International Journal for Ignited Minds (IJIMIINDS)

Design Optimization of Observatory Dome Structure Using Finite Element Analysis

Sharath Kumar C S[a], Mohammed Imran[b], Dr. A R Anwar Khan[c] & Nagabhushana[d]

[a]PG Scholar, Dept. of Mechanical Engg, Ghousia College of Engineering, Ramanagara, India.
[b]Asst. Prof, Dept. of Mechanical Engg, Ghousia College of Engineering, Ramanagara, India.
[c]Principal & HOD, Dept. of Mechanical Engg, Ghousia College of Engineering, Ramanagara, India.
[d]Professor, Department of Mechanical Engineering, KSIT, Bengaluru, India.

ABSTRACT

Observatories are used for tracing the celestial objects and other space applications. Generally these are mounted on high altitudes to prevent possible obstruction from other structures and radiofrequency disturbance. The observatory domes are mainly subjected to structural weight, wind loads, snow loads and seismic loads.

Keywords *– Dome, Structure, Shell, FEA, Optimization.*

1. INTRODUCTION

Shells are the widely used construction methods for dome structures. There are no standard rules to specify the design methodology of shells. A different country follows different methodology for the usage of shells. Possibly this may due to the various environmental conditions exists in that particular countries for dome shell construction. For example hot countries, temperature effects are more effect in designing the structure along with no possibility for snow Loads. Whereas in Cold countries, thermal effects continues but the snow load is more pronounced.

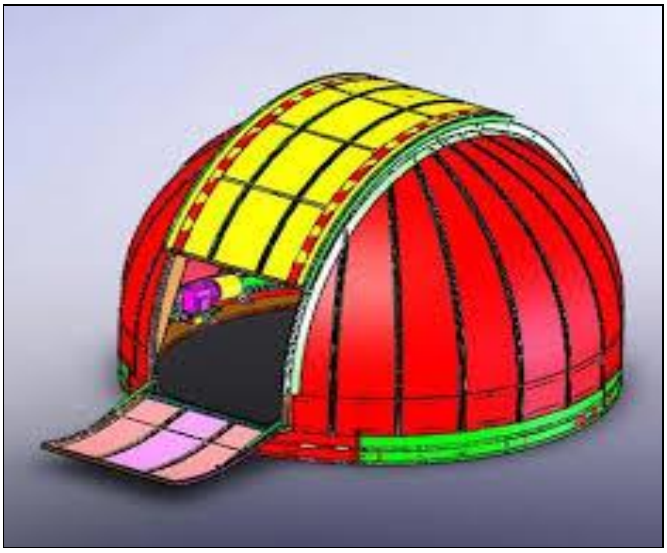

Fig 1: Dome Structure.

Observatory domes are mainly used to trace celestial objects through a telescopic arrangement provided inside. So the domes structures are mainly for protection of the inside telescopic structures. So only structural design is enough for the observatory dome structures as it is stationary and subjected to known loads.

2. LITERATURE SURVEY

Galletly et al [1], has detailed the deficiencies in the ASME code for unfired pressure vessels with torispherical heads. The code calculation gives maximum stress at the knuckle region, but the actual stress is only half the code calculation. Also the code has not specified when the yielding will takes place and possible buckling loads.

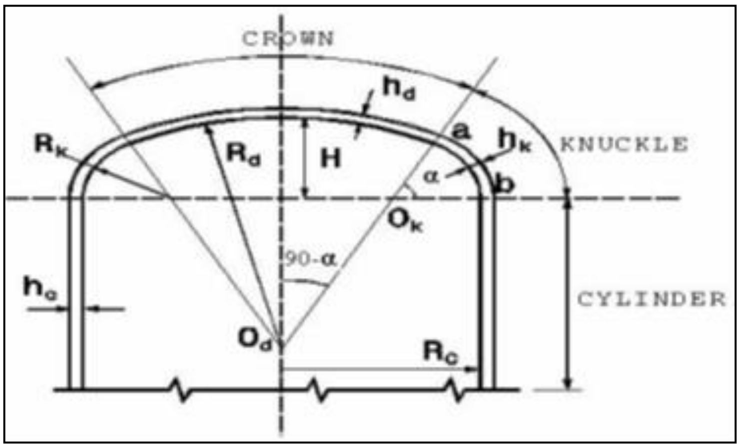

Fig 2: Dome Resembling Torispherical Heads of Pressure Vessel.

