Stress Analysis of RCC Intz Type Elevated Storage Reservoir with Discrete Modelling by Using ANSYS

Avinash M. Thorat
Lecturer
Department of Civil Engineering
Karmayogi Engineering & Polytechnic College, Shelve, Pandharpur, Solapur, Maharashtra, India

Abstract— the present study focuses on the nonlinear analysis of reinforced concrete Intz type elevated storage reservoir (ESR). In this study component wise static analysis with tank full condition is carried out. Means the components of the RCC Intz type ESR like top dome, top ring beam, cylindrical tank wall, bottom ring beam, conical dome slab bottom spherical dome, bottom circular girder, supporting frame with columns and braces and footings are modelled and static analysis by considering all the design loads is carried out in order to find maximum stresses produced in the tank components. Concrete is modelled using SOLID65 element and reinforcement is modelled using LINK180 element. Also discrete type of modelling technique is used in order to model rebar in the concrete. For modelling and analysis Ansys Mechanical apdl software is used.

Keywords: Elevated Storage Reservoir, Reinforced Concrete, Discrete Type Reinforcement Modelling, SOLID65, LINK180, Static Analysis, ANSYS Mechanical APDL

I. INTRODUCTION

Water is becoming unavailable for most of the farmers in India. As the only source of water is rain for the most of the places in India, it is necessary to store the large amount of water as much as possible. Water is an elixir of life.

Liquid storage tanks are used extensively by municipalities and industries for storing water, inflammable liquids and other chemicals. Water tanks are very important for public utility and for industrial structure having basic purpose of to secure constant water supply at the longer distance with sufficient static head to the desired destination under the effect of gravitational force. The height of the elevated tank depends on the area to be covered for the water supply. Wider the area to be served higher will be the required elevation of the tank.

In this study nonlinear analysis of RCC Intz type ESR is carried out. The design problem is taken from a standard book and as per the design procedure the components of RCC water tank like top dome, top ring beam, cylindrical tank wall, bottom ring beam, conical dome slab, bottom spherical dome, bottom circular girder, supporting frame with columns and braces and footings are modelled and analysed by considering all the design loads. Analysis is carried out by using finite element method based computer software ANSYS mechanical apdl as in [6]. In the present study SOLID65 and LINK180 elements are used to model the concrete and rebar respectively. By analysing all the component parts of the tank the maximum stresses are found out and also respective stress patterns are shown.

II. MODELLING USING ANSYS

A. Elements Used for Modeling

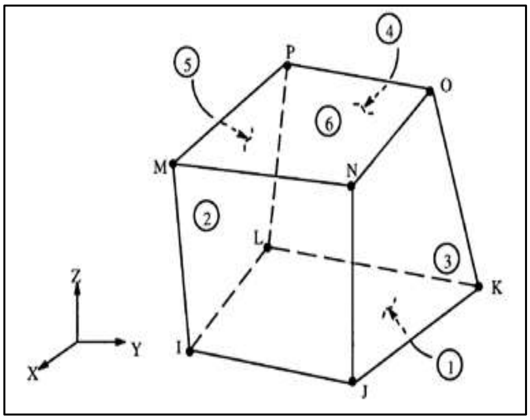

Fig. 1: Solid65 element

Element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal *x*-, *y*-, and *z*-directions. SOLID65 is used to model 3-D reinforced concrete solid elements and has the capacity of cracking in tension and crushing in compression.

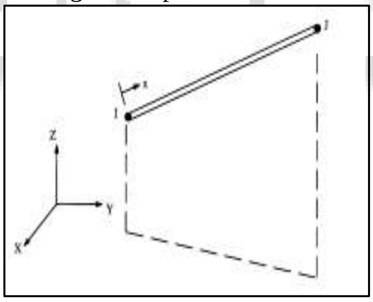

Fig. 2: Link180 element

Fig.2 shows LINK180 element which is a spar element can be used to rebar, model trusses, links etc. This 3D element is a uniaxial tension-compression element with three degree of freedom at each node: translations in the nodal x, y, and z directions.

