International Journal of Technical Innovation in Modern Engineering & Science (IJTIMES)
Impact Factor: 5.22 (SJIF-2017), e-ISSN: 2455-2585
Volume 4, Issue 09, September-2018

A PARAMETRIC STUDY ON LAMINATED PLATES AND SHELLS UNDER STATIC LOADING (USING FINITE ELEMENT METHOD)

P.Veera Sekhar M.Tech[1], P.Raghava M.Tech[2]

[1]*M.Tech Schlor, Civil engineering department, Visvodaya engineering college, kavali,india.*
[2]*Assistant Professor, Civil engineering department, Visvodaya engineering college, kavali,india.*

***Abstract*— A finite element bending analysis of laminated plate and shells are presented in this paper. A degenerated shell element is obtained by degenerating a 3-D solid element. The external faces of the element are curved, while the sections across the thickness are generated by straight lines. The finite element formulation of an 8- noded 3D degenerated shell element with 5 degrees of freedom per node is derived for the analysis. The intent of this investigation, is to validate the accuracy and convergence of the present formulation, by comparing the deflection of thin laminated plates and shells under static loading obtained by, the present finite element program ABAQUS, with manual calculations of finite element method (FEM) results.**

Keywords*— Laminated plates, Dome, single barrel and FEM, shells

I. INTRODUCTION

The finite element method (FEM) is an approximate numerical procedure used for finding problems. The finite element problems were represented by a partial differential equations which will be developed as useful minimisation. The finite element method became popular with the advancements in digital computers since they allow engineers to resolve massive systems of equations quickly and with efficiency. In most structural analysis applications, it's a necessity to calculate displacements and stresses at numerous points of interest. The deflections at each node of the finite element model are obtained by resolution the equilibrium equations. The stresses and strains then is also obtained from the stress-strain and strain-displacement relations. The analytical half was carried out within the finite part program (ABAQUS) then the manual calculations were carried out for the models which are analyzed within the finite element program (ABAQUS) and therefore the obtained results were plotted as load-deflection graphs.

A. Finite element formulation of a degenerated shell element

The formulation of the shell element is predicated on the essential concept of Ahmed et al. (1970), wherever the three-dimensional solid part used to model the shell is degenerated with the help of certain extractions obtained from the thought that one in every of the dimension across the shell thickness is sufficiently little compared to different dimensions. The detail derivation of this element for isotropic case is offered within the literature (Ahmad (1970), Zienkiewicz (1977) and Bathe and Ho).

Let us consider an 8–noded degenerated shell element, obtained by degenerating a 3D solid element. The external faces of the element are recurved, whereas the sections across the thickness are generated by straight lines. Pairs of points, i_{top} and i_{bottom}, each with given Cartesian coordinates, prescribe the shape of the element.

Let ξ, η be the two curvilinear coordinates in the mid surface of the shell and let ζ be a linear coordinate in the thickness direction. Assuming that ξ, η and ζ vary between –1 and +1 on the respective faces of the element, the Cartesian coordinates of any point of the shell are expressed as

$$\begin{Bmatrix} x \\ y \\ z \end{Bmatrix} = \sum_{i=1}^{8} N_i(\xi,\eta) \left\{ \begin{Bmatrix} x_i \\ y_i \\ z_i \end{Bmatrix}_{mid} + \frac{\zeta}{2} t_i \begin{Bmatrix} l_{3i} \\ m_{3i} \\ n_{3i} \end{Bmatrix} \right\} = \sum_{i=1}^{8} N_i(\xi,\eta) \left\{ \begin{Bmatrix} x_i \\ y_i \\ z_i \end{Bmatrix}_{mid} + \frac{\zeta}{2} t_i v_{3i} \right\}$$

Where Ni (ξ, η) are the quadratic serendipity shape functions in (ξ, η) plane, $\mathbf{t}_i$ is the thickness of the shell at I^{th} node and v_{3i} is the nodal vectors along the thickness direction.

