RESEARCHERID
THOMSON REUTERS

[Burad* *et al.*, 6 (7): July, 2017]
IC™ Value: 3.00

ISSN: 2277-9655
Impact Factor: 4.116
CODEN: IJESS7

IJESRT

INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

NON-LINEAR FINITE ELEMENT ANALYSIS OF SHELLS USING PARAMETRIC STUDY

Jairaj Burad*, Dr. S. S. Angalekar

* PG student, Department of Civil Engineering, Sinhgad college of Engineering
Pune, India
Associate professor, Department of Civil Engineering, Sinhgad college of Engineering
Pune, India

DOI: 10.5281/zenodo.823033

ABSTRACT

The shells are the structural elements which are characterized by their geometry. Which can replace heavy slabs for the roofing of a building at a large height. They also provide a large column free area. The strength of the shell is through its geometry and not through its mass. Thus, it provides less amount of building materials. So the non-linear finite element analysis of the shells help us to know their behavior after the elastic limit. A reliable nonlinear finite element analysis of shell structures, there are two main ingredients to be considered i.e., formulation of appropriate shell element and development of numerical algorithms for solving the equations of motion. The study is performed on a Finite element analysis based software i.e., ANSYS. The non-linear analysis is performed by parametric study in which different parameters of the shells are changed and the result is analysed and is compared with the analytical results from previous papers. As there is rapid increase in the use for thin shells for the construction to reduce the amount of materials providing the same strength, this study is helpful for the construction industry.

KEYWORDS: Non-Linear Analysis, Finite Element Method, Thin Shells

I. INTRODUCTION

A Shell is a thin, curved membrane or slab, usually of reinforced concrete, that functions both as structure and covering, the structure deriving its strength and rigidity from the curved shell forms that utilize the natural strength and stiffness of shell forms with great economy in the use of material. Thin-shell structures are light weight constructions using elements. These elements are typically curved and are assembled to large structures. Reinforced concrete shells, due to their special characteristics from the economical and strength point of view, have been extensively used to cover large spans as hangars, industrial buildings exhibition halls and sports grounds. With the arrival of the finite element method and the development of computers with memory for handling bandwidth of equations, the analysis of concrete shell structures has progressed significantly, and most of the problems concerned with the design of such structure were overcome.

Simultaneously, the techniques to deal with nonlinear problems and the conception of analytical models to predict a more realistic behaviour of reinforced concrete structures, in the special shells, has systematically advanced over the last years Finite element modelling for use in the analysis of several types of reinforced concrete thin shells can be carried out by using flat finite elements, curved elements formulated on the basis of shell theories and elements derived from three-dimensional elements by applying the degeneration process. Flat elements proved to be adequate in the analysis of shells with no severe gradient in stress variation. One disadvantage of flat elements is the possible presence of discontinuity bending generated at the junction lines due to their geometric shape approximation. Many general curved shell finite elements were developed and are available to be incorporated into finite element codes. Usually, these finite elements have a greater number of nodal unknowns than the flat elements, making the imposition of boundary conditions more difficult, besides increasing the bandwidth of the system of linear equation. Rigid body modes are also more difficult to be reproduced due to the curved geometry. However, they give a better geometrical representation of the shell surface, providing very good results.

ISSN: 2277-9655
Impact Factor: 4.116
CODEN: IJESS7

II. FINITE ELEMENT METHOD

The finite element method (FEM) is a numerical technique for finding approximate solutions to boundary value problems. It uses subdivision of a whole problem domain into simpler parts, called finite elements. FEM provides methods in which the structure is divided into very small but finite number of elements, to approximate a more complex equation over a larger domain. Engineering structures that have complex geometry and loads, are very difficult to analyze or have no theoretical solution. However, in FEA, a structure of this type can be analysed

Complex Engineering problems without knowing the governing equations can be solved. FEA software provides a complete solution including deflections, stresses, reactions etc. FEA technique facilitates an easier and a more accurate analysis.

