Finite Element Analysis Of Hyperbolic Paraboloid Shell By Using ANSYS

[1] S. S. Kadam, [2] G. R. Gandhe , [3] D. H. Tupe

Department of Civil Engineering, Deogiri Institute Of Engineering And Management Studies,

Dr. BAMU University, Aurangabad, Maharashtra, India.

---***---

Abstract ***Finite element analysis is performed on hyperbolic paraboloid shell by using finite element analysis based on software ANSYS(15). And the comparison results are studied and compared out by using the material properties of the reinforced cement concrete and ferrocement and the appropriate loads acting on the structure. Analysis results shows the effect of stress results with respect to their characteristics strength. Hyperbolic paraboloid shell are used for large scale iconic structures worldwide. These analysis and design methodologies are not codified in most countries and there is limited design guidance available. Various forms of shell structure are available for the covering large area but as shell is a highly indeterminate structure it involves complex mathematical calculations. As a reliable finite element package program, ANSYS (15), is used for structural analysis. This results of the analysis showed that reasonably accurate results were obtained even when the modeling of shells using fewer elements compared to other shell element types.***

Keywords ***Thin HP shell, Finite Element Analysis, Static Structural analysis, Regression Analysis***

1.Introduction

Structural designers are a new breed of engineers, who act as bridges between conventional structural engineers and architects. Their role requires them to formulate the best possible design based on certain requirements, which depends on several factors. Thus, it is fair to say that they define the backbone of project. This causes a lot of responsibilities to fall on the designer. A good structural designer tries to incorporate several factors to make a complete design. This includes taking into account aesthetic, structural and constructional factors. Incorporating every factor heavily influences the design, as will be evident from this particular project. This approach obviously causes several obstacles, which can cause complications in the design process. This makes it imperative for a structural designer to have intuition and foresight, along with good structural knowledge to deal with these problems. It remains the primary responsibility of the structural designer to ensure that the structure performs all the functions, it was meant to perform before the start of the design process. Shells experience something called 'snap-back' behaviour, which is characterized by sudden loss in load carrying capacity, leading to immediate failure. Furthermore shells collapse without warning, unlike other structures which show considerable visual deformations before collapse. Now, The Finite Element analysis can be used to comparison by reinforce cement concrete and ferrocement material together Finite Element method based on software ANSYS (15), They can be measure how many large deformation generate in which material to be better results. ANSYS provide the most essential tools for a structural designer

2.Problem Statement

2.1Finite Element Modeling In this paper different results found for two sides fixed uniformly distributed

Properties	RCC	Ferrocement
Young's modulus	27386Mpa	79056Mpa
Density	2400Kg/m^3	2000Kg/m^3
Tensile ultimate strength	30Mpa	250Mpa
Poisson's ratio	0.2	0.3
Ordinary mild steel	130 N/mm^2	
High yield strength steel	190 N/mm^2	
½" Sq. welded mesh		0.235Kg/m^2
1" Stucco wire		0.10Kg/m^2
1" Chicken wire		0.190Kg/m^2
1mm size of mesh		20mm

load hyperbolic paraboloid shell model as follows shown geometry

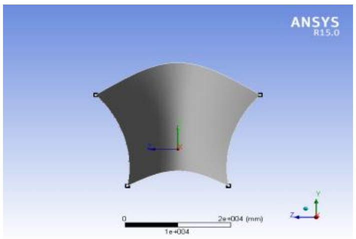

Fig.1Geometry of HP shell

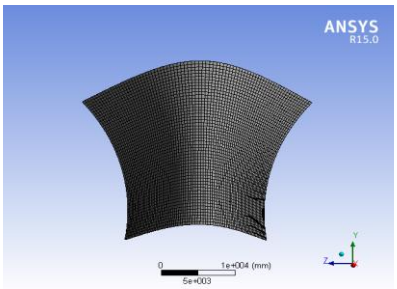

Fig.2Model after applying meshing

Above fig.1 shows the geometry of HP shell and fig.2 shows the quadrilateral fine element of HP shell and the model of meshing called as quadrilateral fine element of HP shell.

The different results found for this HP shell as including two different materials like reinforce cement concrete and ferrocement by comparing those materials to how many deformation generate and which material is better to use for HP shell in better activity in future for less cost and less labour.

In this paper used in ANSYS software based on finite element method like static analysis and also to prepare the regression analysis on graph to calculate the value of static results are compared.

3.Analysis

3.1Finite Element Method

In Finite Element method many engineering problems analytical solutions are not the suitable because the complexity of this material properties, boundary conditions and the structure itself.

