International Journal of Advance Research in Engineering, Science & Technology(IJAREST),
ISSN(O):2393-9877, ISSN(P): 2394-2444,
Volume 2, Issue 5, May-2015

Finite Element Analysis of 2D Object

Maulik C. Vadnere[1], Assistant Professor Mehta Gauravkumar Bharatbhai[2]

[1]Bhagvan Mahavir College of Engineering and Technology, Surat, maulik_vadnere@yahoo.com

[2]Bhagvan Mahavir College of Engineering and Technology, Surat, gauravzzz007@gmail.com

Abstract

Finite Element Analysis of 2D objects (plate with a centrally located hole) using Ansys software with elements, meshing & boundary conditions. This shall be achieved by considering the plate with a centrally located hole with uniform applied stress along its two edges. Then the result is analysed in terms of the stress produced in the vicinity of the hole using different meshing elements in Ansys. The plate is modelled & analysed 1/4th of the plate instead of the whole as the plate is symmetric about horizontal & vertical axis in both geometry & loading.

Keywords— *FE analysis, Ansys, Numerical model, triangular elements, quadrilateral elements, meshing*

I. INTRODUCTION

This research paper will provides the necessary opportunity to explore the different modelling approaches used in ANSYS with elements, meshing & boundary condition. The primary concerned of this paper will be on the issues arising from the plate with a centrally located hole with uniform applied stress along its two edges. The plate is modelled & analysed 1/4th of the plate instead of the whole as the plate is symmetric about horizontal & vertical axis in both geometry & loading as shown in figure-2. The purpose behind considering the 1/4th of the plate for analysis is to take the advantage of the symmetry of the object which would save modelling & solution efforts.[2]

Assumption: The analysis is assumed to be Plane stress & elasto-static.

Table 1. Plate Dimension and Material Property

Dimensions:	Material properties:
W=Width of the plate=150	E= Young modulus=200KN/mm^2
H= Height of the plate=100	ν= Poisson's ratio= 0.33
T= Uniform thickness=20	Y= Yield stress= 200 N/mm^2
R= Hole radius= 25 mm	σ_{xx}= Uniform applied stress= 75 N/mm^2

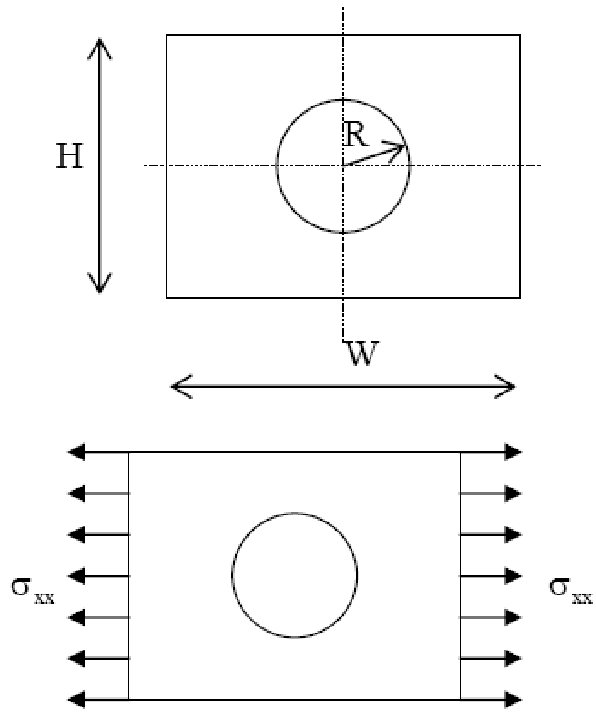


Figure 1. Stress condition on plate

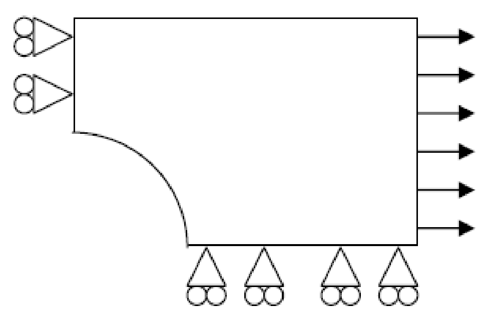


Figure 2. ¼ part of the plate for analysis purpose

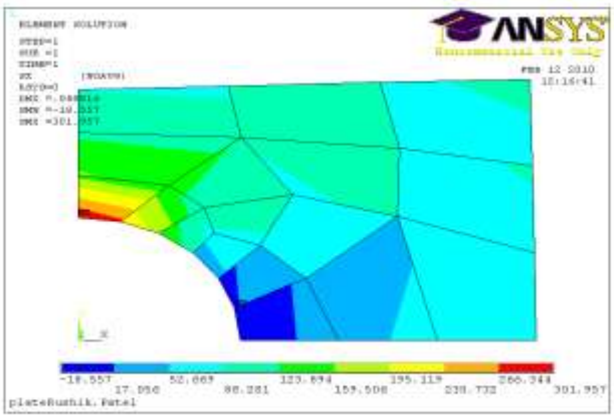
II . FE ANALYSIS

The result of the finite element analysis using Ansys software depends on the types of meshing i.e. elements types (triangular elements or quadrilateral elements).
The 1/4th of the plate is analysed using two different meshing as mentioned below in order to see the impact on the accuracy of the results.[2],[3]

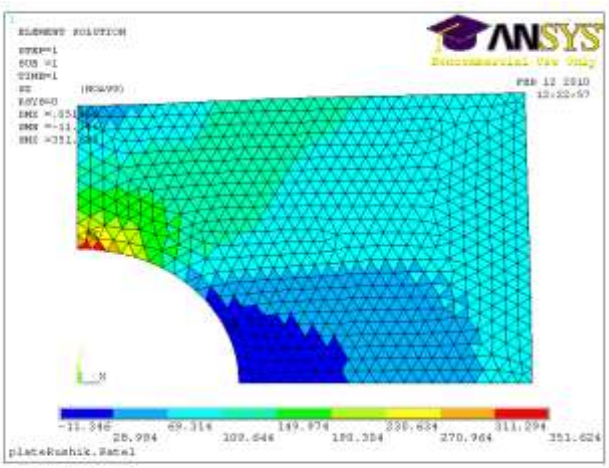
A Coarse meshing (4 nodes 182 element type without refining mesh)

B Fine meshing (4 nodes 182 element type using triangular elements with refining mesh).