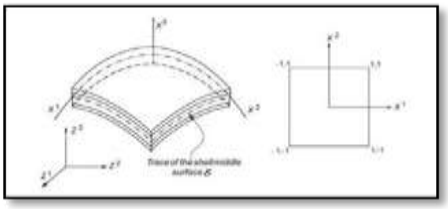

Fig:1 The generic shell element and the parametric space

The square plate is undergoes through the non-linear effect i.e the degenerated shell element is having the nodal points at the centers along with the corner nodes

In the natural coordinate system (ξ , η) the eight shape functions are,

$$N_1 = \tfrac{1}{4}(1-\xi)(\eta-1)(\xi+\eta+1) \qquad N_5 = \tfrac{1}{2}(1-\eta)(1-\xi^2)$$
$$N_2 = \tfrac{1}{4}(1+\xi)(\eta-1)(\eta-\xi+1) \qquad N_6 = \tfrac{1}{2}(1+\xi)(1-\eta^2)$$
$$N_3 = \tfrac{1}{4}(1+\xi)(1+\eta)(\xi+\eta-1) \qquad N_7 = \tfrac{1}{2}(1+\eta)(1-\xi^2)$$
$$N_4 = \tfrac{1}{4}(\xi-1)(\eta+1)(\xi-\eta+1) \qquad N_8 = \tfrac{1}{2}(1-\xi)(1-\eta^2)$$

We have $\sum_{i=1}^{8} N_i = 1$ at any point inside the element. The displacement field is given by

$u = \sum_{i=1}^{8} N_i u_i \quad , \quad v = \sum_{i=1}^{8} N_i v_i$

which are quadratic functions over the element. Strains and stresses.

Quadrilateral element is quadratic functions, which are better represented. The displacement field may be defined in terms of the three displacement components (u_i, v_i and w_i) and two rotational components (α_i and β_i) at the mid - surface nodes as follows

$$\begin{Bmatrix} u \\ v \\ w \end{Bmatrix} = \sum_{i=1}^{8} N_i(\xi,\eta) \left\{ \begin{Bmatrix} x_i \\ y_i \\ z_i \end{Bmatrix}_{mid} - \frac{\zeta t_i}{2} \begin{bmatrix} l_{1i} & l_{2i} \\ m_{1i} & m_{2i} \\ n_{1i} & n_{2i} \end{bmatrix} \begin{Bmatrix} \alpha_i \\ \beta_i \end{Bmatrix} \right\} = [N_D]\ \{\delta\}$$

where the displacement components (u_i, v_i and w_i) are taken along Cartesian coordinate system (x, y and z) and the rotational components (α_i and β_i) are taken about two mutually perpendicular lines tangential to the mid - surface (not necessarily along ξ - η) having unit vectors as $\{v_{1i}\} = [l_{1i} \quad m_{1i} \quad n_{1i}]^T$ and $\{v_{2i}\} = [l_{2i} \quad m_{2i} \quad n_{2i}]^T$

$\{\delta\} = [u_1 \quad v_1 \quad w_1 \quad \alpha_1 \quad \beta_1 \quad u_2 \quad v_2 \quad \dots \quad \dots \quad \alpha_8 \quad \beta_8]^T$

Strain - Displacement Relations: The components of strain and stresses in directions of orthogonal axes related to the surface ζ = constant are only considered. Let z' be a normal to this surface at any point and let the other two orthogonal axes be x' and y' which are tangent to z'. The strain components are defined by

Jacobian: The derivatives of the global displacements *u*,*v*,*w* with respect to the curvilinear coordinates are related to the derivatives with respect to Cartesian coordinates through the Jacobian by

$$\begin{Bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial u}{\partial y} \\ \dots \\ \dots \\ \frac{\partial w}{\partial z} \end{Bmatrix} = [\Gamma] \begin{Bmatrix} \frac{\partial u}{\partial \xi} \\ \frac{\partial u}{\partial \eta} \\ \dots \\ \dots \\ \frac{\partial w}{\partial \zeta} \end{Bmatrix} \text{ where } [\Gamma] = \begin{bmatrix} [J]^{-1} & 0 & 0 \\ 0 & [J]^{-1} & 0 \\ 0 & 0 & [J]^{-1} \end{bmatrix} \text{ and } [J] = \begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} & \frac{\partial z}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} & \frac{\partial z}{\partial \eta} \\ \frac{\partial x}{\partial \zeta} & \frac{\partial y}{\partial \zeta} & \frac{\partial z}{\partial \zeta} \end{bmatrix}$$