III. ELEMENT LIBRARY

To study the responses of the shell under different loading conditions different 2D and 3D elements are used. Four noded (Shell 63) and eight noded (Shell 281) 2D elements are taken along with 8 noded (Solid 73) and 20 noded (Solid 186) 3D elements in Ansys APDL 16.0 software package. In case of stress analysis problems with simple straight boundaries like beams or plates subjected to in plane forces, rectangular elements are used. The main advantage of these elements is the simplicity in formulation it is easy to develop the relevant routines giving them good exercise to deal with more complex element later.

IV. METHODOLOGY

Cylindrical shell element will be analysed in both 2D and 3D with possible us of 4 noded, 8 noded, and 20 noded elements. The thickness of concrete is 12.5 cms. The radius of shell (R) is kept constant at 7.62 m. The shell element will be withheld by span to radius ratio as 1, 2 and 3. Semi central angle (Φ) will be taken as 40 degrees. The length of the span (L) will be taken according to the span to radius ratio as 7.62 m (short shell), 15.24 m (moderate shell) and 22.86 m (long shell).
Young's modulus (E) = 0.250 x 10^8 KN/m^2
Poisson;s ratio (μ) = 0.15
Density of concrete (γ) = 0.250 x 10^2 KN/m^3

Loading to be considered:
Self-weight
Self-weight and Wind Load
Self-weight and Earthquake Load
Due to Symmetry only a quarter part of shell is taken for the analysis. The different meshing sizes taken for the analysis are: 2x2 mesh, 4x4 mesh and 8x8 mesh. The response of these meshing is studied for different loading conditions in elastic state.

The meshing of short shell is shown in the figure below:

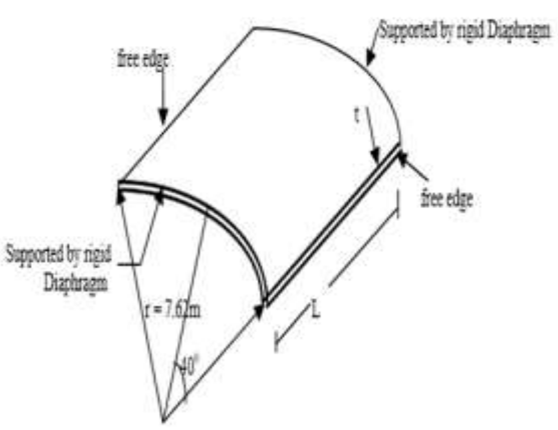

Figure 1- Geometry of the shell.

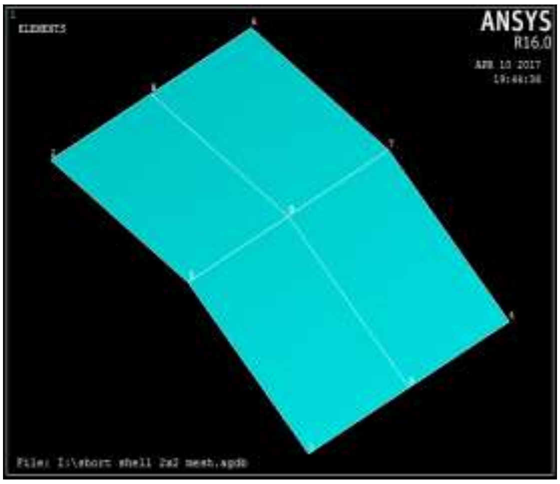

Figure 2- 2x2 mesh of the shell.

