Bending analysis of simply supported thin circular metal plate under point load

Pratik Shinde[1], Manish Jha[1], Mayur Roge[1] , Ravi Singh[1] , Ansari saad[2]
[1]Student, Dept of ME, ARMIET, Shahapur, Maharashtra, India,
[2]assist.Prof.Dept of ME, ARMIET,Shahapur, Maharashtra,india

Abstract: An attempt is made in this study to perform static bending analysis of an isotropic circular plate using four noded shell 181 element which is based on Reiseiner Mendelian's Plate Theory. Modelling and analysis of an isotropic circular plate is done by FEA software ANSYS APDL. In this work effect of various types of meshes on deflections and bending stresses in circular plate is investigated and finally convergence of FEA results with analytical solution is studied. The results are obtained for aspect ratio 10 and 20. Simply supported and clamped boundary conditions with uniformly distributed load are considered for present study.

Then the results obtained are compared with the exact solution which is carried out with the use of the finite element analysis software, Ansys. The results obtained from the present method show good agreement with those from Ansys.

Keywords: Isotropic circular plate; Finite element method; 4 noded shell 63 element; Multi zone mapped meshing; Free meshing

I. INTRODUCTION

1.1 Introduction to Plates

Plates are straight, flat and non-curved surface structures whose thickness is slight compared to their other dimensions. Generally plates are subjected to load conditions that cause deflections transverse to the plate. Geometrically they are bound either by straight or curved lines. Plates have free, simply supported or fixed boundary conditions. The static or dynamic loads carried by plates are predominantly perpendicular to the plate surface. The load carrying action of plates resembles that of beams or cables to a certain extent. Hence plates can be approximated by a grid work of beams or by a network of cables, depending on the flexural rigidity of the structures. Plates are of wide use in engineering industry. Many structures such as ships and containers require complete enclosure of plates without use of additional covering which consequently saves the material and labor. Nowadays, plates are generally used in architectural structures, bridges, hydraulic structures, pavements, containers, airplanes, missiles, ships, instruments and machine parts. Plates are usually subdivided based on their structural action as

1.Stiff Plates, which are thin plates with flexural rigidity and carry the loads two dimensionally. In engineering practice, a plate is understood as a stiff plate unless specified

2.Membranes, which are thin plates without flexural rigidity and carry the lateral loads by axial shear forces. This load carrying action is approximated by a network of stressed cables since their moment resistance is of a negligible order of magnitude.

3.Flexible Plates, which represent a combination of stiff plates and membranes. They carry external loads by the combined action of internal moments and transverse shear forces.

4. Thick Plates, whose internal stress condition resembles that of three dimensional structures.

1.2 Introduction to Plate Analysis

The analysis of plates first started in the 1800s. Euler [1] was responsible for solving free vibrations of a flat plate using a mathematical approach for the first time. Then it was the German physicist Chladni [2] who discovered the various modes of free vibrations. Then later on the theory of elasticity was formulated. Navier [3] can be considered as the originator of the modern theory of elasticity. Navier's numerous scientific activities included the solution of various plate problems. He was also responsible for deriving the exact differential equation for rectangular plates with flexural resistance. For the solution to certain boundary value problems Navier introduced exact methods which transformed differential equations to algebraic equations. Poisson in 1829 [4] extended the use of governing plate equation to lateral vibration of circular plates. Later, the theory of elasticity was extended as there were many researchers working on the plate and the extended plate theory was formulated. Kirchoff [5] is considered as the one who formulated the extended plate theory. In the late 1900s, the theory of finite elements was evolved which is the basis for all the analysis on complex structures. However the analyses using finite elements are now being carried out using comprehensive software which requires high CPU resources to compute the results. Another method for analysis of plates statically and dynamically was later developed for arbitrary shapes using advanced finite elements. Actually there was a method called the weighted residual method which was used in analysis of plate even before the finite element method of analyzing the plate was formulated.

1.3 Symbolic Software

Symbolic software packages such as MATHEMATICA are useful for solving algebraic and symbolic systems. Most of the older symbolic packages which were previously developed were written in LISP but Mathematica is based on the C language to solve problems. Mathematica was first released in 1988. It has had a profound effect on the way computers are used in technical and other fields. Mathematica uses a generic way of writing codes and thereby it is widely used in various fields. A program written in Mathematica is simple, robust and it can be easily understood thereby making it simple for anyone to use it. The framework of Mathematica is such that it is split into two parts, the kernel and the front end. The kernel interprets expressions (Mathematica code) and returns the result. The front end of Mathematica is Graphical User Interface (GUI) which allows us to create edit and format the notebook. More advanced features include 3D picturing, indexing and slide show creation. The advantage of using Mathematica lies in its built-in functions. It has the largest database of algorithms. It is also helpful in numerical computation, symbolic computation, data interpretation etc. Symbolic software also addresses the finite element method and is useful in finding shape functions, creating different types of meshes and can solve problems for different materials.

1.4 Analysis Software

Ansys Inc. has developed many different software packages and amongst those is the ANSYS workbench platform. It is the framework upon which advanced engineering simulations are built. It is the advanced version developed in recent years which has the schematic view and drag drop option thereby making the complex process of the user much simple. Ansys Workbench combines the strength of core problem solvers with project management tools necessary to manage project workflow. In Ansys, Workbench Analysis is built as systems whichcan be combined together into a project. The project is driven by a schematic workflow that manages the connections between the systems.

II. PROBLEM IDENTIFICATION

In world there are lots of appliances and instrumentation in which circular plates are used. Generally circular plates are widely used in engineering appliances such as roof and floor of building,deck slab of bridge, foundation of footing, water tanks and turbine disk.