Elemental Stiffness Matrix:

The Elemental Stiffness Matrix is then given by

$[K] = \int_v [B][D'][B]^T\, dx\, dy\, dz = \int_v [B][D'][B]^T |J|\, d\xi\, d\eta\, d\zeta$

II. METHODOLOGY

The software used for analyzing the compression members is 'ABAQUS'.

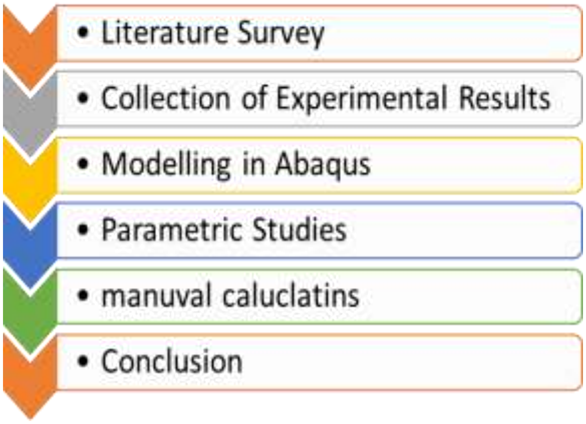

Fig. 1: Objectives Flow Chart

The flow chart representing the various steps followed in this project is summarized as shown in the figure. 1

The flow chart representing the various steps followed in this project is summarized as shown in the figure

The aim is to calculate and compare the deflection values in both analytically and manual methods. A shell structure is modeled using a Finite Element Analysis (FEA) Software i.e. ABAQUS software. The unit load is considered in this project. The unit load was applied to the different shell models using ABAQUS and the deflection values obtained are noted and tabulated.

In order to carry out the manual calculations, one square plate is considered with the clamped boundary conditions. First, the shape functions and displacements were found for the mid surface nodal points then the stiffness matrix is obtained. By using all the three parameters the deflections were obtained. The obtained deflection values are

tabulated and compared with the plate elements of degenerated shells. Further the degenerated shell elements are carried out for the analytically in ABAQUS and deflection values are tabulated and compared.
The deflection values obtained in ABAQUS from all the degenerated shell elements will be compared with the values obtained in the manual and the variation will be studied.
As seen from the above methodology flow chart, the first step involves collection of various literatures related to the current study. The next step is to collect experimental results from the collected literatures and studying their results.

III. FEM ANALYSIS PROCEDURE

A. **Bending Analysis**

The bending analysis is carried to predict the deflection value and the corresponding deflected shape. These are used as parameter in determining the post buckling strength and have additional application for incorporating the input values of the geometric imperfection using first buckling mode shape values.

B. **Procedure for Bending Analysis in ABAQUS**

1. Initially the models are assigned material properties and cross sectional member properties and meshing of the model.
2. An additional step is created in ABAQUS. For bending analysis; all boundary condition from initial step is propagated to this newly created step.
3. For bending analysis to take place an initial displacement of 1mm is applied at the required location on the section.
4. From the obtained results of bending analysis; over-all deflection is selected for incorporating the imperfection modelling.

C. **Procedure for Modelling plates and shells**

The plate element models are created with a part tree in Abaqus software. The model tree was provided with material property with 7.85kg/mm^3 density, elastic property of 2x10^{5}N/mm^2 and yield stress and strain clutched from the experiment. The section is then assigned to obtain the given property onto the model. Now the assembly of model tree had been done so as to provide uniform mesh and hence we create datum coordinates and create partition. The partition is created for the pitch distance for the number of bolts obtained in the calculation. In the step module the step is given as linear perturbation and the number of Eigen values required is given as 50. The models were meshed using Part Instance and seeded later. The analyses were resulted with the job done and monitor the obtained results.

