Master's Dissertation at the Div. of Structural Mechanics

EARTHQUAKE ANALYSIS OF PIPE SUPPORTS IN NUCLEAR POWER PLANT

Johan Mattsson

Presentation
August 2011

Report
will be published as report TVSM-5182

Supervisors
Per-Erik Austrell
Div. of Structural Mechanics, Lund

Magnus Ohlson
FS Dynamics AB

Daniel Burman
FS Dynamics AB

In cooperation with
FS Dynamics AB

Examiner
Kent Persson, ***PhD***
Div. of Structural Mechanics, Lund

The work is performed at
Div. of Structural Mechanics at the Faculty of Engineering, LU and FS Dynamics AB

LUND UNIVERSITY

Background
The conventional finite element program Pipestress is used to analyze pipe systems from a structural point of view. Components like pipes, velvets and branches can be modeled, but not the pipe supports. Only a stiffness is added at the node on the pipe system were the pipe support is attached. For the dynamic analysis is the response spectrum then added directly on to the pipe, when it not can be added on the pipe support. The missing response from the pipe supports is suspected to change the result in a non-conservative way.

Purpose
The purpose is to study if the conventional methods used to analyze piping systems for building vibration is generally applicable.

Method
The first objective is to make a literature study, which includes building vibration theory and how it is applied in the finite element method. The regulation of earthquake design is researched to make the subsequent modelling in a sufficient way.

Different kinds of pipe supports are mapped to find a suitable pipe system. A simplified model in the finite element program Abaqus is created of the building where the pipe system is located. An acceleration time history is added to the foundation of the model and response spectra are created in the pipe support attachments points.

In the next step a model of the pipe system is created in the conventional used finite element program Pipestress.

A dynamic analysis is made with the response spectra received from the Abaqus building model.

A full model is created in Abaqus by combining the initial Abaqus building model and the pipe system from Pipestress and by adding the pipe supports according to drawings. An analysis is made with the acceleration time history as input and the result is compared with the Pipestress output.

A last comparison is made between an Abaqus analysis of the pipe system with supports and the Pipestress analysis of the pipe system without supports. In this way are response spectra used as input in both analyses and Pipestress is tested if it is conservative.