



A Parametric Study of Seismic Behavior of Concrete Gravity Dam using Abaqus

Rahul Chaudhary¹, Vishisht Bhaiya¹, Param Gajbhiye¹

¹Sardar Vallabhbhai National Institute of Technology, Surat, Gujarat 395007, India

Abstract

The Seismic analysis of a concrete gravity dam is a complex problem due to the mutual interaction between reservoir water, rock foundation, and concrete dam. To design earthquake-resistant dams, it is essential to have accurate and reliable analysis procedures to predict the dam response. In this study, seismic analysis of the concrete gravity dam using the finite element model via ABAQUS has been investigated for the Koyna earthquake (1967). A 2-D Finite Element Model of the dam was constructed to simulate a dam-reservoir system. The reservoir water domain is represented as an incompressible and non-viscous fluid continuum having a finite length in the upstream direction. The foundation of the dam is assumed as rigid. Both the mass concrete and foundation are assumed to have homogenous, isotropic, linearly elastic properties. Seismic dam responses are expressed in terms of dam deformations, dam stresses, spectral displacements, mode shapes, natural frequency, and response time history. Spectral results confirm that there is a significant effect of the natural period on the seismic behaviour of the dam. Considering the dam-reservoir-foundation interaction, the maximum principal stress, minimum principal stress, top board displacements, natural frequencies, and mode shapes are analyzed and the results are discussed by comparing results obtained from the proposed FEM model in the literature.

Keywords: Concrete gravity dams, Seismic analysis, Dam-reservoir system, Finite element model