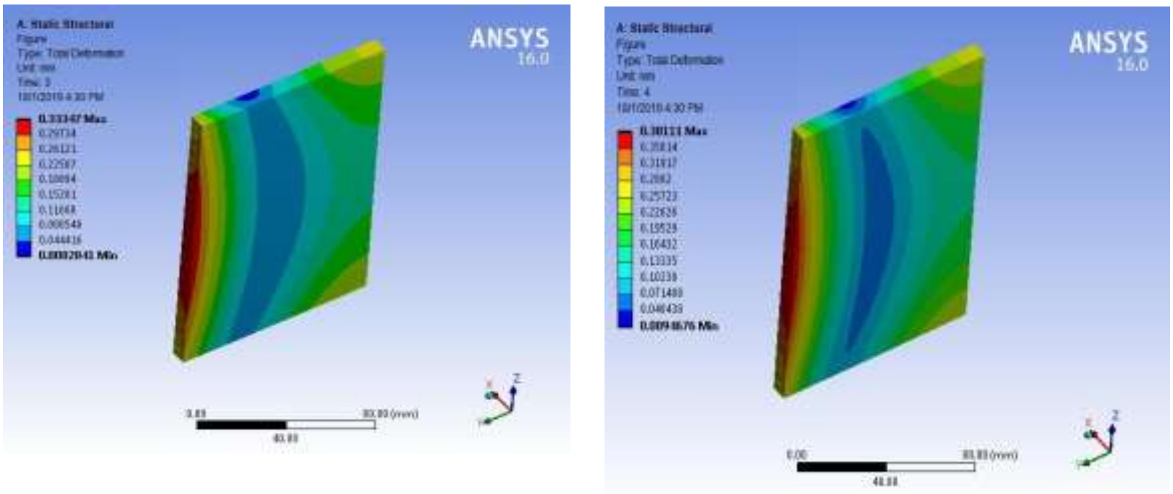



**Fig-6**: Deformation in without cutout composite Plate.

**Deformation in With Elliptical cutout composite Plate:**

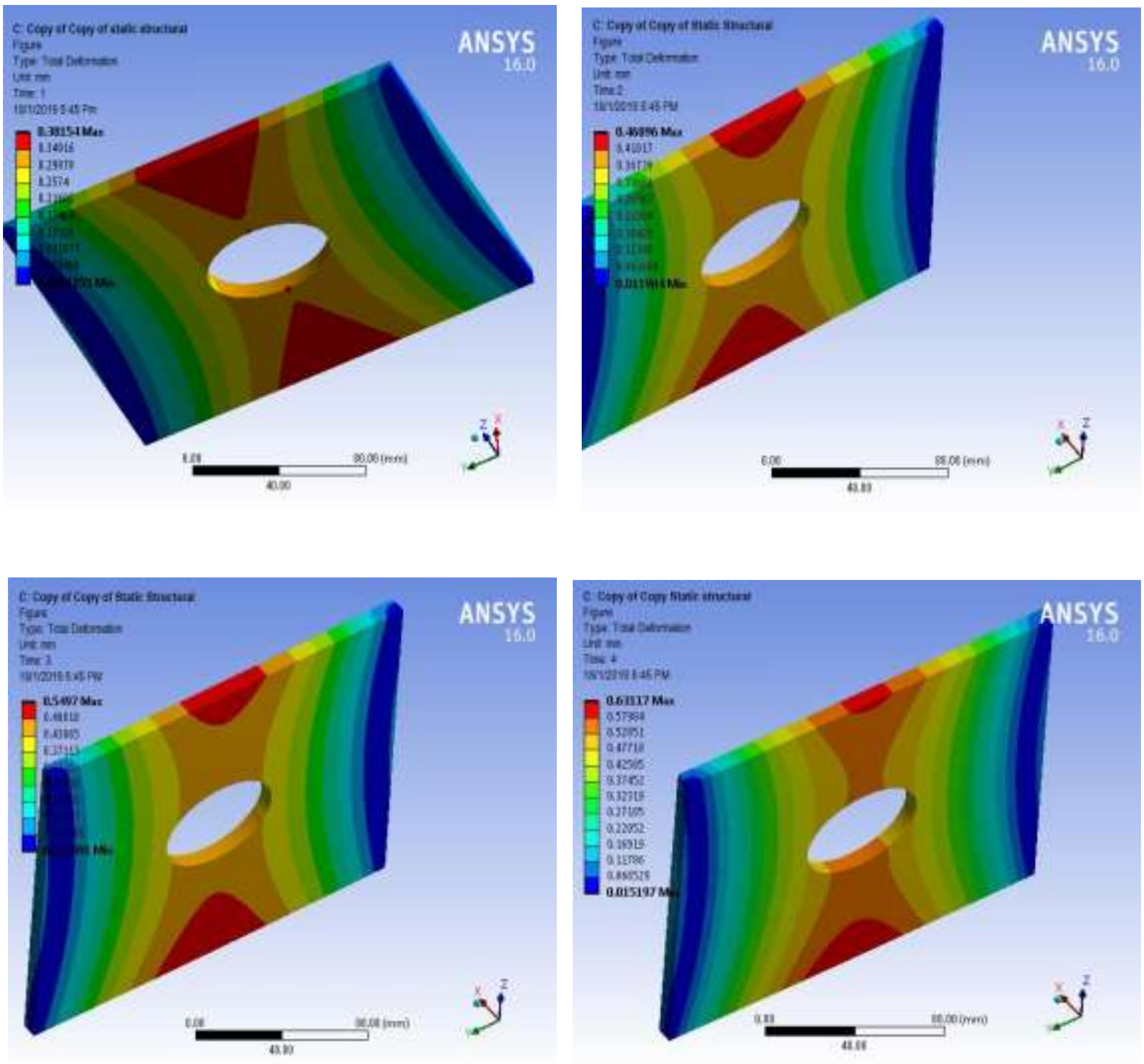



**Fig-7:** Deformation in With Elliptical cutout composite Plate.

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
Volume:02/Issue:05/May-2020 www.irjmets.com

The above figures shows the deformation occurs in the without cut out composite plate and with cut out composite plate after forces applied, the blue region shows the minimum deformation while red region shows maximum deformation in these plates.