Square Plate element:

Material properties of the models are the Young's modulus E = 2 X 10^{11} N/m^2; the Poisson's ratio μ = 0.3 and the ratio of side to thickness (a / h) is 10 and length of square plate is 2000mm X 2000mm. The square plate with clamped boundary conditions under a uniformly distributed (UDL) load intensity is unit load.

Single barrel:

Material properties of the models are the Young's modulus E = 2 X 10^{11} N/m^2; the Poisson's ratio μ = 0.3 and the thick ness barrel is 6mm and the length of the barrel is 2000mm. width of the barrel is 1000mm. the crown height of the barrel is 200mm. single barrel with clamped boundary conditions under a uniformly distributed (UDL) load intensity is unit load.

Dome:

Material properties of the models are the Young's modulus E = 2 X 10^{11} N/m^2; the Poisson's ratio μ = 0.3 and the thick ness of the dome roof is 6mm and the side plates are 4mm and the length of the dome is 600mm. width of the dome is 600mm. the crown height of the dome is 300mm. The dome with clamped boundary conditions under a uniformly distributed (UDL) load intensity is unit load

IV.**RESULTS AND DISCUSSION**

A.General

The results obtained after performing FEM analysis for plate, single barrel and dome models have been listed in detail.

A. Analytical Results

The liner elastic analysis were carried out for the plate, single barrel and dome for the fixed boundary conditions. And compared with manual calculations

B.1 Square plate

B.1.1 Square plate with 2 X 2 mesh.

Bending analysis is initially carried out for the member by requesting for Eigen value using which the imperfection is implemented into the Abaqus file. Figure 2 shows the deformed shape of the member before implementing the imperfection in ABAQUS.

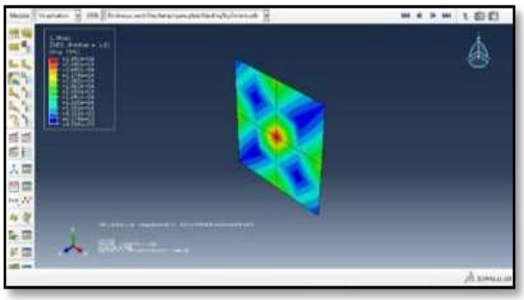

Fig:2 Deformation of square plate with 2 X 2mesh

B.1.1.1 Load vs. deflection graph for Square plate with 2 X 2 mesh.

The following figure 3 shows Load vs. deflection graph of Square plate with 2 X 2 mesh.

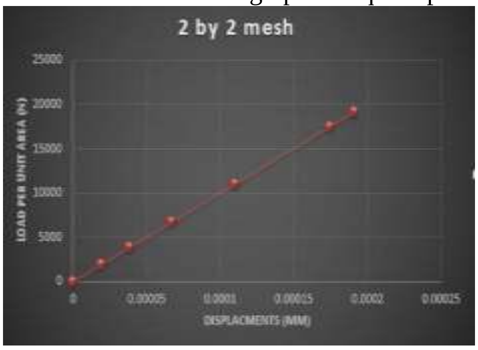

Fig: 3 Deflection graph of square plate with 2 X 2 mesh

B.1.2 Square plate with 4 X 4 mesh

Bending analysis is initially carried out for the member by requesting for Eigen value using which the imperfection is implemented into the Abaqus file. Figure 4 shows the deformed shape of the member before implementing the imperfection in ABAQUS.

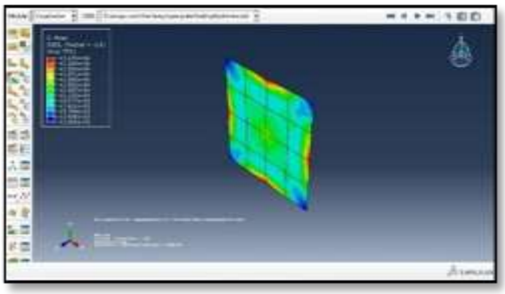

Fig: 4 Deformation of square plate with 4 X 4 mesh

B.1.2.1 Load vs. deflection graph for square plate with 4 X 4 mesh

The following figure 5 shows Load vs. deflection graph of Square plate with 4 X 4 mesh.