B. Material Properties

In the present study SOLID65 element is used for modeling of concrete. For concrete linear isometric properties like elastic modulus, poisons ratio. The compressive uniaxial stress-strain relationship for the concrete model was obtained using following equations as in [8],

$$f_c = \frac{E_c\,\varepsilon}{1+\left(\frac{\varepsilon}{\varepsilon_0}\right)^2} \quad (1)$$

$$\varepsilon_0 = 2\frac{f_c'}{E_c} \quad (2)$$

$$E_c = \frac{f}{\varepsilon} \quad (3)$$

Where

f = stress at any strain ε, psi
ε = strain at stress f
ε_0 = strain at the ultimate compressive strength $f_{c'}$

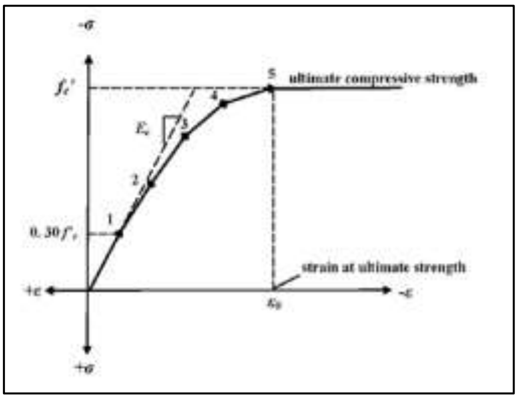

Fig. 3: Typical uniaxial compressive and tensile stress-strain curve for concrete

For steel LINK180 element is used. For steel elastic modulus, Poisson ratio and bilinear properties like yield stress, tangent modulus.

C. Reinforcement Modelling Using Discrete Modelling Technique

The reinforcement in the discrete model (Fig. 4) uses bar or beam elements that are connected to concrete mesh nodes. Therefore, the concrete and the reinforcement mesh share the same nodes and concrete occupies the same regions as the reinforcement.

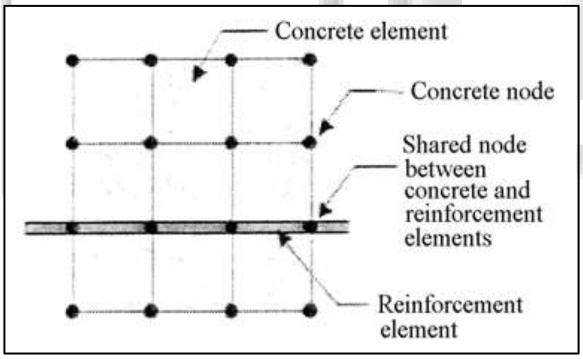

Fig. 4: Discrete modelling of reinforcement

III. ANALYSIS AND RESULTS USING ANSYS

An intz type water tank of 1 million litres capacity, supported on an elevated tower comprising of 8 columns. The base of tank is 16m above the ground level. The depth of foundation is 1m below the ground level. Grade of concrete M20 and grade of steel is Fe 415 are adopted. As per the problem designed in the standard book, all the structural elements of the intz water tank are modeled and analysed in the Ansys mechanical apdl finite element based software.

As the tank is having the basic shape that is circular, the symmetry is applied along the z-direction. So half models of all the structural members are modeled and analysis is carried out.

Extracted data from the problem:
Capacity of the tank = 1 million litres = $1000m^3$
Height of the supporting tower = 16m
Number of columns = 8
Depth of foundation = 1m below the ground level
Material grades =
Concrete = M20, Steel = Fe415

Maximum stresses are within the permissible limit as per IS 456-2000. Flexural strength,$f_r = 0.7\sqrt{f_{ck}}$. This is equal to 3.1305 N/mm2.

A. TOP DOME

Tank domes are very complex types of structures for modeling by ansys mechanical apdl. Such type of model are modeled by extrusion method. Top dome in the intz tanks are used for covering the tank from top so that stored water cannot become dirty.

Thickness of the top dome = 300mm and it is tapered to 100mm, diameter of the dome = 12m, central rise of the tank = 1/6 X 12 = 2m.

