Journal of Advances and Scholarly Researches in Allied Education
Vol. XV, Issue No. 2, (Special Issue) April-2018, ISSN 2230-7540

Numerical Study of Gravity Dam with Gallery Study under Influence of Sloshing Wave Using ANSYS.16

Shyamal Pise[1]* Prof. G. R. Patil[2]

[1] M.E Structural Engineering 2nd Year Student, JSPM's Rajarshri Shahu college of Engineering, Pune

[2] Savitribai Phule Pune University, Pune

Abstract – Drainage gallery is very critical in case of gravity dam. Numerical three-dimensional FEA flow analysis is a very efficient method for the stress analysis of the subsurface drainage systems of concrete gravity dams. The gravity method of the analysis does not consider the size of opening and the elastic property of dam material. Thus the objective of study is comprises of the Finite Element Method which considers the size of opening, elastic property of material, and stress distribution because of geometric discontinuity in cross section of dam. The pressure due to wave also plays important role on dam wall. This paper deals with the study of stresses analysis of koyna dam of 103.00m high above ground level with varying thickness in accordance with IS 6512. This dam has been analyzed for different types of load using ANSYS software by assuming fixity at the base. The analysis of dam structure has been carried out using 8 noded SHELL 181 elements with uniform SHELL thicknesses

Keyword: - Drainage Gallery, Gravity Dam, Finite Element Analysis, ANSYS.16

---------------------------------x---------------------------------

1. INTRODUCTION

It has been observed that large openings in the dam induce higher stresses in the vicinity of them. In some cases, these openings have contributed increase in stresses. The variation of stress according to different shapes is much more. A system of openings is provided within the body of gravity dams to serve various purposes (Shirkande & Dawari, 2011). The layout, size and shape of the openings are based on their requirements in dam. Openings develop zone of tensile stresses since concrete is not designed to take up any tension hence it become necessary to reanalyze the dam section with openings. In this project the effect of size variation and shape variation of large galleries on dam is studied. Finite element analysis of the dam with large opening is necessary to determine the general stress field, as large openings (are addition more than 1.5m x 2.25m) are not considered in the IS: 12966: 1990. Therefore, in this paper, analysis of concrete gravity dam for different shapes & sizes of openings is presented. The analysis is done using ANSYS.16as analysis tool which based upon Finite Element Method.

OBJECTIVE OF STUDY

- To study dynamic response of gravity dam due to wave pressure by developing FEM of gravity dam using Ansys.
- To do study design parameters such as Toe, Heel, Stem for hydrodynamic wave pressure.
- To find principal tensile stress location in gravity dam for future tensile crack production.
- To check effect of opening of gravity dam subjected to wave pressure at different height.
- To check effect on stresses in dam after changing toe angle.

LOADING ON GRAVITY DAM IN ACCORDANCE WITH IS 6512 :

- The following loads shall be considered:

- Dead load.
- Reservoir and tail water loads,
- Uplift pressure,
- Earthquake forces,
- Earth and silt pressures,
- Ice pressure,
- Wind pressure,
- Wave pressure, and
- Thermal loads.

2. SYSTEM DEVELOPMENT

Gravity dam is generally considered as a solid plain strain structure. Its thickness is much greater than its other two dimensions; therefore it has been analyzed as 2D plain strain structure. But in this work it is analyzed as three dimensional problem Gravity dam is subjected to various forces like hydrostatic pressure, uplift pressure etc. due to which it causes stress concentration within its body. Such stress concentration leads to localized failure zones in the structure. Though the stress concentration is to be localized can leads to drastic damage to important structure like dam. The dam structure failure is analyzed using tools like Finite Element Method & ANSYS 16

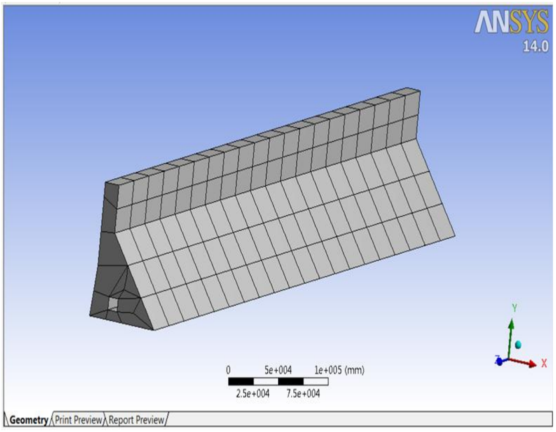
2.1 Finite Element Modelling

The following elements of ANSYS.16are used for validation and modeling of gravity dam.