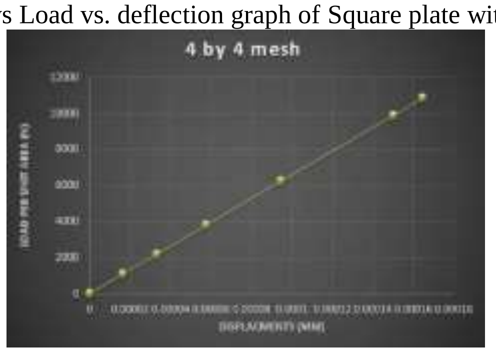

Fig: 5 Deflection graph of square plate with 4 X 4 mesh

B.1.3 Square plate with 8 X 8 mesh

Bending analysis is initially carried out for the member by requesting for Eigen value using which the imperfection is implemented into the Abaqus file. Figure 6 shows the deformed shape of the member before implementing the imperfection in ABAQUS.

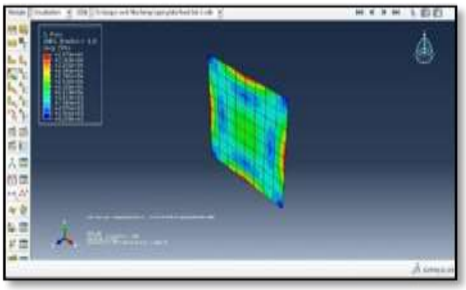

Fig:6 Deformation of square plate with 8 X 8 mesh

B.1.3.1 Load vs. deflection graph for square plate with 8 X 8 mesh

The following figure 7 shows Load vs. deflection graph of Square plate with 8 X 8 mesh..

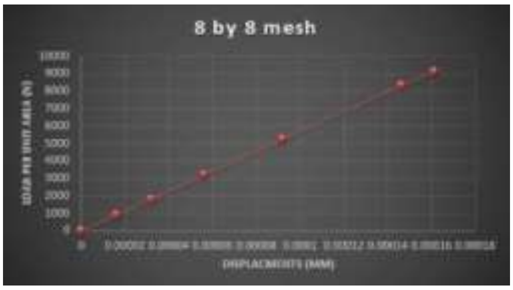

Fig: 7 Deflection graph of square plate with 8 X 8 mesh

B.1.4 Square plate with 50mm interval mesh

Bending analysis is initially carried out for the member by requesting for Eigen value using which the imperfection is implemented into the Abaqus file. Figure 8 shows the deformed shape of the member before implementing the imperfection in ABAQUS.

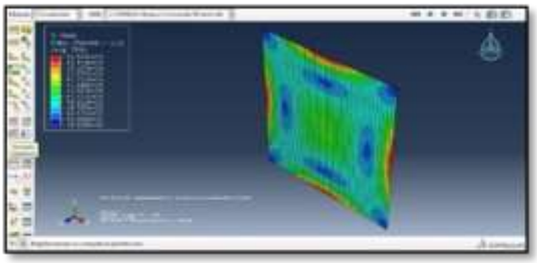

Fig:8 Deformation of square plate with 50 mm interval mesh

B.1.4.1 Load vs. deflection graph for square plate with 50mm interval mesh

The following figure 9 shows Load vs. deflection graph of Square plate with 50mm interval mesh.

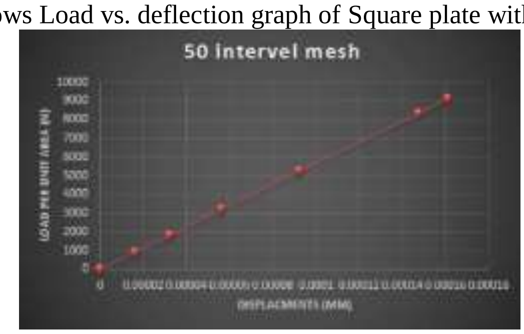

Fig: 9 Deflection graph of square plate with 50mm interval mesh

B.1.5 Load vs. deflection graph Comparison of square plate with different mesh divisions

The fig 10 represents a comparison of square plate with different mesh divisions.