Plates used in these applications are usually subjected to the lateral load which results in bending of plate. Hence study and analysis of bending of plate is of utmost importance.

So we are in this project are doing the analysis and also considering materials which are generally used for these applications and which will be best for the uses will be identified.

III. METHODOLOGY

1) **Gathering of materials with machining finish.**

2) Doing the anasys analysis using following method. Following is the ANSYS solution obtained for simply supported circular plate of 10mm thickness with UDL:STEP 1:- Geometry Preferences 1. Go to Main Menu -> Preferences 2. Check the box that says Structural 3. Click discipline option h-method 4. Click OK.
Circle 1. Go to Main Menu -> Preprocessor -> Modeling -> Create -> Areas -> Circle -> Solid Circle
2. Put the center of the circle at the origin with radius 0.1 m. Under: WP X enter 0 WP Y enter 0 Radius put .1
3. Click OK

STEP 2:- Preprocessor A] Element Type

1. Go to Main Menu -> Preprocessor -> Element Type -> Add/Edit/Delete

2. Click Add
3. Click Shell -> 4node 181 the element 4. Click OK.
B] Real Constants and Material Properties 1. Go to Main Menu -> Preprocessor -> Real Constants -> Add/Edit/Delete
2. Click Add
3. Click OK
4. Under Real Constants for SHELL181 -> Shell thickness at node I TK(I) enter .01 for the thickness
5. Click OK
6. Click Close. Then specify Young's Modulus and Poisson's Ratio
7. Go to Main Menu -> Material Props -> Material Models
8. Go to Material Model Number 1 -> Structural -> Linear -> Elastic -> Isotropic 9. Enter 200xE9 for Young's Modulus (EX) and 0.3for Poisson's Ratio (PRXY) 10. Click OK.
C] Meshing
1. Go to Main Menu -> Preprocessor -> Meshing -> Mesh Tool
2. Go to Size Controls: -> Global -> Set
3. Under NDIV enter 10. This function will divide all geometry edge lengths into 10 equal segments

4. Press OK
5. Under Mesh Tool -> Mesh select Areas
6. Under Mesh Tool -> Shape: select Quad and Mapped
7. Click Mesh. Following window will appear after meshing is done.

STEP 3:- Loads A] Displacements

1. Go to Main Menu -> Preprocessor -> Loads -> Define Loads -> Apply -> Structural -> Displacement -> On Lines
2. Under List of items enter 1,2,3,4 and click Apply
3. In the new window, select Lab2 -> UX
4. Under value put 0
5. Click Apply
6. Repeat steps 3-6 for the constraints UY,UZ, and ROTZ. We constrain ROTZ to prevent any twisting.
8. Click OK.

B] Uniform Pressure

1. Go to Main Menu -> Preprocessor -> Loads -> Define Loads -> Apply -> Structural -> Pressure -> On Areas
2. Click Pick All
3. Under VALUE enter 275000 4. Under LKEY enter 2. This selects surface 2 5. Click OK.

STEP 4:- Solution 1. Go to Main Menu -> Solution ->Solve -> Current LS (solve). LS stands for Load Step. This step may take some time depending on mesh size and the speed of your computer (generally a minute or less).

STEP 5:- General Postprocessor A] Deflection 1. Go to Main Menu -> General Postprocessor -> Plot Results -> Contour Plot -> Nodal Solution -> DOF Solution ->Z-Component of displacement -> OK.

2. Select the Front View.

B] Von-Mises Equivalent Stress

1. Go to Main Menu -> General Postprocessor -> Plot Results -> Contour Plot -> Nodal Solution -> Stress -> Von Mises stress -> OK.

3)do the mathematical calculations with selected method.

4) use universal testing machine to test the following test:
a) tensile test
b) compression test
c) poel test
4) bend test
5) puncture test

IV. EXPERMENTATION

The whole experiment has yet to be conducted.

V. Result

Result expected to be giving approximately same values for bending in mathematical method and ansys also.

VI CONCLUSION

Due to due experimental delay it can't be known now.

ACKNOWLEDGMENT

The author gratefully acknowledges for the valuable suggestion by Asst. Prof. Rakesh R. Poojari (Mechanical Engineering) and special thanks Dr. Ashwani Kshirsagar (Vice Principal) for their extreme and valuable supports.

REFERENCES

[1]. P. S. Gujar and K. B. Ladhane, “Bending Analysis of Simply Supported and Clamped Circular Plate”, SSRG International Journal of Civil Engineering, vol. 2, issue 5, pp.69-74, 2015.

[2]. B. C. L. Vanam, M. Rajyalakshmi and R. Inala, “Static Analysis of An Isotropic Rectangular Plate using FEA”, Journal of Mechanical Engineering Research,vol. 4(4), pp.148-162, 2012.

[3]. G. C. Mekalke, M. V. Kavadeand S. S. Deshpande, “Analysis of a Plate with a Circular Hole by FEM”, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), ISSN: 2278-1684, pp.25-30, 2012.

[4]. S. Boedo, “Evaluation and comparison of plate finite elements in ANSYS”, NAFEMS World Congress 99, Newport Rhode island, USA,1999.

[5]. M. Vrabie and S. Baetu, “Comparative Analysis of the Bending Theories for Isotropic Plates- Case Study”, BuletinulInstitutuluiPolitehnic Din Iasi, Publicat de UniversitateaTehnica, Gheorghe Asachi din IasiTomul LIX (LXIII), 2013.