SOLID186 Element Description:

SOLID186 is a higher order 3-D 20-node solid element that exhibits quadratic displacement behaviour [21].

Fig.1 Finite Element Meshing of gravity dam.



The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions. The element supports plasticity,

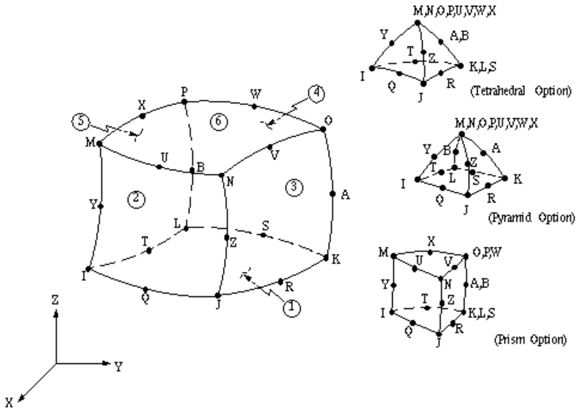


Fig.2 Solid 186 geometrical representations

SURF 151: SURF 151 may be used for various load and surface effect applications. It may be overlaid onto a face of any 2-D thermal solid element (except axisymmetric harmonic elements PLANE75 and PLANE78). The element is applicable to two-dimensional thermal analyses. Various loads and surface effects may exist simultaneously. The element is defined by two to four node points and the material properties. An extra node (away from the base element) may be used for convection or radiation effects. The element x-axis is along to the I-J line of the element also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials.

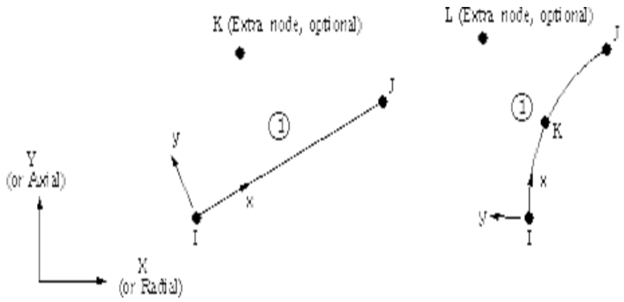


Fig.3 SURF 151

CONTA174: CONTA174 may be used to represent contact and sliding between two surfaces (or between a node and a surface, or between a line and a surface) in 2-D or 3-D. The element is applicable to 2-D or 3-D structural and coupled field contact analyses. This element is located on the surfaces of solid, beam, and shell elements. 3-D solid and shell elements with mid side nodes are supported for bonded and no separation contact. For other contact types, lower order solid and shell elements are recommended. Contact occurs when the element surface penetrates one of the target segment elements (TARGE169, TARGE170) on a specified target surface. Coulomb friction, shear

www.ignited.in

stress friction, and user defined friction with the USERFRIC subroutine are allowed. This element also allows separation of bonded contact to simulate interface delamination.

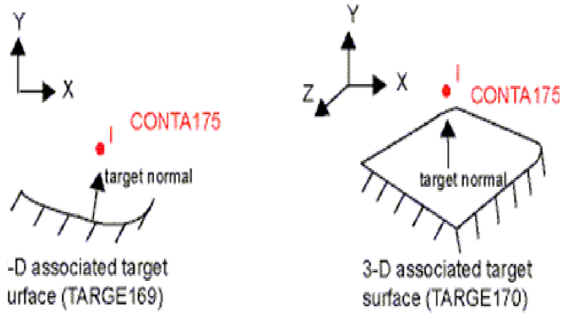


Fig.4 Geometrical representation of CONTA175

TARGET170: TARGE170 is used to represent various 3-D "target" surfaces for the associated contact elements). The contact elements themselves overlay the solid elements describing the boundary of a deformable body and are potentially in contact with the target surface, defined by TARGE170. This target surface is discretized by a set of target segment elements (TARGE170) and is paired with its associated contact surface via a shared real constant set. You can impose any translational or rotational displacement on the target segment element. You can also impose forces and moments on target elements. See Section 14.170 of the ANSYS Theory Reference for more details about this element. To represent 2-D target surfaces, use TARGE169, a 2-D target segment element.