3.2ANSYS

ANSYS is the commercial FEM packages having the capabilities ranging from a simple, linear, static analysis to the complex, nonlinear, transient dynamic analysis. This is available in modules. Each module is the applicable to specific problem. For this example, Ansys/civil is the applicable to the civil structural analysis. Similarly Ansys/Flotran is the CFD software applicable to the fluid flow. Advantages of the Ansys compared to the other competitive software's is the availability of the bundled software of pre, post and the Processor.

3.3 Static Analysis The objective of static analysis is to obtain the total deformation on structure of HP shell as well as Normal stresses on X and Y direction, for RCC and Ferrocement material.

3.4 Regression Analysis In this regression analysis the results obtain on ANSYS software by FEM in static analysis that results are obtain the graph and on graph to calculate value of static analysis results and then on graph to calculate two equation find out for RCC and Ferrocement in equation put the 'x' value in X- axis and to calculate 'y' value this value match the final results that comparison called as regression analysis.

4.Results And Discussion

A] Static Analysis For RCC

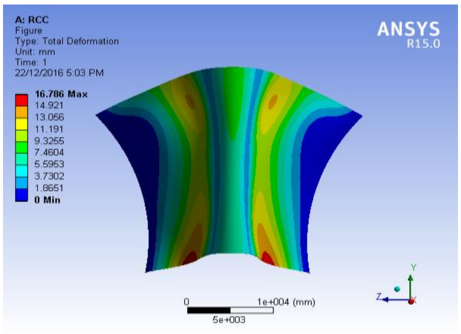

Fig.3 Total Deformation of RCC

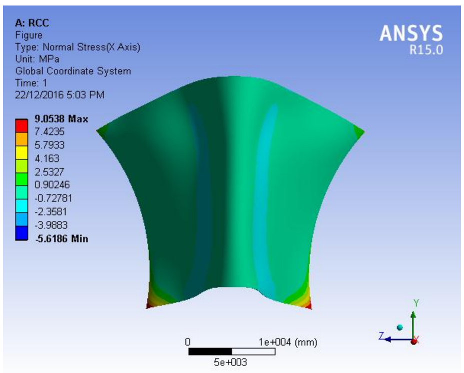

Fig.4Variation of Normal Stress on X-axis

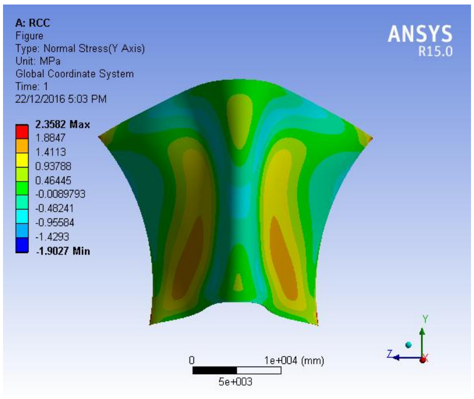

Fig.5Variation of Normal Stress on Y-axis

B] Static Analysis for Ferrocement

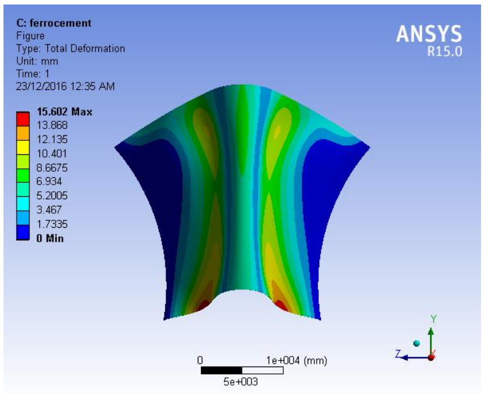

Fig.6 Total Deformation of Ferrocement

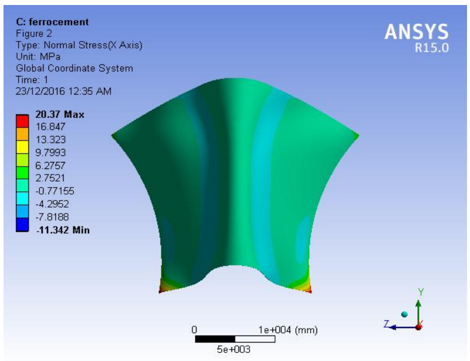

Fig.7Variation of Normal Stress on X-axis

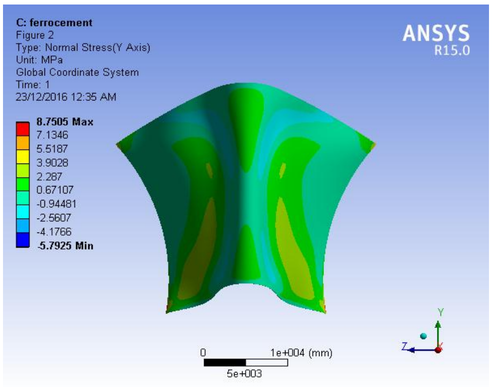