ISSN: 2277-9655
Impact Factor: 4.116
CODEN: IJESS7

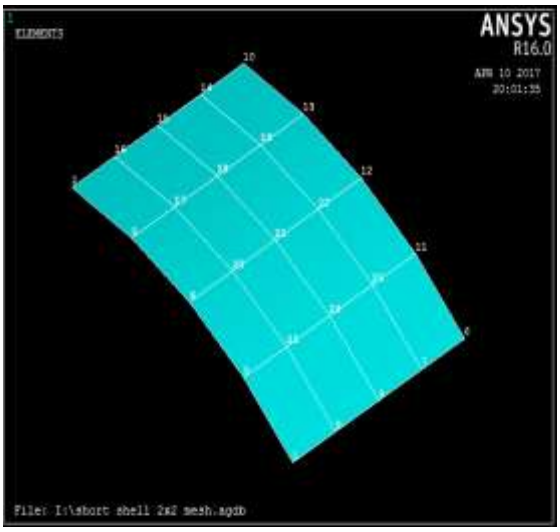

Figure 3- 4x4 mesh of the shell.

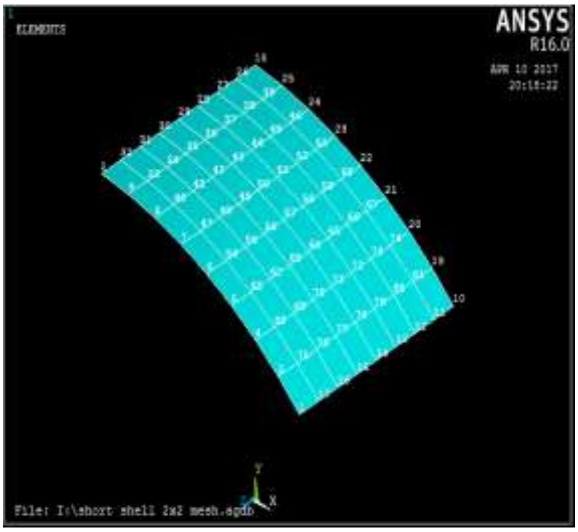

Figure 4- 8x8 mesh of the shell

V. RESULTS AND DISCUSSION

The deflected profiles for self-weight acting on the shells are calculated using ANSYS APDL 16.0. Different meshing sizes are adopted for analysis and the results are investigated for the variation of displacement for different shells. Results for Longitudinal Displacements at different regions are investigated and plotted against semi-central angles. The variation of displacement of different types of shells for different meshing sizes (2x2 mesh, 4x4 mesh, 8x8 mesh) are shown in the graphs shown below:

1. For Self-weight :

(A) 4 Noded 2D Shell Element

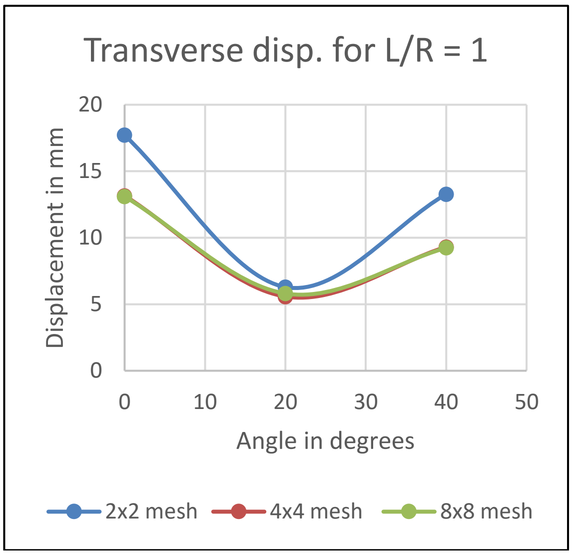

Figure 5- Transverse Displacement for short shell

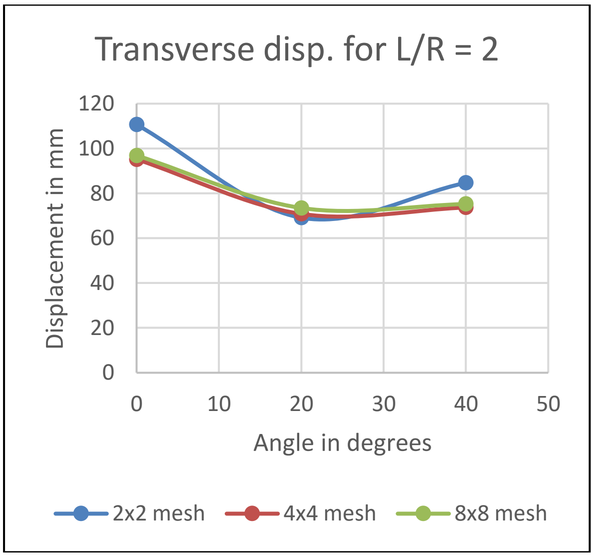

Figure 6- Transverse Displacement for moderate shell

Its has been observed that as the meshing size is increased we get accurate results. The results for mesh sizes 4x4 and 8x8 gives nearly similar as compared to the 2x2 mesh. The differences in the transverse displacement of the centre of the shell and the edges decrease as the span/depth ratio increases.

ISSN: 2277-9655
Impact Factor: 4.116
CODEN: IJESS7

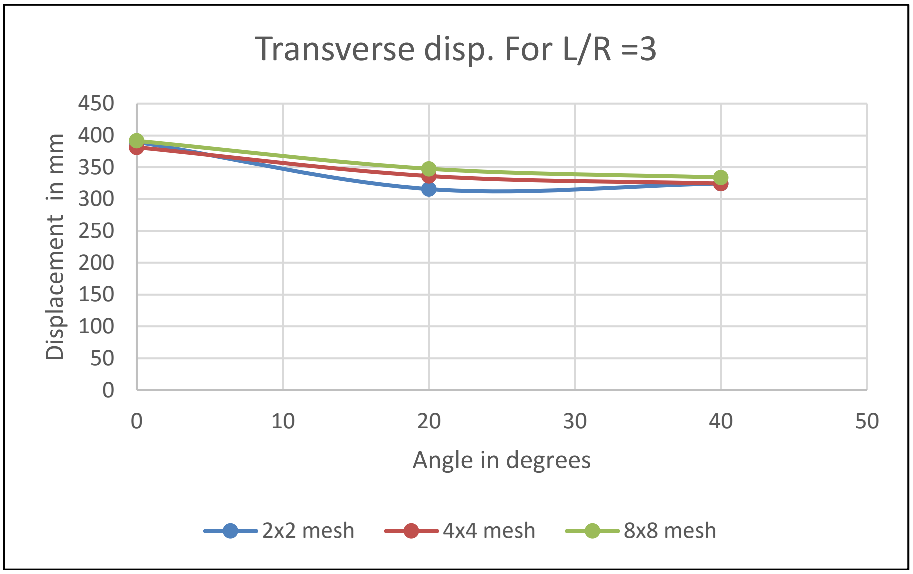

Figure 7- Transverse Displacement for long shell

Its has been observed that as the meshing size is increased we get accurate results for the transverse displacements. The results for mesh sizes 4x4 and 8x8 gives nearly similar as compared to the 2x2 mesh. The differences in the transverse displacement of the centre of the shell and the edges decrease as the span/depth ratio increases.

The results of the transverse displacements of the shells shows that we get more accurate results for 8x8 meshing. So we take the values obtained for 8x8 meshing size for different span to depth ratios and also for different elements (2D & 3D).

The variation of the transverse displacements to the different span to depth ratios (1, 2, 3) are shown below for a 4 noded 2D element.

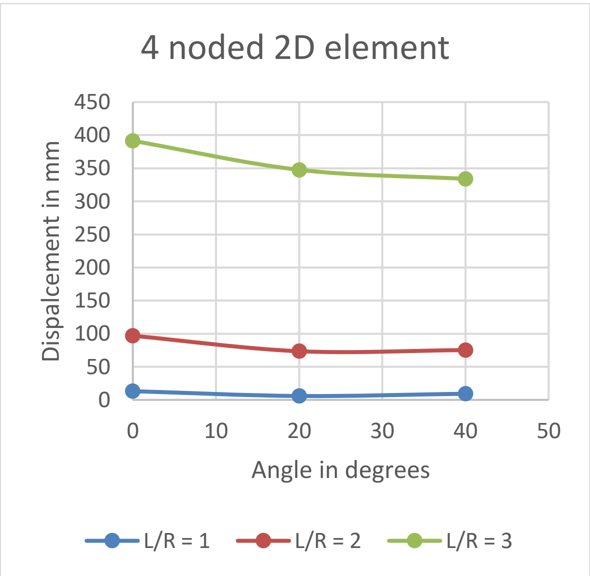

Figure 8-Displacements for various L/R ratios .

