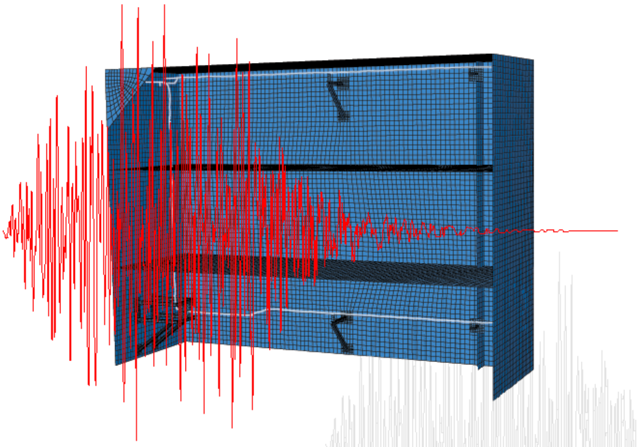
Lund University



EARTHQUAKE ANALYSIS OF PIPE SUPPORTS IN NUCLEAR POWER PLANTS

JOHAN HERLUF MATTSSON

Structural Mechanics

Master's Dissertation

Department of Construction Sciences
Structural Mechanics

ISRN LUTVDG/TVSM--12/5182--SE (1-49)
ISSN 0281-6679

EARTHQUAKE ANALYSIS OF PIPE SUPPORTS IN NUCLEAR POWER PLANTS

Master's Dissertation by

JOHAN HERLUF MATTSSON

Supervisors:
Per-Erik Austrell *Senior Lecturer*,
Div. of Structural Mechanics, LTH, Lund

Magnus Ohlson *M.Sc.* & Daniel Burman *M.Sc.*,
FS Dynamics AB

Examiner:
Kent Persson *PhD*,
Dept. of Construction Sciences, LTH, Lund

Printed by Media-Tryck LU, Lund, Sweden, November, 2012 *(Pl)*.

For information, address:
Division of Structural Mechanics, LTH, Lund University, Box 118, SE-221 00 Lund, Sweden.
Homepage: http://www.byggmek.lth.se

Preface

It has been exciting to work on this project. And this is my first contribution to the field of structural analysis. For the help I would like to thank Daniel Burman and Magnus Ohlson from FS Dynamics in Gothenburg for the consulting. But I am also thankful to Kent Persson and Per-Erik Austrell at the division of Structural Mechanics in the department of Civil Engineering at Lunds University.

Abstract

This is the resulting report from a master thesis project in the field of structural analysis, with a focus on earthquake engineering.

In this project are pipe supports' impacts on seismic pipe design analysed. This is done by modelling a building, pipe supports and a pipe system in two different computer softwares. Pipestress is commonly used for design of pipes at nuclear plants, and Abaqus has applications in many fields of mechanics.

Earthquake data for seismic design at nuclear plants in Sweden is used as input and different kinds of analysis are performed. Output from the two softwares is compared to see how the responses differ and if it is related to the pipe supports.

The support eigenmodes and forces calculated, differ between the two softwares. The difference in eigenmodes and therefore also the forces may be explained by modelling issues concerning the pipe bends. However, the modelling technique for the pipe supports does not seem to impact the result.

Contents

1. Introduction

Earthquakes happen frequently in Sweden. Most of them are of small magnitude and harmless to structures. But this risk is important to deal with at nuclear power plants to avoid major accidents. Earthquake engineering is therefore applied while validating existing plants and in design of new components.

1.1 Background

1.1.1 Nuclear Power Plants in Sweden

Today Sweden has 10 operating nuclear reactors located at Ringhals (4 reactors), Oskarshamn (3 reactors) and Forsmark (3 reactors). The reactors in Oskarshamn, Forsmark and Ringhals 1 are boiling water reactors, BWR, and Ringhals 2-4 are Pressure water reactors, PWR. The pipe system to be investigated belongs to a Swedish nuclear power plant.

A BWR consists of four main components. First of all, a reactor tank within a reactor enclosure, where water is vaporized by uranium fission. Secondly, turbines where steam from the reactor rotates a shaft, which drives an electric generator. The turbines are working due to a seawater cooled condenser which makes the steam rush through the turbines. The condensed water is finally pumped back to the reactor where the process starts all over. A PWR also has steam generators, which separates the reactor from turbine systems.

1.1.2 Pipe Supports at Nuclear Power Plants

Within a nuclear power plant there are enormous amounts of piping with various pipe support designs. Pipes are mounted on consoles, fixed to base structures or hanged in pendulums. Design varies, but depends mainly on loading, safety level, pipe dimension, attaching structure and distance between pipe and attaching structure. It also has to consider surrounding installations, passageways, installation complexity and costs.

The forces on a pipe system and its supports come from normal operation, start up, shut down and emergency. The main design loads are dead weight, live load from content, temperature and pressure transients, earthquake loads etc. Pipe systems outside the plant's building are to be designed for loads from snow and wind. The protecting building structure shall rather than the pipe system be designed for loads from bombs and colliding airplanes.

There are different design requirements depending on the task and location in a nuclear power plant. A safety classification system in four steps is used, where class 1 is strictest and class 4 follows common industrial code [1]. Systems in class 1-3 have to be checked for seismic loads. American Society of Mechanical Engineers (ASME) and the American Society of Civil Engineers (ASCE) are normally used in class 1-3 and European standard for class 4.

The distance between pipes and building structures varies, but a short distance makes it easier to construct a stiff pipe support. The supports are normally made out of standard hot rolled steel profiles, which are welded together. The supports are mostly mounted onto the building structure with expanders or are casted into the structure's concrete. The supports pipe attachments are normally preventing movement of the pipe perpendicular to the pipe flow. The rotation of the pipe is hard to prevent, but firm support structures are used to achieve pipe anchors.