Fig.8Variation of Normal Stress on Y-axis

For RCC Fig.3 shows the total deformation is 16.786mm, Fig.4 and Fig.5 Normal stresses on X and Y axis respectively is 9.0538Mpa and 2.3582Mpa respectively. For Ferrocement Fig.6 shows the total deformation of ferrocement is 15.602mm, Fig.7 and Fig.8 shows the Normal stresses on X and Y axis respectively are 20.37Mpa and 8.7505Mpa respectively.

Table1 Comparison by Total Deformation of RCC and Ferrocement for static analysis

X	TD Rcc ANSYS	TD Rcc Reg.	TD Fr ANSYS	TD Fr Reg.
1000	0	0	0	0
2000	1.8651	1.9347	1.7335	1.6666
3000	3.7302	3.8347	3.467	3.3666
4000	5.5953	5.7347	5.2005	5.0667
5000	7.4604	7.63475	6.934	6.7668
6000	9.3255	9.5347	8.6675	8.4669
7000	11.191	11.4347	10.401	10.1670
8000	13.056	13.3346	12.135	11.8671
9000	14.921	15.2346	13.868	13.5673
10000	16.786	17.1346	15.602	15.2675

Table2 Comparison by Normal Stress on X co-ordinate

X	6x RCC ANSYS	6x RCC Reg.	6x FR ANSYS	6x FR Reg.
1000	-5.6186	-5.6488	-11.342	-11.3659
2000	-3.9883	-4.0488	-7.8188	-7.8659
3000	-2.3581	-2.4488	-4.2952	-4.3659
4000	-0.72781	-0.8488	-0.77155	-0.8658
5000	0.90246	0.7511	2.7521	2.6342
6000	2.5327	2.3511	6.2757	6.1342
7000	4.163	3.9511	9.7993	9.6343
8000	5.7933	5.5511	13.323	13.1345
9000	7.4235	7.1511	16.847	16.6346
10000	9.0538	8.7511	20.37	20.1348

Table3 Comparison by NS on Y co-ordinate

X	6y RCC ANSYS	6y RCC Reg.	6y FR ANSYS	6y FR Reg.
1000	-1.9027	-1.8760	-5.7925	-5.8729
2000	-1.4293	-1.3760	-4.1766	-6.5999
3000	-0.95584	-0.8760	-2.5607	-2.5449
4000	-0.48241	-0.3760	-0.94481	-0.9079
5000	-0.00898	0.123	0.67107	0.7111
6000	0.46445	0.6239	2.87	2.3121
7000	0.93788	1.1239	3.9028	3.8951
8000	1.4113	1.6239	5.5187	5.4601
9000	1.8847	2.1239	7.1346	7.0071
10000	2.3582	2.6239	8.7505	8.5361