For rigid target surfaces, these elements can easily model complex target shapes. For flexible targets, these elements will overlay the solid elements describing the boundary of the deformable target body.

3. PROBLEM STATEMENT

Koyna concrete dam (Fig.1) In Maharashtra, India, has been chosen for the finite element modeling only in case 1 (Dam with empty reservoir and rigid foundation). The dam is 103m high; width of the dam base is 70m and crest width is 14.8mThe material properties of the dam are modulus of elasticity, mass density and Poisson's ratio which 31027MPa, 2354 kg/m and 0.2, respectively. The empty reservoir and rigid foundation) geometry variables of dam are given in Table. The natural frequencies for Cases 1-4 from the finite The first four natural frequencies of dam from the element model and the literature are given in Table 3 finite element model] and It can be observed that a good conformity has been Table 5 .achieved between the results of present work with those of reported in the literature .Also, the very small percentage error showed excellent accuracy of the proposed model for dam-reservoir-foundation system.

Parameter	Value (m)
b	14.80
b1	0.00
b2	1.3713
b3	1.45987
b4	1.61837
b5	50.75
H1	11.25
H2	11.975
H3	52.75
H4	39.0
H5	66.50

Table 1 Parameters of Koyna dam

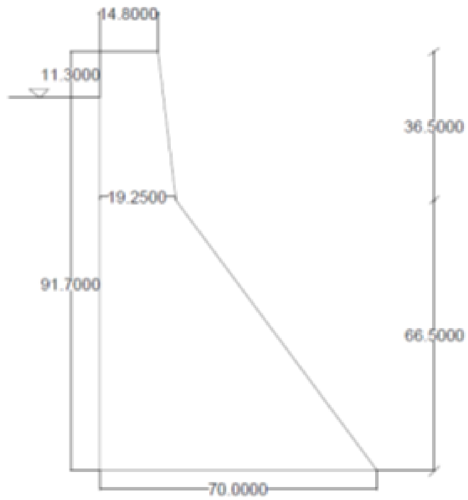


Figure 5.Typical cross-section of Koyna dam

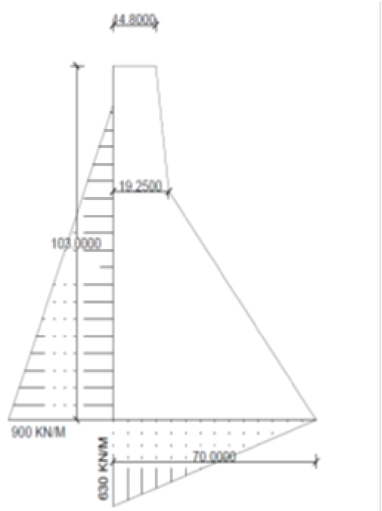


Figure 6.Pressure diagram of Koyna dam.

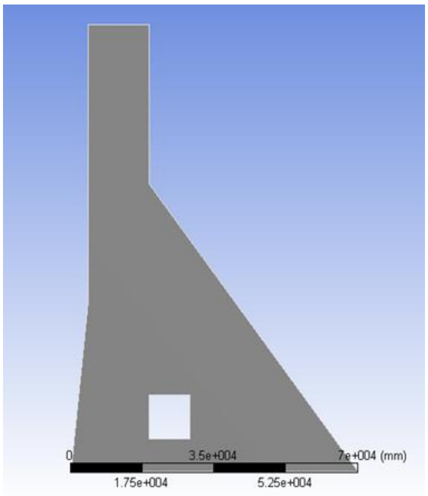


Fig.6 Koyna dam with rectangular opening

4. MATERIAL PROPERTIES

Following are the material properties which are used in modeling

am Body	Modulus of Elasticity	22400MPa
Water	Mass density of water	1000kgm^{-3}
	Speed of pressure wave	1400ms^{-1}
	Wave reflection coefficient	0.817
Foundation Rock	Elastic modulus of foundation rock	68.923Mpa
	Poisson's ratio of foundation rock	0.33
	Mass density of foundation rock	0.00

