Transactions, SMiRT-24
BEXCO, Busan, Korea - August 20-25, 2017
Special Session for IAEA EESS WA2.3
(Hybrid Simulation to Assess Performance of Seismic Isolation in NPPs)

SIMULATION OF SEISMIC ISOLATION USING ABAQUS SOFTWARE

Carsten Block[1], Markus Jeßberger[2], Fritz-Otto Henkel[3]

[1] Engineering Director, Wölfel Engineering, Hoechberg, Germany
[2] Project Manager, Wölfel Engineering, Hoechberg, Germany
[3] Senior Expert, Wölfel Engineering, Hoechberg, Germany

ABSTRACT

The Extra-budgetary Programme of the International Seismic Safety Centre (ISSC-EBP) is a series of technical study projects supported by IAEA Member State institutions that collaboratively contribute to enhance seismic safety and safety against external hazards at nuclear installations. Within Task 2.3 “Hybrid Simulation to Assess Performance of Seismic Isolation in Nuclear Power Plants” in area 2 “Design for external hazards” one activity was to simulate the seismic behaviour of a base isolated nuclear island.

The target of the benchmark was the behaviour of the isolators, not the modelling of the superstructure. Hence, it was requested that all participants use the computational models for the superstructure provided by KEPCO E&C in SAP 2000 format, adapted to their computer codes with as few changes as possible. The model provided by KEPCO E&C corresponds to the Korean APR1400 nuclear island (Reactor, Fuel and Auxiliary buildings.

The aim of Wölfel was to investigate the capabilities of commercially available software to simulate the nonlinear behaviour of a multi lead-core rubber bearing. For this purpose the software ABAQUS was used. In this presentation, the approach for transferring the stick model from SAP 2000 to ABAQUS and two different approaches for a simplified modelling of the seismic isolators are described. The main analysis results are presented and the possibilities and limits are demonstrated.