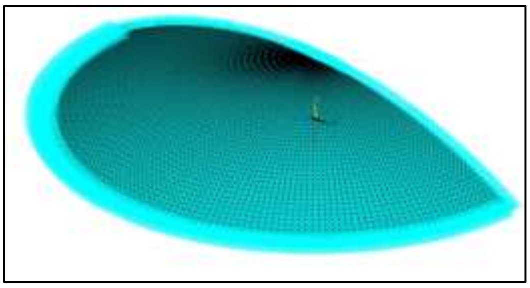

Fig. 5: Top dome symmetric along Z direction with fixed boundaries

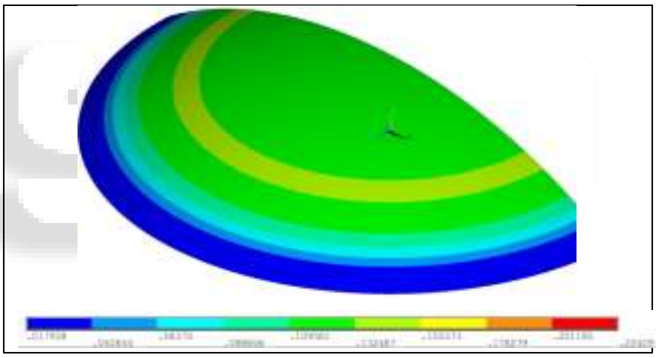

Fig. 6: maximum stress = 0.224 < 3.1305 N/mm2

As the dome is placed at the top of the tank, only self-weight is acting on the dome. Here in this analysis dome is supported by the top ring beam, therefore dome is fixed at its base.

B. Top Ring Beam

The horizontal component of the meridional thrust at the base will induce an outward push of the cylindrical wall carrying the dome. In order to prevent this, a ring beam is provided at the base of the dome.

In this work, the top ring beam is of 300mm X 300mm size provided. This beam is modelled and symmetry along z- direction is provided. This ring beam is subjected to self-weight of the top dome created a meridional thrust, self-weight of the beam and the analysis is carried out.

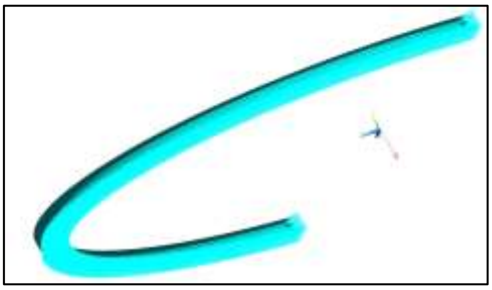

Fig. 7: Top Ring Beam with Fixed Boundary Conditions

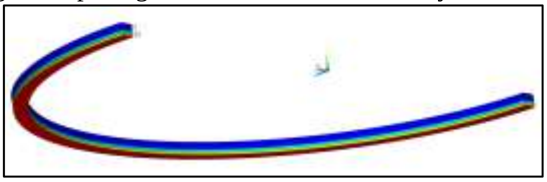

Fig. 8: maximum stress = 0.002897< 3.1305 N/mm2

Here the stresses are developed in the beams due to the horizontal component of the meridional thrust offered by the top dome.

C. Cylindrical Tank Wall

The cylindrical tank walls are designed for maximum water pressure applied on the sides of the tank. As the water pressure will be maximum at the base of the wall, therefore the cantilevering action will be took place at one fourth of the total height of the tank wall. Therefore the tank wall should be designed for the hoop tension caused by the horizontal water pressure.

In this analysis, the tank wall having diameter = 12m with wall thickness = 300mm and tank height = 8m is modeled which is symmetrical along z-direction.