1. From the following figure we can see that the accuracy of the deflection value increases by increasing the mesh divisions
2. The square plate with 8 X 8 mesh is having deflection of **0.00016104**

3. The square plate with 50mm interval mesh is having deflection of **0.000161081**
4. When comparing with 8 X 8 mesh to 50mm interval mesh we observed that the variation between 8 X 8 mesh to 50mm interval mesh is 1%
5. So I adopted 50 mm interval mesh case for the remaining analysis

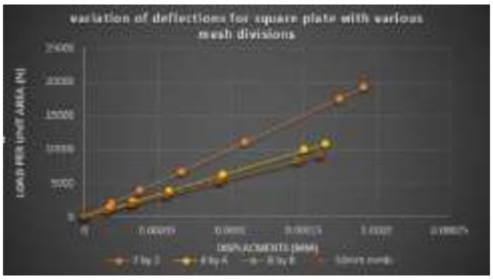

Fig:10 Deflection graph of square plate with different mesh divisions

B.2 Single barrel

Bending analysis is initially carried out for the member by requesting for Eigen value using which the imperfection is implemented into the Abaqus file. Figure 11 shows the deformed shape of the member before implementing the imperfection in ABAQUS.

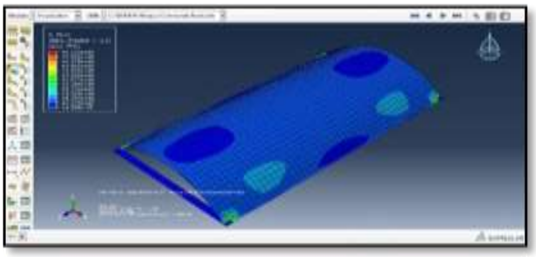

Fig:11 Deformation of single barrel with 50 mm interval mesh

B.2.1 Load vs. deflection graph for Channel Member without Perforations

The following figure 12 shows Load vs. deflection graph of Single Barrel with 50mm interval mesh.

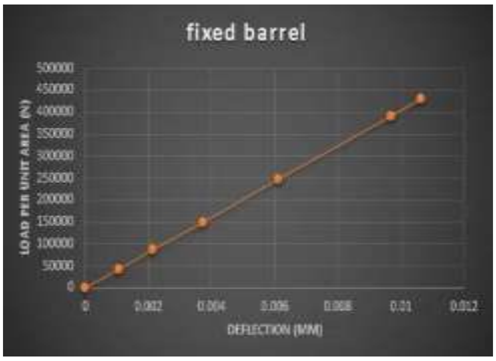

Fig: 12 Deflection graph of single barrel with 50mm interval mesh

B.3 Dome

Bending analysis is initially carried out for the member by requesting for Eigen value using which the imperfection is implemented into the Abaqus file. Figure 13 shows the deformed shape of the member before implementing the imperfection in ABAQUS.

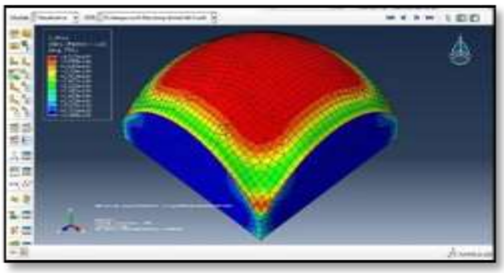

Fig:13 Deformation of dome with 50 mm interval mesh

B.3.1 Load vs. deflection graph for Channel Member with Single Perforations

The following figure 14 shows Load vs. deflection graph of dome with 50 mm interval mesh

