ANALYSIS OF BARRAGE USING FINITE ELEMENT METHOD- AN OVERVIEW

Abhay Kohli¹, Mohit Kumar²

¹M.E. Civil Engineering Student, PEC University Of Technology, Chandigarh, India.
²Assistant Professor, Civil Engineering Department, PEC University Of Technology, Chandigarh, India.

ABSTRACT

In this review paper, the emphasis is on highlighting the versatility of the Finite Element Method (FEM) in the analysis of the hydraulic structures like barrages and discussing the merits of the Finite Element Method (FEM) over the orthodox approach that is being employed in India to analyse the raft foundation of the barrage as per the Indian standards (IS: 11130-1984). The conventional approach adopted for the analysis of raft foundation has been of Hetenyi’s, which recommends the raft floor to be designed as a beam on elastic foundation. The literature available on this aspect has been studied in detail [3 number technical papers]. In these studies the deformations and resultant moments have been computed for the different load cases using both the methods and their values have critically been compared in order to draw the comparison between the two methods. In one such study, the dynamic analysis of the barrage at different stages of the construction has also been carried out as the sporadic nature of earthquake phenomenon necessitates the need of analysing massive hydraulic structures using computational methods like FEM. This review paper critically analyses the work carried out earlier to form the basis for further study.

Keywords: Barrage, Raft Foundation, Finite element Method, Hetenyi’s Method.

1. INTRODUCTION

A barrage is a diversion head work structure which has been employed across the natural drainage to divert drainage water into an off taking canal or a tunnel. In a barrage, the crest is kept at a lower level and the majority of the water is held back with the help of gates. The barrage, being a massive structure requires a strong foundation at bed which is capable of distributing heavy load over a large area. The raft foundation is found to be most suitable for the barrage structure as it not only caters for the massive superstructure load but also eliminates the extensive dewatering problems that generally hamper the construction practices.

The barrage that was selected for the available study is 49.256 m long, 85 m wide and 25 m high; a part of Larji Hydroelectric power project in Beas basin in Kullu district, Himachal Pradesh. It consists of 5 bays of 11 m with construction joints; one between bay-2 and bay-3 and the other between bay-4 and bay-5 has been provided. The single pier has a thickness of 4 m and the double pier has 2.25 m thick. The raft thickness varied from bay-1 to bay-5, with 3 m from bay 1 to 4 and 1 m for bay 5 Pandey A.D. et al [4]. The diversion structure is partly supported by rock and partly by alluvium. The bays 3-4 of the barrage have been considered for the comparative analysis of the raft foundation of the barrage, Venkatesh et al [5]. The oggee shaped barrage raft floor of bays 3-4 has been separated by expansion joints on either side from the rest of the bays.

India is divided into four zones on the basis of seismicity. The barrage falls under Zone V as per IS: 1893-2002. An earthquake, a catastrophic phenomenon, is unpredictable in nature with respect to space and time. The hydraulic structures constructed in seismically active areas should be analysed for the dynamic load in order to avoid sporadic failure of the structure as the possibility of failure at any intermediate stage of construction can’t be ruled out. A total of fourteen stages were considered for analysis by the author, Pandey A.D. et al [4].

2. METHODS OF ANALYSIS

2.1 FINITE ELEMENT METHOD (FEM)

The Finite Element Method is a numerical procedure which gives an approximate solution to a complex differential equation problem restricted within defined boundary conditions. With the advent
of computer and the related softwares, the ease of employing finite element procedure to numerous boundary value problems has increased manifold. It has become relatively convenient to analyze complicated geometries with the Finite Element Method.

The basic steps that we follow in the Finite Element Method are discretization of the continuum / meshing the continuum; calculation of element stiffness matrix; assembling the element stiffness matrix; calculation of element load vectors; assembling of element load vectors; imposition of boundary conditions; imposition of external forces; calculation of displacement vectors; calculation of stress and strain fields. Venkatesh et al [5].

In the referred works, ANSYS has been used to simulate the forces acting on the raft foundation and to compute the stresses arising from the forces acting in the plane of continuum. This software provides the following platform to carry out the FEM analysis:

1. Generation of the FEM Model.
2. Assignment of the material properties to the geometry.
3. Meshing of the continuum as per the geometry requirements along with focus on the zones where stress concentration(s) are likely to occur.
4. Imposition of the boundary conditions.
5. Appropriate loading of the continua.

ANSYS gives output in the form of deformations and stresses at different points of the structure.

In addition to this, the site dependent Response Spectrum was used for carrying out the dynamic analysis.

2.2 HETENYI’S METHOD

In this method of beams on elastic foundation, Hetényi has brought forward the concept of semi infinite beams and beams of finite length by imposing end conditions on beams of unlimited length. Hetényi has assumed the soil medium as Winkler’s media for analysing the raft as a beam on elastic foundation and the reaction force at any point in medium is proportional to the deflection at that point. These reaction forces are acting vertically upwards opposing the deflection of the beam.

The deflection of beam supported on elastic foundation can be represented in the form of a differential equation which is given by:

$$EI \frac{d^4y}{dx^4} = -ky$$

where $EI$ – Flexural Rigidity of the beam.

$k$- Elasticity of the soil medium.

$y$- deflection of the beam at any point.