Table 2: Material Properties of Koyna dam

5. RESULT ANALYSIS

After validation of model, different shape and different locations are used. Empty reservoir condition and full reservoir condition is studied under different parameters .The results are listed in table.01

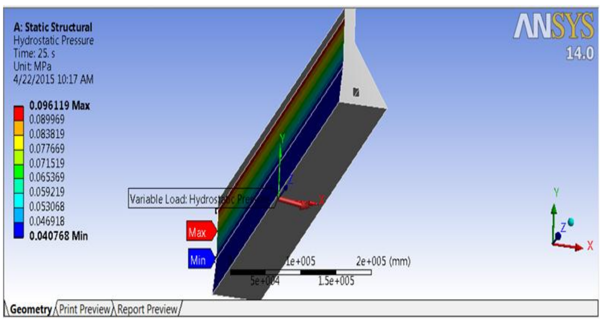


Fig.7 Pressure Dia. In ANSYS

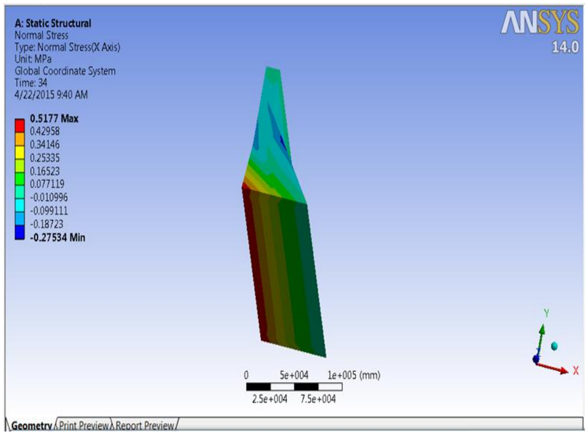


Fig.8 Normal Stress in Koyna dam

Opening Type	DAM WITHOUT OPENING	
Condition	Empty reservoir	Full reservoir
NORMAL STRESS (MPa)	0.42718	0.57
SHEAR STRESS (MPa)	0.21853	0.379
DEFRMATION X (mm)	6.0755	12.67
DEFRMATION Y (mm)	0.53233	0.6004
Opening Type	DAM WITH OPENING (rectangular)	
Condition	Empty reservoir	Full reservoir
NORMAL STRESS (MPa)	0.50627	0.48735
SHEAR STRESS (MPa)	0.66839	0.58881
DEFRMATION X (mm)	6.1224	12.37
DEFRMATION Y (mm)	0.54223	0.61057
Opening Type	DAM WITH OPENING(circular)	
Condition	Empty reservoir	Full reservoir
NORMAL STRESS (MPa)	0.60268	0.60971
SHEAR STRESS (MPa)	0.60307	0.75226
DEFRMATION X (mm)	12.49	12.506
DEFRMATION Y (mm)	0.62011	0.62294

Table No.3 Stress analysis and deflection

6. CONCLUSION

The first step of simulation is aimed to validate the models developed by comparing them to previous work [20]. By finite element Analysis 16, a three dimensional finite element model of Koyna gravity dam is proposed using ANSYS 16. Dams with full reservoir and empty reservoir condition are analyzed. In this study, normal stress shear stress and deflection along X and Y direction is observed.

The rectangular opening of 4×4 m is more effective than circular opening of same size. The stresses and deformation are less in case of rectangular opening and to reduce the deflection rectangular opening should be at bottom near hill of gravity dam.

7. REFRENCES

A. S. Shirkande, V. B. Dawari (2011). "3 D Stress Analysis around Large Openings in Concrete Gravity Dam", *International Journal of Earth Sciences and Engineering ISSN 0974-5904*, Volume 04, No 06 SPL, October 2011, pp. 600-603.