**Stress in Composite Plate**

**Stress in without cut out Composite Plate:**

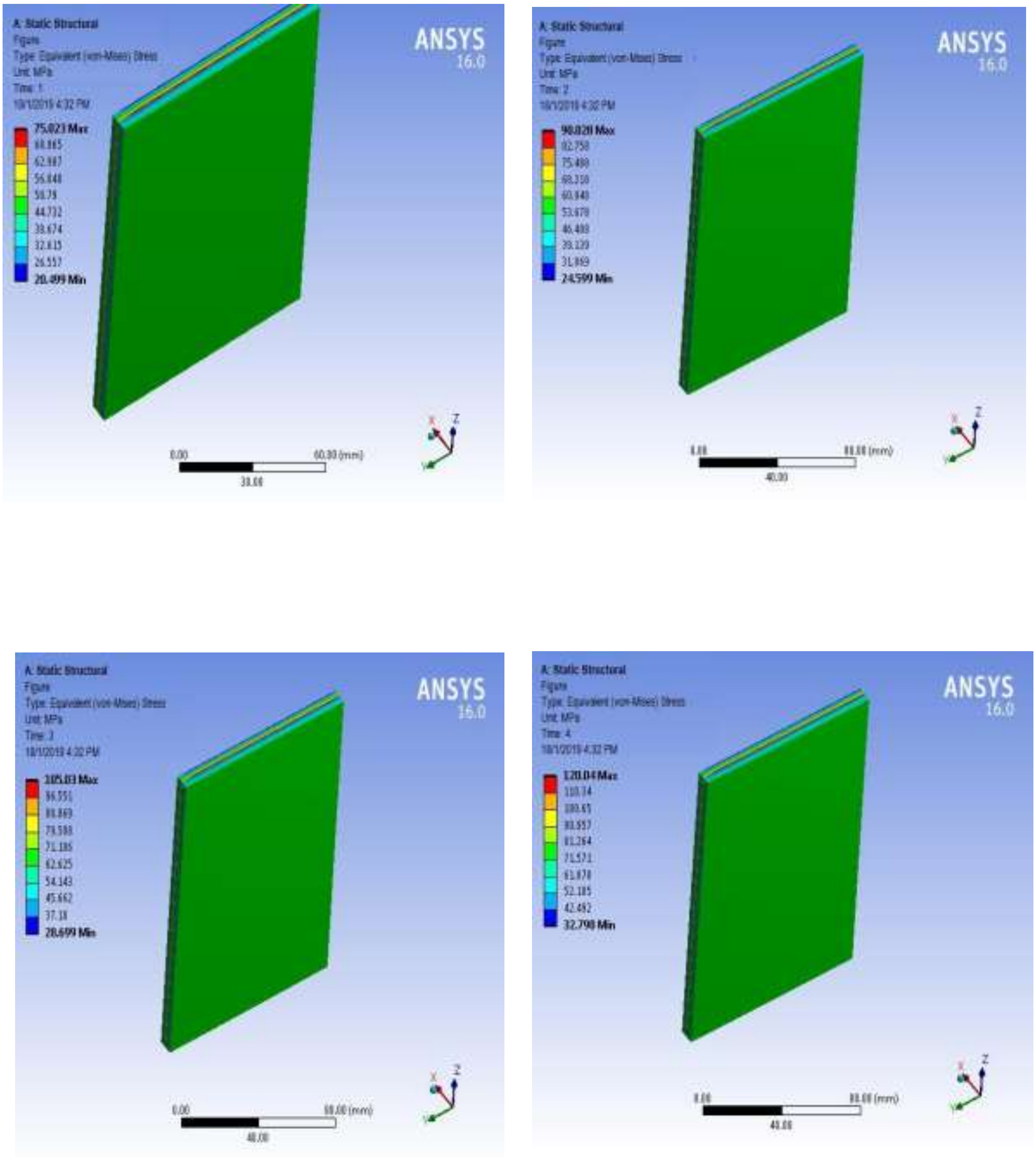



**Fig-8:** Stress in without cut out composite plate.

e-ISSN: 2582-5208
International Research Journal of Modernization in Engineering Technology and Science
Volume:02/Issue:05/May-2020 www.irjmets.com

**Stress in with elliptical cut out Composite Plate:**

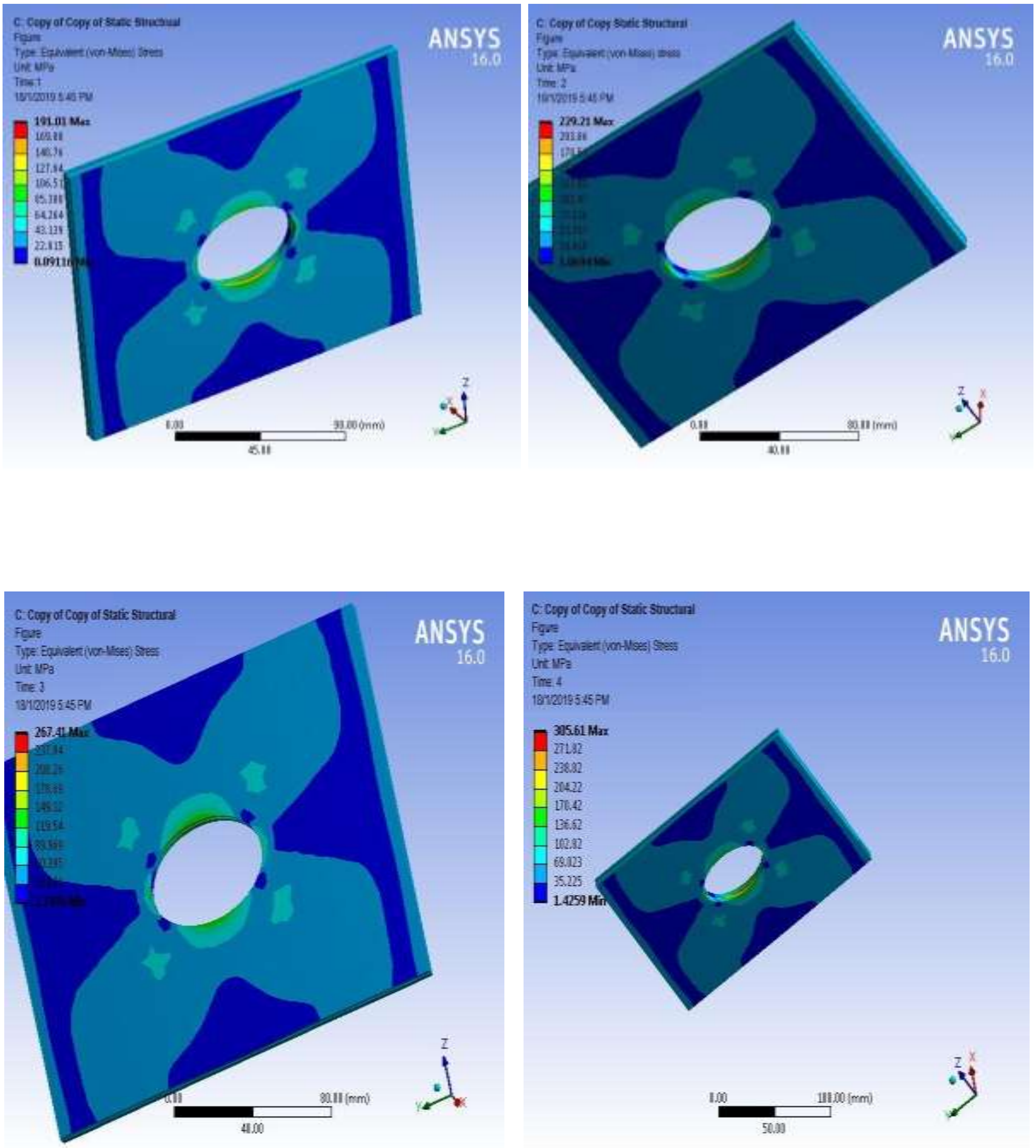



**Fig-9:** Stress in with elliptical cut out composite plate.

The above figures shows the stress occurs in without cut out and with cut out composite plate after forces applied, the blue region shows the minimum stress while red region shows maximum stress in the plates. The stress produced near the cutout is maximum in both cross-sections.