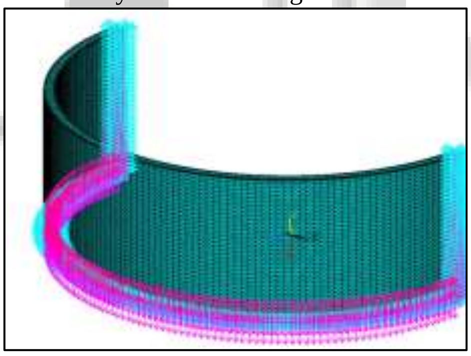

Fig. 9: Cylindrical Tank Wall with Fixed Boundary Conditions

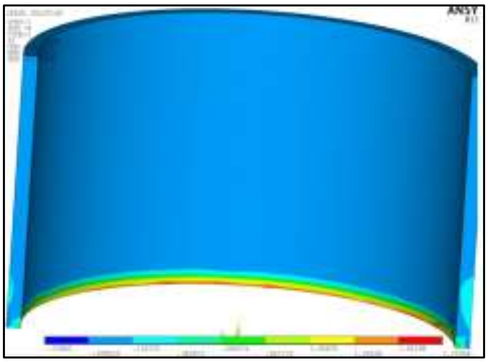

Fig. 10: Maximum Stress = 1.753< 3.1305 N/mm2

D. Bottom ring beam

In this analysis, the ring of the size 1200mm X 600mm is modeled. As this ring beam is provided at the junction of the cylindrical tank wall and conical slab, the weight of the wall and the reaction of the conical slab pushes it outward. Due to this horizontal component of the reaction is produced. Therefore, in the ansys model of the beam, the beam is analyzed for the self-weight of the tank wall, self-weight of the beam and where there is a conical slab, beam is fixed.

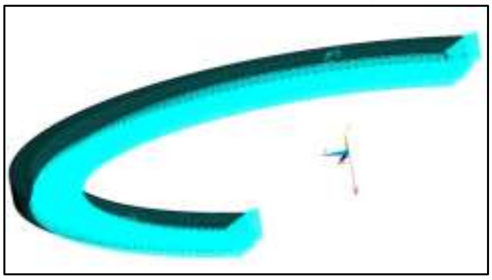

Fig. 11: Bottom Ring Beam Fixed At Bottom

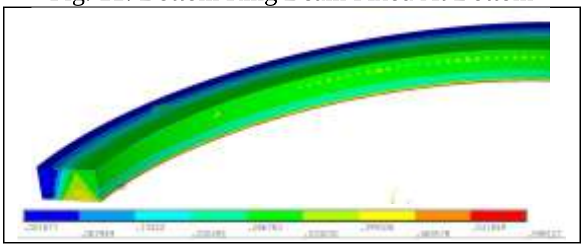

Fig. 12: Maximum Stress = 0.598 < 3.1305 N/mm2

E. Conical Dome Slab

Conical dome will be designed for a hoop tension caused by water pressure. This slab will also be designed for a possibility of its spanning between the ring beam at its top and the ring girder at its bottom.

In the present study, the model of conical dome slab is constructed in the ansys software by revolving it about the y axis. The beam is made symmetrical along z direction. The conical dome slab is subjected to water pressure acting on its surface, total coming from the top dome, top ring beam, cylindrical wall and the self-weight of the slab.

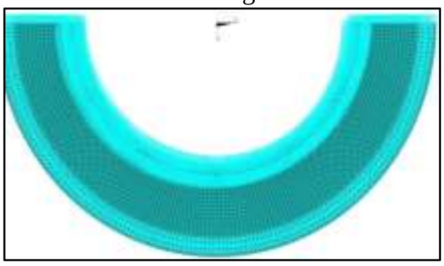

Fig. 13: Conical Dome Slab Fixed At Bottom

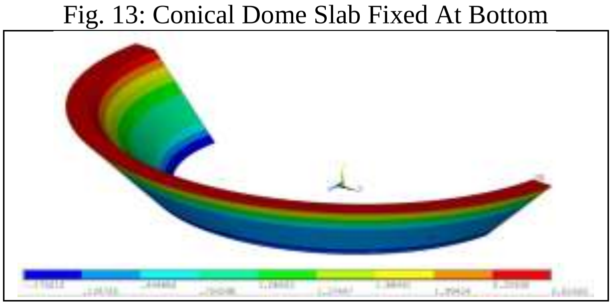

Fig. 14: Maximum Stress = 2.614<3.1305 N/mm2

F. Bottom Spherical Dome

The design of the floor dome is similar to that of the top dome. The design load for the dome includes the self-weight of the dome and weight of water column above the dome. The dome reinforcements are designed for meridional thrust and circumferential forces.

The floor dome is supported by the bottom circular girder, so the dome is fixed at its base as in Fig. 15.