A simple formulae for finding the thickness of shell structure;

$$\sigma_d = (1.238Pr^2)/t^2 \quad (1)$$

Where, σ_d = Design stress
P = External pressure
r =Radius of the dome
t =Thickness of the Shell

3. FE MODELING

Mesh is the base of any finite element analysis, without which problem can't be executed. In this project two types of elements are used for meshing of the structure. 2D Shell elements is used for the dome structure and one dimensional beam188 elements are used for ribs of the structure which adds strength to the structure with minimum weight.

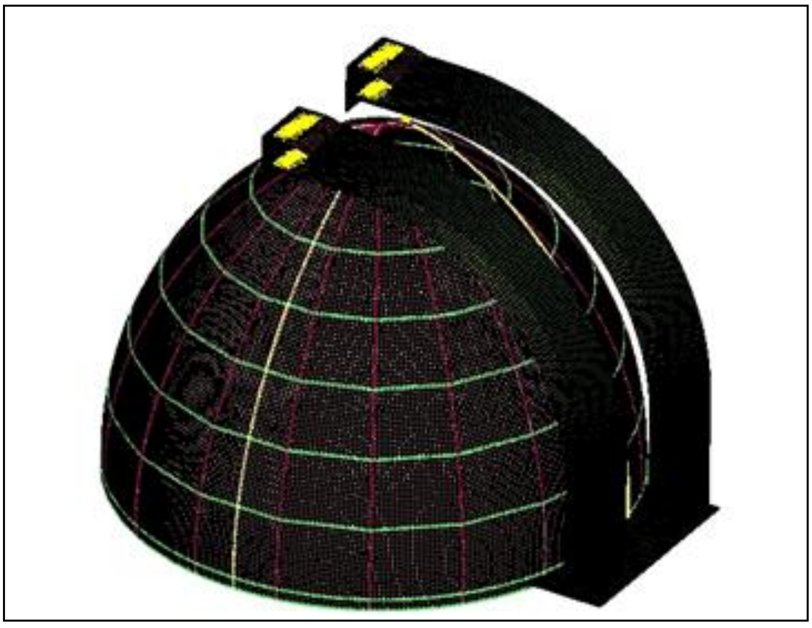

Fig 3: Finite Element Representation of the Dome Structure.

The Fig 3, shows finite element representation of the dome structure. Using 2D shell elements for any type shell geometry is the easiest method of FE method due to its 2 dimensional representation of the geometry which uses fever no of elements that helps in reducing the total DOF in the model which interns help in reducing the stiffness matrix.

4. RESULTS AND DISCUSSIONS

The observatory dome structure is analyzed for different load cases. Initially structure is analyzed for self-weight and other structural loads. The results were represented for stress and deformation for the assembly along with the stresses on individual components. Finally structure is optimized for weight until optimized structure is obtained. The results are represented with necessary pictorial views. To shortcut the results, only initial and final iterations are discussed in this paper.

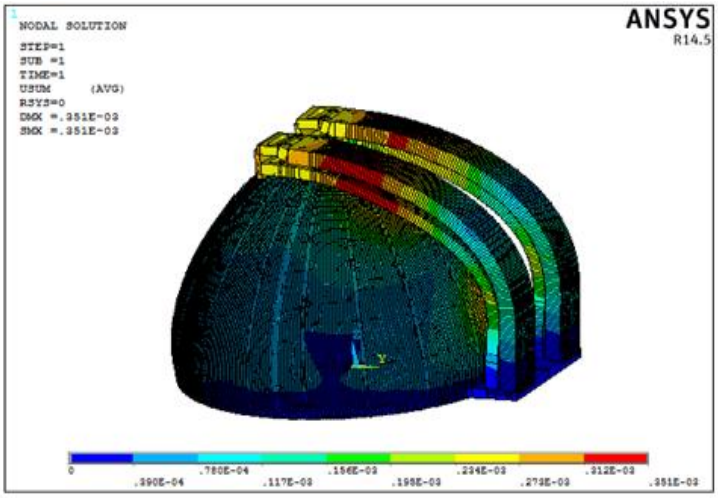

Fig 4: Deformation plot of the Problem.