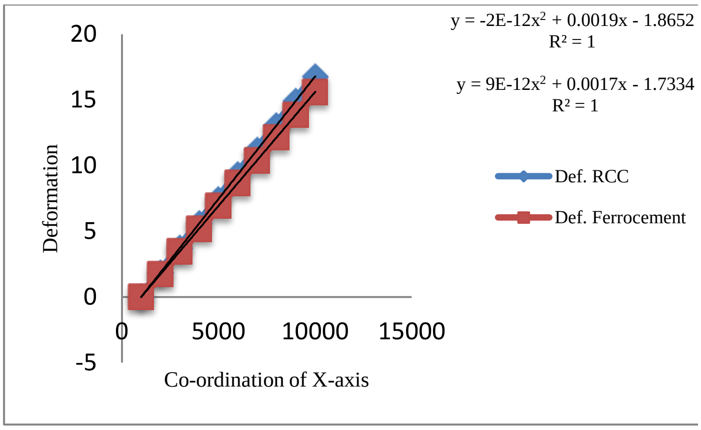

Fig.9Total deformation for static Analysis

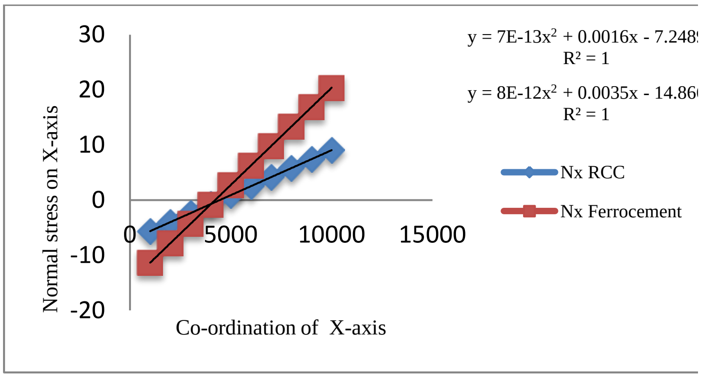

Fig.10Normal stress on X co-ordinate

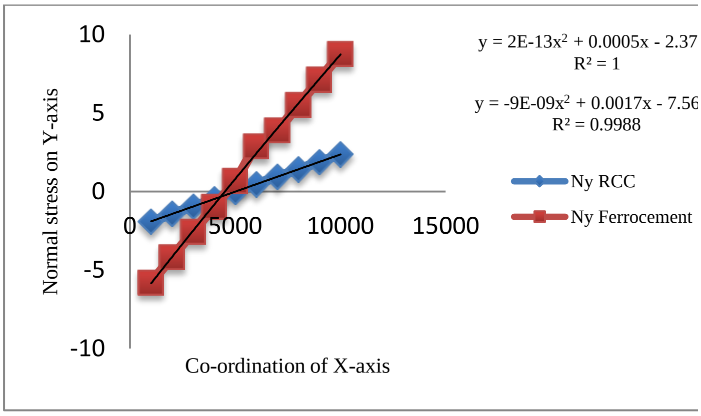

Fig.11Normal stress on Y co-ordinate

Above table 1 shows the comparison by total deformation of RCC and Ferrocement for static analysis by compared with Regression analysis on graph equation shown Fig.9, It may be that the value of regression analysis is near about statically analysis in ANSYS is done. Above table 2 shows the comparison by Normal stresses on X co-ordinate of RCC and Ferrocement for static analysis by compared with Regression analysis on graph equation shown Fig.10, It may be that the value of regression analysis is near about statically analysis in ANSYS is done. Above table 3 shows the comparison by Normal stresses on Y co-ordinate of RCC and Ferrocement for static analysis by compared with Regression analysis on graph equation shown Fig.11, It may be that the value of regression analysis is near about statically analysis in ANSYS is done.

5. Conclusions

In this paper shows the FEM based software ANSYS is used to analysis the HP shell by using two different materials like RCC and Ferrocement. It includes also the Regression analysis on graph equation to shows the results are same or nearly equals. Actually, In this project also determine how many deformation generate by using material RCC by compared with Ferrocement . And, Both results are shown the Ferrocement material is better use to compared by RCC material because, The total deformation of RCC in ANSYS is 16.786mm and in regression is 17.1346mm is higher than the total deformation of Ferrocement in ANSYS is 15.602mm and in regression is 15.2675mm. So, It may be conclude that the Ferrocement is better active as compared to RCC material to better use in future scope for better activity and also less labour are required, minimum cost, less time are required in Ferrocement

REFERENCES

[1]Vlasov,V.Z.(1944), The Basic Differential Equations in the General Theory of Elastic Shells, Prikladnaya Matematika Mekhanika, Vol.8. (Translated from Russian

into English by TNACA, Technical Memorandum 1241, February, 1951).

[2]Vlasov,V. Z. (1958), "Allgemeine Schalentheorie and Ihre Anwendung in der Technik", Akademie –Verlag GmbH, Berlin.

[3]"Indian Standard Code of Practice for Design of Loads (Other than Earthquake) for Buildings and Structures", PART 3,1987.

[4]J. N. Bandopadhya and A. K. Aditya (1989), Simplified Bending analysis of doubly curved shell structures, Computers & Structures, 33, 3, 781-784.

[5]University of the Witwatersrand, School of Civil and Environmental Engineering, "Groin Vault Design", CIVN4003: Design Project 2008, Adil Saloojee Daniel Cromberge Muhammad Bodhania Shaahid Hansa Zakarriyyah Ameen, Supervised by Professor M. Gohnert and Dr. A. Fitchett

[6]CRC Press - Technology and engineering, "Handbook of Structural Engineering", Wai-Fah Chen, E.M. Lui

[7] University of the Witwatersrand, School of Civil and Environmental Engineering, "Skills development and employment creation through small public buildings in South Africa", Anne Susan Fitchett

[8] University of the Witwatersrand, School of Civil and Environmental Engineering, "Design & construction Criteria for domes in Low-cost housing", G. Talocchino