Hetényi discussed rigidity criteria in terms of $\lambda L$, where $\lambda$ is known as the characteristic of the system, which accounted for the width, length and elastic properties of the media, Venkatesh et al [5].

$$\lambda L = \sqrt[4]{\frac{K_0 BL^4}{4EI}}$$

where $K_0$- modulus of the supporting medium

$B$- width of the footing

$L$- length of the footing

$E$- modulus of elasticity of footing material

$I$- moment of inertia of footing.

3. ANALYSIS AND DISCUSSIONS

3.1 COMPARATIVE ANALYSIS

In the finite element approach, soil and rock media has been modelled by three-dimensional eight noded isoparametric brick elements and the cut-off, pier, abutment wall and beam has also been modelled by eight noded isoparametric brick elements. The modelling of the raft has been done with
the help of four noded three dimensional shell elements. The depth of soil and rock media has been considered 80m from the crest level and the soil and rock media has been considered up to 35m in transverse direction and 50m in both upstream and downstream direction. Venkatesh et al [5]. The mesh has been refined from coarse to fine until the moments at the section converged under gravity load in two models. The total number of elements adopted for finite element model was 18744 which eventually resulted in 21204 nodes. The boundary conditions imposed on the finite element model consisted of restraining the foundation media at 80m depth in vertical direction and ends along and across the direction of flow were restrained in horizontal direction. Venkatesh et al [5]. 

In the theory of beams on elastic foundation by Hetényi, the beams are categorised on the basis of the value of $\lambda L$ which are as follows:

- **Group I:** short beam, $\lambda L < \pi/4$
- **Group II:** beams of medium length, $\pi/4 < \lambda L < \pi$
- **Group III:** long beam, $\lambda L > \pi$, Venkatesh et al [5]

It has been assumed that the other end of the beam is infinitely far away and the beam has been analysed as semi infinite beam in which forces applied at one end have a negligible effect on the other end. Venkatesh et al [5].

The end conditioning forces $P_0$, $M_0$ were calculated on the basis of $A\lambda x$, $B\lambda x$, $C\lambda x$ and $D\lambda x$, Hetényi [2].

The moments and deformations for different load considerations at different sections of the raft have been calculated by method of superposition.

![Figure 1: Plan of bays 3-4, Venkatesh et al [5]](image)

Three sections were considered for the comparative analysis viz. upstream section A-A, ogee section B-B and downstream section C-C. Venkatesh et al [5].

While analysing the raft with finite element method using ANSYS, the variation of foundation media was accounted for, which was not in case of Hetenyi’s method where the entire analysis were pivoted on the characteristic of the system, $\lambda$ which only takes into account the presence of only one type of foundation media.

The moments and deformations calculated for the representative load cases of gravity load alone, gravity load-differential head and gravity load-earthquake load by the two methods varied significantly. Venkatesh et al [5].

### 3.2 DYNAMIC ANALYSIS

Three separate Finite Element (FE) models were considered for the study by Pandey et al [4] viz. model A, B and C. The bays 1&2 were selected for Model A, bays 3&4 for Model B and bay 5 for Model C. The influence of the surrounding soil has been considered up to 30m in transverse direction and 50m in both upstream and downstream directions.
A total of fourteen stages were considered for the analysis. The raft floor was the 1st stage of construction (completed in single lift), the subsequent stages following it were of 1.5m construction at each stage and the last two stages were of 4.8m and 3.7m respectively.

As per free vibration analysis carried out by the author, the model which completed rested on rock had the highest fundamental frequency at all stages of construction while the model resting on the soil media had the lowest frequency. With the addition of mass to the structure at every stage, the frequency of the structure kept on decreasing with each construction stage. However, there was an increase in frequency of the structure when a breast wall was constructed which increased the stiffness of the structure and thereby increased its frequency. ANSYS software was used for the analysis and the site dependent response spectrum, DEQ [5] was used for simulation of earthquake excitation. Pandey et al [4].

For the response spectrum analysis, the first 10 modes were considered. The stresses increased up to the 12th stage of construction and reduced thereafter owing to the additional rigidity provided by the breast wall to the structure.

4. SIGNIFICANCE OF THE STUDY

A thorough insight into these studies has provided a platform for the analysis of barrage and its substructure and has enabled the reader to figure out the pros and cons of the various methods that are being used for carrying out the analysis of barrages and other massive hydraulic structures. The methodologies to be followed have been clearly defined by the authors and with these works they have encouraged the researchers to carry out significant forward work in this field. The inception of the computational methods particularly Finite Element Method (FEM) based tools in the modern day engineering has facilitated the analysis of the complex structures along with the variation in the supporting foundation media more precisely and has helped to locate the areas which require particular attention from the safety point of view of the structure and the foundation together.

5. CONCLUSION

After the review of the aforementioned studies, it can be safely concluded that the Finite Element Method and other computational methods facilitate the exact configuration of the structure as a whole and the foundation material(s) up to a desired depth which comes under the influence of the structure, when loaded and is much better than the Hetenyi’s Method of beam on elastic foundations in context to taking into account the variation of foundation media, the geometrical disposition of the raft floor and spatial variation of the stiffness while analysing the raft floor of the barrage. Besides this, the dynamic analysis of barrage at various stages of construction has brought into limelight, the need of analysing the barrage structure at each and every stage of the construction. It can also be inferred from the dynamic study of the barrage that mass and stiffness play a key role in the behaviour of the barrage in response to time varying load like earthquake.
REFERENCES