The Fig 4, shows the maximum deformation observed in the initial model. The maximum deformation induced in the model is around 0.35mm (0.00035m) as shown in the above figure and the maximum deformation is observed at the door structure.

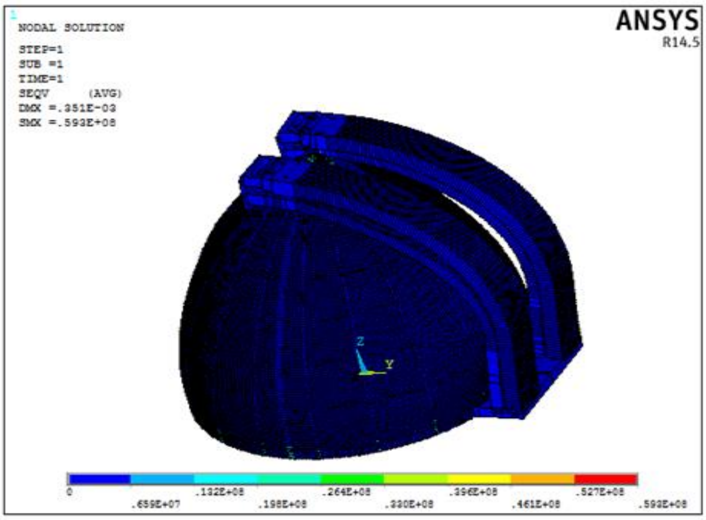

Fig 5: von-Mises Stress for the Complete Structure.

The Fig 5, shows complete von-Mises stress in the structure. The maximum stress observed in the in the dome structure is around 59.2MPa which is less than the allowable stress of the material, hence, design can be optimized for the current design. From the above figure, it is clear that the von-Mises stresses induced are very less over the entire model apart from the high stress region, this shows that the model is over design or the current design has the ability to take higher loads.

After design iterations, the final set results are presented below for the structural stability based on stress and deformation.

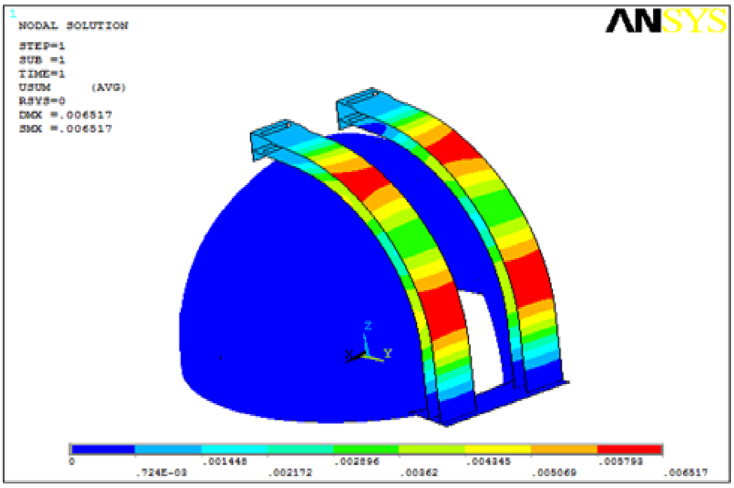

Fig 6: Deformation in the Dome Structure.

The Fig 6, shows deformation induced in the optimized structure. The maximum deformation (0.0065mm) induced in optimized structure is less than the critical deformation value of 6.57mm. So the structure is sufficiently stiffer for the given loading conditions and can withstand the given loads probably may take much higher loads.