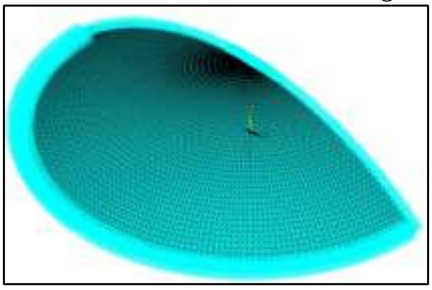

Fig. 15: Bottom Spherical Dome

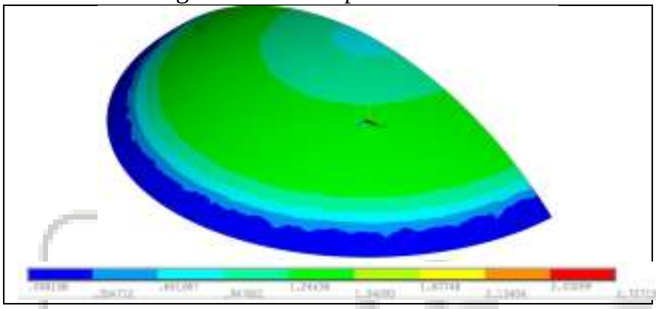

Fig. 16: Maximum Stress = 2.727 < 3.1305 N/mm2

G. Bottom Circular Girder

In this analysis the beam girder of 600mm X 1200mm is modeled. It is supported by the 8 number of columns. Here as symmetry is applied along the z-direction, only at four columns the beam is fixed and analysis is carried out.

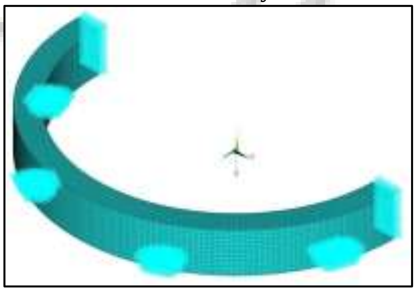

Fig. 17: Bottom Circular Girder fixed where columns are present

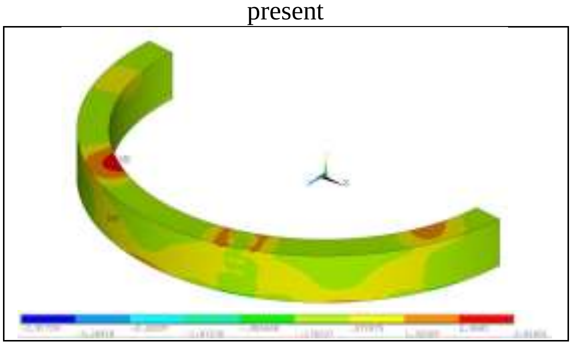

Fig. 18: Maximum Stress = 2.816<3.1305 N/mm2

H. Supporting Frame with Columns And Braces

The supporting tower comprises of 8 columns equally spaced on a circle of 8m diameter. The tower consists of columns braced together at regular intervals. In this columns and braces are modelled together and analysed. Most of the frames are failed at joints therefore in this we can see the critical location of the structure. Column is having 650mm diameter with 16m height and brace is having 300mm X 300mm size with 3.06m length.

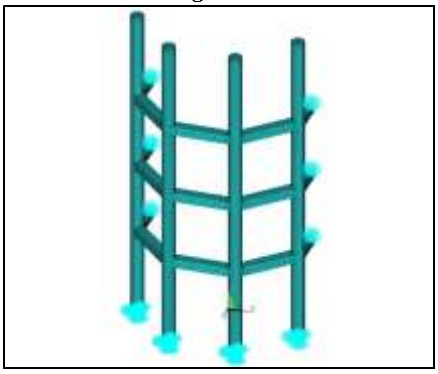

Fig. 19: Supporting Frame with Columns and Braces

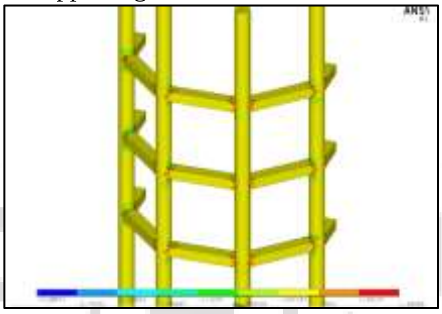

Fig. 20: Maximum Stress = 1.582<3.1305 N/mm2

I. Foundation

The foundations for a circular group of columns generally comprises of a ring beam with raft slab. The ring beam is designed for maximum bending and torsional moment while the annular raft slab is designed for maximum soil pressure from the bottom.