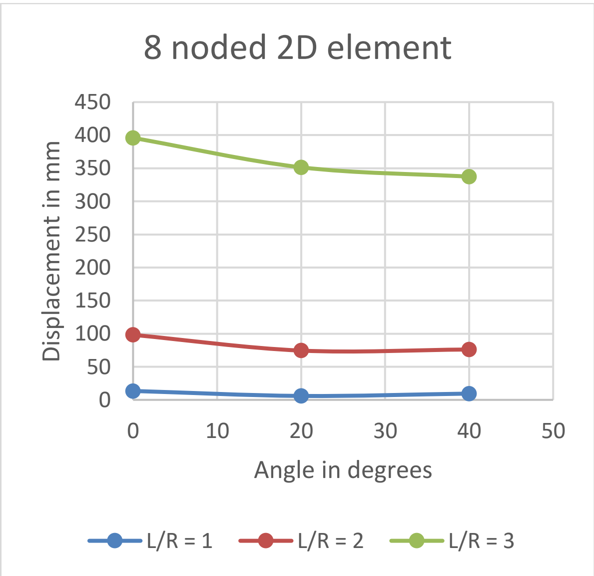

Figure 9- Displacements for various L/R ratios

ISSN: 2277-9655
Impact Factor: 4.116
CODEN: IJESS7

The graphs above shows that the transverse displacements increase by smaller value by increasing the L/R ratio from 1 -2 and it drastically increases for the L/R ratio 3. It indicates that longer shells are more prone to failures as the self-weight deflection is very large.

The variation of transverse displacements for different elements shows similar behaviour for all the four elements used.

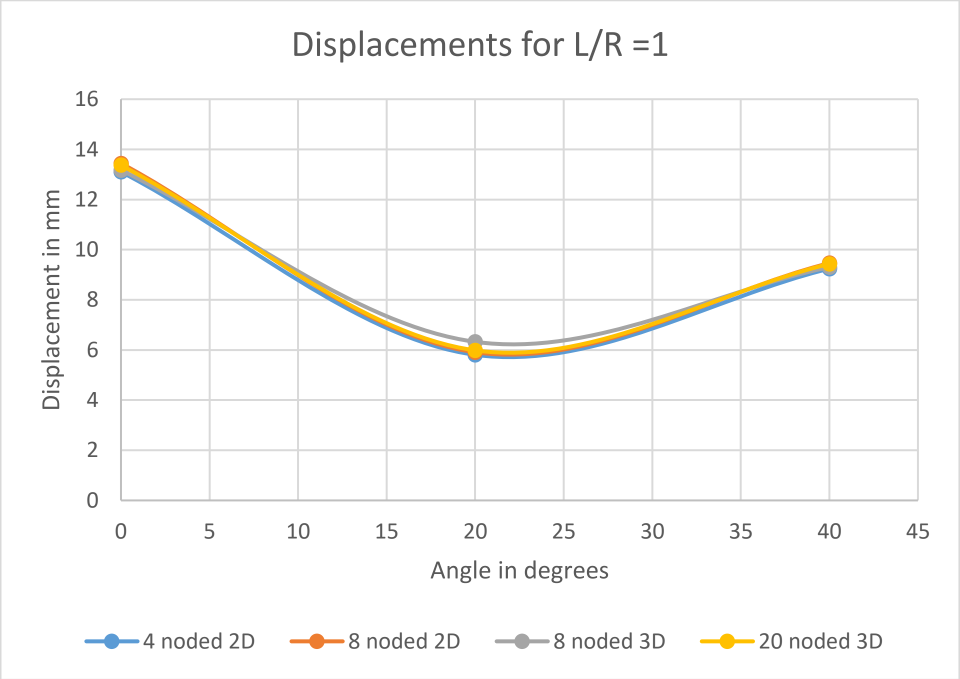

Figure 10-Displacements for various elements .