The results of the two stress contour plots depicted as below.



Stress contour plot-1 without refining the mesh



Stress contour plot-2 with refining the mesh using triangular elements

A Default Meshing – Course Meshing :

- Element type: Solids (8 node plane 182)
- Element Shape: Triangular
- Element Size: Default
- The stress field was obtained as shown in plot-1 which gives the X component of the stress. Plot shows the discontinuity in the stress fields. Hence, refined meshing is needed instead of default meshing for continuous stress field. This leads to more accurate results.[1],[2]

B Fine Meshing :

- Element type: Solids (8 node plane 182)
- Element Shape: Triangular
- Element Size: 40 (at three edges of the quarter portion as shown in Plot 2)
- Fine meshing was used to obtain more accurate result. The obtained stress field is shown in Plot 2 which gives the X component of the stress. In process of obtaining accurate results, refinement is carried out at the curvature edge and two edges which are connected with arc. These three edges of the quarter portion are shown in plot. The elements size for refinement is 40. [1],[2]
- Please refer plot 3 for full plot of plate model.

III . RESULT

- The maximum stress value for coarse meshing is 301.957 N/mm^2 & for fine meshing is 351.624 N/mm^2 .This can be seen from both stress contours.
- The minimum stress value for coarse meshing is -18.557 N/mm^2 & for fine meshing is -11.346 N/mm^2 .This can be seen from both stress contours.
- The stress distribution in both kind of analysis is different because of the different meshing & this gives the difference in the maximum & minimum stress for both cases.
- The stress contour plot-1 for coarse meshing contains the large size of elements with minimum numbers which resulting less accurate result whereas stress contour plot-2 for fine meshing contains the small size of elements with maximum numbers which resulting accurate result.

IV . CONCLUSION

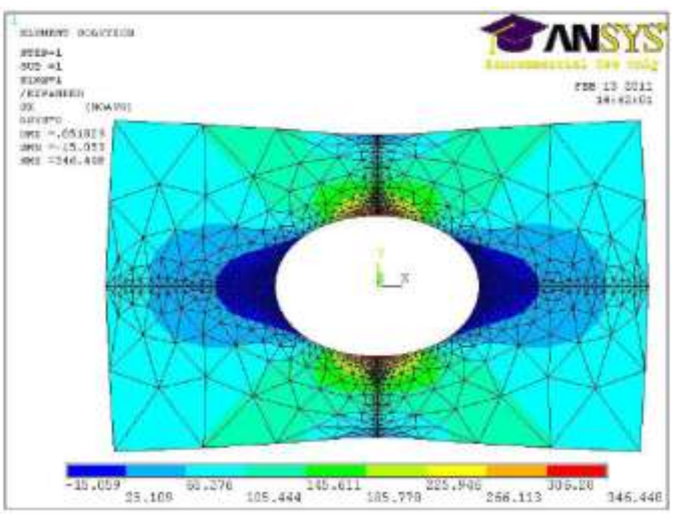
In the view of the above results, following conclusions are made.

- The accuracy of the stress results solely depends on the meshing methodology in finite element method. In this course work the coarse meshing gives less accurate results whereas the fine meshing gives very accurate results. This can be easily seen in both stress contour plots 1 &2. The maximum stress for coarse meshing is less as compare to fine meshing. This proves that as the mesh is refined or fine mesh is used, it gives high stress.[2]

- The analysis will consume more time if fine meshing is used because of more elements & similarly it will take less time if the coarse meshing is used because of less elements.
- The selection of the meshing methodology depends on the nature of the problems & time constraint. If the accuracy of the result is more essential due to criticality of the problems & there is no time constraint, the fine meshing is used for accurate results whereas if the accuracy is not so essential but time is a constraint then the coarse meshing is advisable.[2]
- The selection of the meshing also depends on the shape of the objects which are to be analysed i.e. which element types appropriately fit to the shape of the objects which is responsible for accuracy of the results i.e. triangular elements or quadrilateral elements.[2]

4.1 Difficulties in modelling of the whole plate:

- The difficulty in modelling of the whole plate is only that it is time consuming. For small problems the symmetry may not be too important but for large problems it can save modelling & solution efforts.[1],[2]
- The purpose behind considering the 1/4th of the plate for analysis is to take the advantage of the symmetry of the object which would save modelling & solution efforts.[1],[2]



Stress Contour Plot- 3 full plate model

V. REFERENCE

[1] Mehta Gauravkumar Bharatbhai, "The finite element analysis of a cracked Plate", International journal of advance research in engineering, science & technology issue-4, vol-2, (2015)

[2] Mehta Gauravkumar Bharatbhai, "Lecture-notes- finite element analysis", Department of Mechanical Engineering- Bhagwan Mahavir College of Engineering & Technology, Surat,(2011-2015)

[3] Estekanchi. H.E. .(1999) A parametric finite element study of cracked plates and shells ,fracture mechanics paper,Sharif University of Technology, Iran.