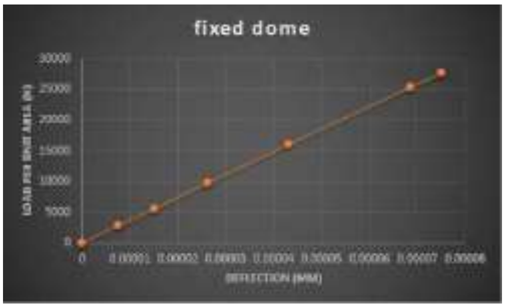

Fig:14 Deformation of dome with 50 mm interval mesh

V. Conclusion

In the current study comprehensive analysis carried out on the manual calculations of the finite element method to the computerised finite element Analysis program (ABAQUS). The deflection behaviour of the plates and shell elements are investigated. In the study degenerated shell elements were analysed for different boundary conditions. The results of the analysis are evaluated in terms of the parameters. On the basis of the results the following conclusions can be done

1. The maximum deflection values obtained from the manual calculations In the case of square plate with 2 X 2 mesh division for fixed boundary condition is **0.0002022 mm**
2. The maximum deflection values obtained from the ABAQUS In the case of square plate with 2 X 2 mesh division for fixed boundary condition is **0.0001912 mm**
3. The variation is nearly **5%** for square plate with 2 X 2 mesh divisions**.** From the literature this variation will be similar in all the models.
4. The maximum deflection values obtained from the ABAQUS In the case of square plate with 4 X 4 mesh division for fixed boundary condition is **0.0001648 mm.**
5. The maximum deflection values obtained from the ABAQUS In the case of square plate with 8 X 8 mesh division for fixed boundary condition is **0.00016104 mm.**
6. The maximum deflection values obtained from the ABAQUS In the case of square plate with 50 mm mesh interval for fixed boundary condition is **0.000160681mm.**
7. From the analysis result in the case of 8 X 8 mesh division and 50 mm mesh interval the maximum deflection values are more or less same. Hence 50 mm mesh interval can be adopted for the analysis using ABAQUS.
8. Since it is difficult to carry the analysis manually for finer mesh intervals such as 50 mm mesh intervals, it is suggested that the ABAQUS program can be adopted.
9. From the analysis result the dome (doubly curved shells) and single barrel (singly curved) we can conclude that the doubly curved shells are stiffer than the singly curved shells.

References

1. A. Vishal Simha and S.N. Patel, **bending analysis of laminated composite plates using degenerated shell elements**, advances in structural engineering and mechanics (ASEM13), 2013.
2. Bathe, K. J., and Ho, L. W., **A Simple and Effective Element for Analysis of General Shell Structures**, *Computers and Structures*, Vol. 13, pp. 673-681, 1981.
3. D Marinkovi´c, H K¨oppe and U Gabbert, **Degenerated shell element for Geometrically nonlinear analysis of thin-walled piezoelectric active structures,** Smart Mater. Struct. **17** (2008) 015030, 2008.
4. McNeal R. H. and Harder R. L., **Refined Four Node Membrane Element with Rotational Degrees of Freedom**, *Computers and Structures*, Vol. 28, 1988, pp. 75-84.
5. Henrik Fredslund Hansen & Christian Gram Hvejsel (2007), **efficient finite element formulation for analysis and optimization of laminated composite shell structures**, Department of Mechanical Engineering, Aalborg University, Denmark.
6. I.E Umeonyiagu, N.P Ogbonna, **development of eight nodes discretekirchoff quadrilateral (dkq8) elements in java,** International Journal of Civil and Structural Engineering Research ISSN 2348-7607, Vol. 5, Issue 1, pp: (73-83), 2017.
7. I. Ergatoudis, b. M. Irons and 0. C. Zienkiewic, **curved isoparametric "quadrilateral" elements for finite element analysis**, ht. J. Solids strucrures. 1968, vol. 4, pp. 31 to 42, 1968.
8. Kaushalkumar M. Kansara (2004)**, Development of Membrane, Plate and Flat Shell Elements in Java,** Department of Civil Engineering Virginia Polytechnic Institute and State University.
9. Pravin Kumar (1987), **large deflection elastic-plastic analysis of cylindrical shells using the finite strip method**, department of civil engineering The University of British Columbia Vancouver, Canada.