The foundation model is modelled and it is subjected to all the loads coming from all the structural members. Analysis is carried out by fixed base condition.

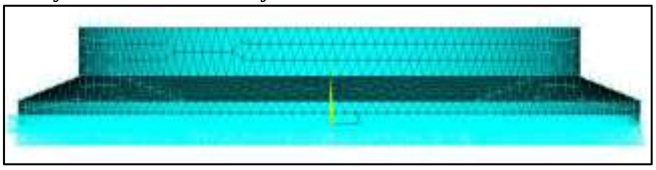

Fig. 21: Foundation Fixed At Base

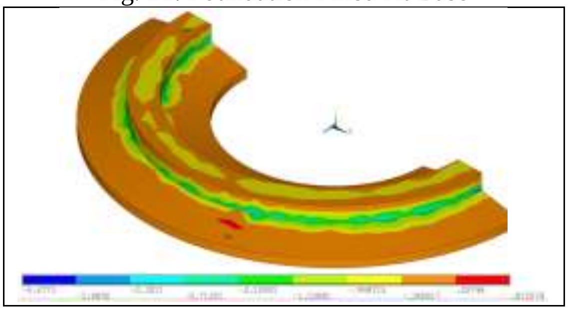

Fig. 22: Maximum Stress = 0.815<3.1305 N/mm2

IV. CONCLUSIONS

This study focuses on the investigation of the performance of nonlinear finite element analysis of reinforced concrete structures using Ansys concrete model. Due to multilinear properties of compressive uniaxial stress-strain relationship for the concrete model are applied, models are behaving as original RCC concretes. All the stresses are within the permissible limit. This indicates the models are not failed. During the analysis, care should be taken that the convergence criteria must be applied properly.

The results obtained are very good and the Ansys concrete models have given satisfactory response to the applied material properties.

ACKNOWLEDGMENT

I want to thank principal and head of the civil engineering department of Karmayogi Engineering & Polytechnic College, Shelve, and Pandharpur for providing the Ansys software and computer lab for working on the present study.

REFERENCES

[1] S. Ramamrutham and R. Narayan, *Design of reinforced concrete structures, 7th Ed,* Dhanpat rai publishing company.
[2] N. Krishna Raju, *Advanced reinforced concrete design (IS-456-2000)*, 2nd Ed, CBS publishers.
[3] Dona Rose K. J., Shreekumar M., Anumod A S, " *A study of overhead water tanks subjected to dynamic loads",* International Journal of Engineering Trends and Technology (IJETT) – Vol. 28, no. 7,oct. 2015.
[4] V. B. Dawari, G. R. Vesmawala," *Application of Nonlinear Concrete Model for Finite Element Analysis of Reinforced Concrete Beams",* International Journal of Scientific & Engineering Research, Vol. 5, Issue 9, Sept. 2014.
[5] L. Dahmani, A.Khennane, and S.Kaci, "*Crack Identification in Rein-forced Concrete Beams using ANSYS Software*," Strength of Materials, Vol. 42, No. 2, pp. 232–240, 2010.
[6] ANSYS Mechanical APDL Verification Manual, Release 15.0 Nov 2013.
[7] Sonali M. Maidankar, G.D. Dhawale, S.G. Makarande,"*Seismic Analysis of Elevated Circular Water Tank using various Bracing Systems",* International journal of advanced engineering research and science, Vol-2, Issue-1, Jan. - 2015.
[8] *A.* J. Wolanski, "*Flexural behaviour of reinforced and prestressed concrete beams using finite element analysis",* M. Sc. Thesis, Marquette University, Milwaukee, Wiscnsin, may 2004.