For centre nodes, 8 noded 3D element shows maximum deflection and for the edge nodes, the 20 noded 3D elements shows the maximum deflection.

VI. CONCLUSION

1. Behavioural response of different 2D and 3D elements is investigated and it is observed that they give behavioural response in a similar manner.
2. The response of the shells is being observed for different mesh sizes (2x2 mesh, 4x4 mesh, 8x8 mesh) and the differences in the transverse displacement of the centre of the shell and the edges decrease as the span/depth ratio increases.
3. The variation of displacement for different L/R ratios indicates that longer shells are more prone to failures as the self-weight deflection is very large.
4. The elastic zone results are firstly matched with the previous results so as to check the authenticity of the results obtained.

VII. FUTURE SCOPE

In the future scope of this study, further researchers may do their work for the different shell thickness and also taking the reinforcement in the shells which may alter the displacements of the shells. It may also be beneficial to change some more parameters and also different grades of concrete to be used.

VIII. ACKNOWLEDGMENT

Authors would like to express their sincere thanks to STE Society, SCOE Management for giving an opportunity and for their valuable support throughout. Authors would like to express their sincere thanks to the help provided by all teaching and non -teaching staff from civil department of Sinhgad College of Engineering, Pune.

IX. REFERENCES

[1] Eugenio Ruocco, 'Elastoplastic buckling analysis of thin-walled strucutures', 'Aerospace Science and Technology', 21 July 2014, pp. 152-165
[2] Ashique Jose, Ramadass S, et.al, 'Parametric study on the structural forces and the moments of Cylindrical Shell Roof using ANSYS', 'American Journal of Engineering Research (AJER)', Vol. 3, 2013, ISSN:2320-0847, pp. 50-61
[3] T. Dirgantara, M.H. Aliabadi, 'Elastoplastic Boundary Element Method for Shear Deformable Shells', 'Engineering Structures 45' 15 July 2012, pp.62-67]
[4] S.S.Angalekar, Dr.A.B.Kulkarni, 'Behavioural Response of Elasto-Plastic Deformations for Concrete Cylindrical Shells Using 4 Noded Flat Plate Elements by Finite Element Method of Analysis with an Incremental Loading Technique', 'International Journal of Computer Technology', Vol 2 (3), July 2011, ISSN:2229-6093, pp.410-413
[5] S.S. Angalekar, et.al, 'Linear Elastic Behaviour of Cylindrical Shell', 'International Journal of Engineering Science and Technology (IJEST)', Vol. 3, No.2 Feb 2011, ISSN: 0975-5462, pp.1648-1654
[6] Hyuk Chun Noh, 'Ultimate Strength if Large Scale Reinforced Concrete Thin Shell Structures', 'Thin walled Structures 43', 2005, pp. 1418-1443
[7] Antonio Bilotta et.al., 'Mixed Finite Elements for the Elastoplastic Analysis of 2D Continua', 'European Congress on Computational Methods in Applied Sciences and Engineering', 2004, pp.614-627
[8] Aloisio Ernesto Assan, 'Nonlinear Analysis of Reinforced Concrete Cylindrical Shells', 'Computers and Structures 80' , 2002, pp. 2177-2184.
[9] Zeinkiwicz O. C. amd Tayler R. L.(2000): "Shells as an Assembly of Flat Elements", Finite Element Method, Vol.-2, Solid Mechanics, Butterworth Publication, pp. 216-243.

CITE AN ARTICLE

Burad, J., & Angalekar, S. S. (2017). NON-LINEAR FINITE ELEMENT ANALYSIS OF SHELLS USING PARAMETRIC STUDY. *INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY*, *6*(7), 101-115. doi:10.5281/zenodo.823033