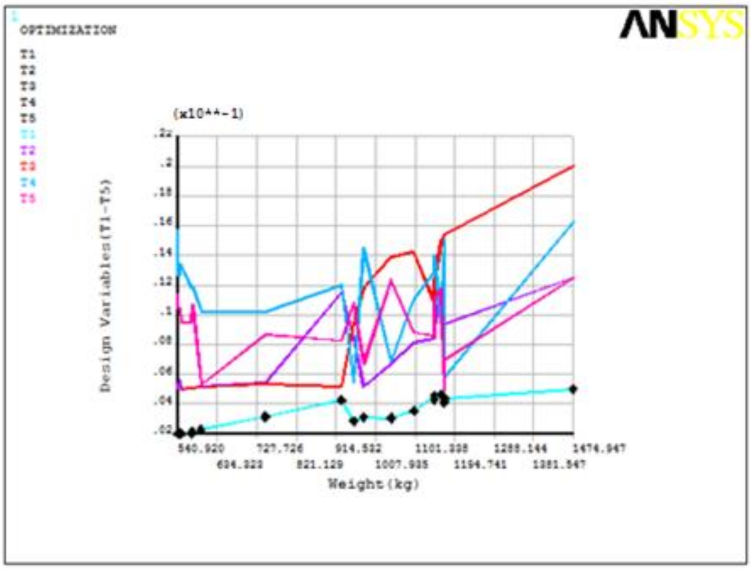

Fig 7: Weight to Design Variables 1-5 Variation.

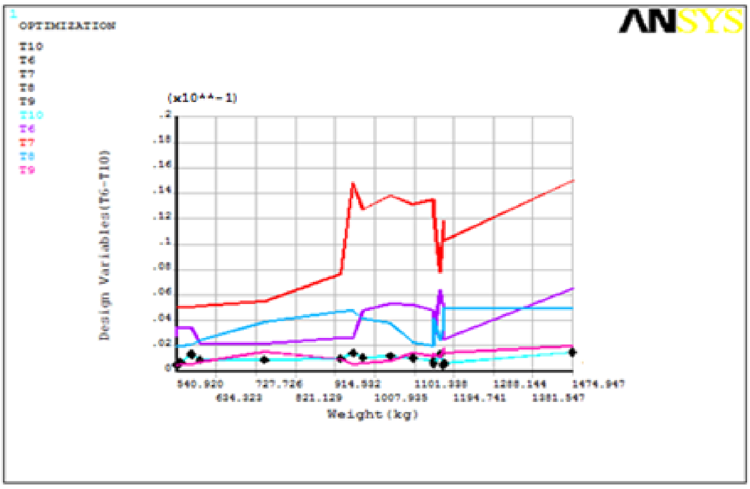

Fig 8: Weight to Design Variables 6-10 Variation.

From the Fig 7&8, shows change of thickness of design variables in the weight convergence. These graphs help the designer to find which design parameter is influencing the weight of the structure. Once the component is identified, it is easy for the designer to select the suitable thickness for the member.

5. CONCLUSION

From the detailed FE analysis, we have observed the fallowing points,

With the use of FE analysis software we can predict the exact component which influence the weight of the structure and which component's thickness can be reduced.

Which components has maximum and minimum stresses.

We could able to succeed in this project, by predicting and optimizing the right design for the dome structure.

REFERENCES

1. Galletly G D; "*Design Equations for Preventing Buckling in Fabricated Torispherical Shells Subjected to Internal Pressure*". Proc I Mech E, London 1986.
2. Shield, RT and Drucker DC, "*Design of Thin Walled Torispherical and Toriconical Pressure Vessel Heads*", ASME Int. J. Appl. Mechanics 28 292-29, 1992.
3. Ganapathy, V.R. and R.S.Charles., "*Analysis of Torispherical Pressure Vessel*" Transaction of the ASCE, Journal of the Engineering Mechanics Division, 102,643-657.1994.
4. Batchelor MJ, Taylor TE (1995) Equivalent torispherical pressure vessel heads. Int J Press Vessel Pip 7(3):229–244.
5. Hurshanarayana, "*Analysis of Torispherical Shell with Radial Nozzle*", The Institution of Engineers, Malaysia, Vol. 67, 1998.
6. Joe Metrisin, "*Stress Analysis of Thin-Walled Pressure Vessels*", Modern Mechanical Engineering, 5, 1-9 Published Online February 2015.
7. Arora, Jasbir S, Introduction to Optimum Design, 2nd edition, Elsevier Academic Press, Amsterdam, 2004.
8. Ravindran, A., K.M. Ragsdell, and G.V. Reklaitis, Engineering Optimization: Methods and Applications, 2nd edition, John Wiley & Sons, Hoboken, New Jersey, 2014.