## VI. DISCUSSION

While thickness of plates is increases then increase in resultant stresses as well as displacement is increases. As the shape of the cutout is changes, the stress and displacement is also changes.

## VII. CONCLUSION

By comparing the various observations, with and without cut outs composite plate. The results will be for stress concentration of composite plate Which will definite by reduce the stress concentration Simple composite. By Comparing the Simple Composite plate, and Elliptical composite plate the Stress concentration of simple composite plate is different For other plate, The stress concentration of elliptical cutout plate is minor difference. The stress concentration of Simple plate is reduces to the other plate.

## VIII. REFERENCES

[1] A.R. Abu Talib, A.A. Ramadhan, A.S. Mohd Rafie, R. Zahari "Influence of cut-out hole on multi-layer Kevlar-29/epoxy composite laminated plates" Materials and Design 43 (2013) 89–98.

[2] C.W. Pein and R. Zahari "Experimental Investigation of the Damage Behavior of Woven fabric Glass/Epoxy Laminated Plates With Circular Cut-Outs Subjected to Compressive Force" International Journal of Engineering and Technology, Vol. 4, No. 2, 2007, pp.260-265.

[3] D.B.Kawadkar, Dr.D.V.Bhope, S.D. Khamankar ," Evaluation of Stress Concentration in Plate with Cutout and its Experimental Verification" International Journal of Engineering Research and Applications (IJERA) ,Vol. 2, Issue 5, September- October 2012, pp.566-571.

[4] H. Murat Arslan M. Yasar Kaltakci Huseyin R. Yerli "Effect of Circular Holes on Cross-Ply Laminated Composite Plates" The Arabian Journal for Science and Engineering, Volume 34, Number 2B October 2009.

[5] Jeom Kee Paik. pusan national ..JK Paik, AK Thayamballi, GS Kim. Thin- walled structures 35 (3) JK Paik. Ships and Offshore Structures 2 (1), 69- 80, 2007.

[6] Mohan Kumar, Rajesh S, Yogesh H, Yeshaswini B R," Study on the Effect of Stress Concentration on Cutout Orientation of Plates with Various Cutouts and Bluntness", International Journal of Modern Engineering Research (IJMER) Vol.3, Issue.3, May-June. 2013 pp-1295-1303.

[7] Nilesh Mohite, Vinay Kale, Sujit Kumbhar, Vaibhav Kengar "Stress and Load-Displacement Analysis of Fiber Reinforced Composite Laminates Under Different Loading Conditions" International Journal of Engineering and Technology, Nov-Dec 2012

[8] Sangram S. Jadhav, Prof. Durgesh kumar, "Fem & Experimental Analysis of Composite Laminate with Elliptical Cut Out Using Reflection Polariscope" Int J AdvEngg Tech/IV/III/July-Sept.,2013/67-71.

[9] N. G. R. Iyengar and S. K. Gupta. ... Books for "Composite Material And Structural Analysis Chapters .. Mcmillan Publishers India Ltd., 83-144, 2010.

[10] Robin Olsson, Jonas Iwarsson. "Experiments and analysis of laminates with artificial damage". Composites science and technology. 2003. 63: 199-201.

[11] S.A.M. Ghannadpour, A. Najafi.. "On the buckling behavior of cross-ply laminated composites plates due to circular /elliptical cutouts Composite structures 2006". 75: 2-3.

[12] Md. Tarekul Alam, M. A. Salam Akanda and Amalesh Chandra Mandal. "Finite difference approach for stress analysis in a composite lamina having a circular hole", International Conference on Mechanical Engineering (ICME 2007).

[13] Nageswara Rao D. K., M. Ramesh Babu, K. Raja Narender Reddy and D. Sunil, "Stress around square and rectangular cutouts in symmetric laminates," Composite Structure, vol.92. pp. 2845-2859, 2010.

[14] Patel Dharmin, PanchalKhushbu, Jadav Chetan," A review on Stress Analysis of an infinite plate with Cutouts", International Journal of Scientific and Research Publications, Vol. 2, Issue 11, November 2012.

[15] Rezaee pazhand J. and M. Jafari, "Stress concentration in metallic plates with special shaped cutout," International Journal of Mechanical Sciences, vol.52, pp. 96–102, 2010.