Ansys 16Manual documentation. ANSYS, INC.

C. S. Krishnamoorty (2010). Finite Element Analysis, *2nd edition, McGraw Hill*, New Dehli, 2010.

IS 12966 (Part 1): (1992). "Code Of Practice For Galleries And Other Openings in Dam", UDC 6227-824-7.

IS 12966 (Part 2): (1990) "Code Of Practice For Galleries And Other Openings In Dams", UDC 627-8-068-624-04.

IS: 10135-1985, "Code of Practice for Drainage System for Gravity Dams, Their Foundations and Abutment", UDC 627-824-7-034-96-006-7.

IS: 6512-1984, "Criteria For Design of Solid Gravity Dams", *First Reprint SEPTEMBER 1998*, UDC 627.824.7.04.

Jeong Kim a, Joo-Cheol Yoon, Beom-Soo Kang (1972). "Finite Element Analysis And Modeling Of Structure With Bolted Joints", *Trans. ASME J. Mech*. Des. 94 (1972) pp. 864–870.

K.M.Pandey and Amrit Sarkar (2011). "Structural Analysis of Nuclear Fuel Element with ANSYS Software", *IACSIT International Journal of Engineering and Technology*, Vol.3, No.2, April 2011.

K.R. Dhawan, D.N. Singh, I.D. Gupta (2009). "2D and 3D finite element analysis of underground openings in an inhomogeneous rock mass, a Central Water and Power Research Station, Khadakwasla, 2009"unpublished.

Laxmikant D. Rangari, Mrs. P.M. Zode, P.G. Mehar (2012). "Stress Analysis Of Lpg Cylinder Using Ansys Software", *International Journal of Engineering Research and Applications (IJERA*) Vol. 2, Issue4, July-August 2012, pp.2278-2281.

Madenci E, Guven I. (2006). "The Finite Element Method and Applications in Engineering", *ISBN: 978-0-387-28289-3*, www.Springer.com, 2006.

Md. Hazrat Ali, Md. Rabiul Alam, Md. Naimul Haque, Muhammad Jahangir Alam (2012). "Comparison of Design and Analysis of Concrete Gravity Dam", *Natural Resources, 2012*, 3, 18-28 doi:10.4236/nr.2012.31004 Published Online March 2012.

Mohamed Abd El-Razek and Magdy M. Abo Elela (2001). "Optimal Position of Drainage Gallery underneath Gravity Dam", *Sixth International Water Technology Conference, IWTC 2001*, Alexandria, Egypt.

Ray W. Clough, Edward L. Wilson (1991). "Early Finite Element Research at Berkeley", *Finite Elements in Analysis and Design 7*, pp. 89-101.

Recep Kanit, M. Sami Donduren (2010). "Investigation of using ANSYS software in the determination of stress behaviors of masonry wall under out of plane cycling load", *International Journal of the Physical Sciences* Vol. 5 (2), pp. 097 108, February, 2010.

S. S. Bhavikatti (2012). Finite Element Analysis, 2nd edition, *New Age International Publishers, New Dehli*, 2012.

Sharique Khan, V. M. Sharma (2011). "Stress Analysis of a Concrete Gravity Dam with Intersecting Galleries", *International Journal of Earth Sciences and Engineering ISSN 0974-5904*, Volume 04, No 06 SPL, October 2011, pp. 732-736.

Utili Stefano, Yin ZhenyuÿJiang Mingjing (2008). "Influences of Hydraulic Uplift Pressures on Stability of Gravity Dam", *Chinese Journal of Rock Mechanics andEngineering*, Vol.27 No.8, August 2008.

Vipin Gupta and Vishal Waghmare (2014). "Study of Structural Behaviour of Gravity Dam with Various Features of Gallery by FEM", *ACEE Int. J. on Civil and Environmental Engineering*, Vol. 3, No. 1, Feb 2014.

Corresponding Author

Shyamal Pise*

M.E Structural Engineering 2nd Year Student, JSPM's Rajarshri Shahu college of Engineering, Pune

E-Mail – pise.shyam33@gmail.com