1.1.3 Pipestress

The finite element based software Pipestress is conventionally used for designing pipe systems at nuclear power plants. By using modal superposition it is possible for the program to make computer power efficient dynamic calculations without using demanding direct integration. A disadvantage is that modal analysis only can be applied on linear calculations and not for plasticity. The software is specified to handle loads from dead weight, thermal expansion, internal pressure and dynamic loads associated to for example earthquakes. Dynamic accelerations can be applied as time histories or response spectra. Due to efficient calculations load cases can be combined and analyzed according to ASME regulations.

The software consists of two subprograms, the first for input coding, called Editpipe, and the second for job processing, named Editpipe Manager. In Editpipe the user is defining different cards depending on analysis, loads, materials, cross sections and geometry. For each card, parameters are added to adjust and define input data. The geometry of the pipe system can be observed before sending the input file to Editpipe Manager. After the analysis, stresses, deformations, and mode shapes are observable in Editpipe. Editpipe manager provides a number of files where monitor data and results can be found.

1.1.4 Abaqus

Abaqus is a software with many applications in both structural and fluid mechanics. It is a finite element based program for static and dynamic problems that can be linear as well as nonlinear. The software does not have any direct connection to any building code and is therefore applicable in various fields. Many kinds of beam, plate, and solid elements are available for structural analysis.

In a similar way to Pipestress, Abaqus is divided into subprograms. In Abaqus CAE the analysis input file is to be created. It is a subprogram with a workspace where geometry, properties, assembly, steps, interactions, loads, and mesh are modified. Some analysis options have to be manually typed in a keyword editor, due to limitations in Abaqus CAE. The job is solved in Abaqus Standard or Abaqus Explicit depending on its character. The result is visualized in Abaqus CAE. This is either done in form of tables, graphs or figures.

1.1.5 Earlier Studies

Supports' stiffness subjected to dynamic loads have been investigated in a study made by the Swedish nuclear power plant calculation group [2]. The result of the study is a table of appropriate stiffness values for different pipe diameters. The design process of pipe systems is iterative due to the dependence between pipe support and piping, but this table avoids this. The result of this paper is useful when checking the pipe supports in this report.

In a master thesis by Burman and Ehrenborg [3] pipe systems in Pipestress and Abaqus have been analyzed and compared. This is done with modal superposition as well as direct integration. One result was a Python code to transform models from Pipestress to Abaqus. A conclusion was that it is possible to use Pipestress models in Abaqus and obtain same natural frequencies.

1.2 Purpose

Pipestress is commonly used at nuclear plants in Sweden to analyze pipe systems in a structural point of view. Components like pipes, bends, valves, T-pieces etc are modelled in detail, but not pipe supports. Stiffness from the supports are added at the node on the pipe where the pipe support is attached. For the dynamic analysis the earthquake response spectrum is added directly to the pipe, since it cannot be added at the building structure. The missing response from the pipe support is suspected to change the design in a non-conservative way. The objective of this master thesis is to investigate if it is the case and eigenmodes and support forces are therefore compared.

2. Theory

There is a lot of theory related to structural dynamics and earthquake engineering. In this chapter is the commonly used response spectrum described, followed by modelling and analysis methods.

2.1 Response Spectrum

A response spectrum is a plot of response, i.e. acceleration, velocity and/or displacement as a function of frequency. An example spectrum is shown in *figure 2.1.* In earthquake engineering periods ($T = 1/f$) are commonly used instead of frequencies unlike in general dynamics. Response spectra are used to design subsystems (i.e. piping) that do not impact their systems' (i.e. building) responses. The system should therefore be much stiffer or/and have a bigger mass than the subsystem.

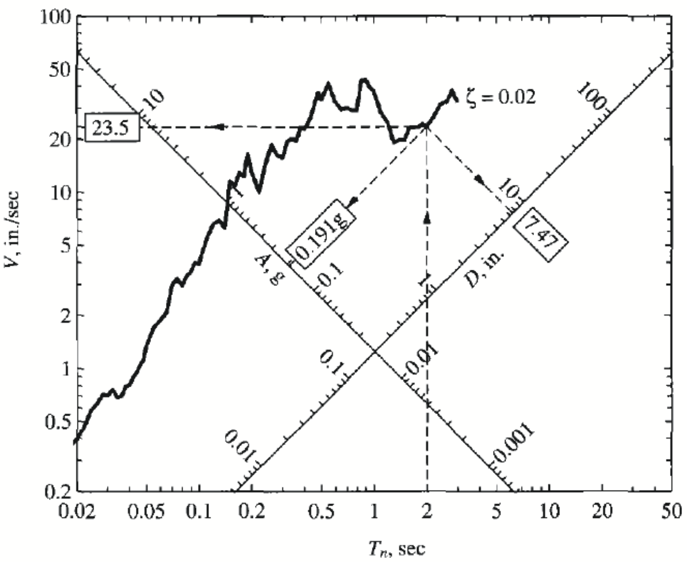


Figure 2.1: Tripartite response spectrum, El Centro ground motion, $\zeta = 2\%$ [4]

A single degree of freedom (SDOF) system is used to create a response spectrum since a substructure can be defined by natural frequency, f_n, and damping factor, ζ [4]. A SDOF system has two nodes, mass (m), stiffness (k) and damping (c) (see *figure 2.2*). The first node in the SDOF system is the structure (in this case the building) and the second node is the substructure (in this case the pipe system with supports).

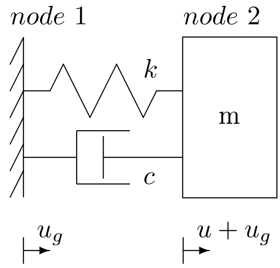


Figure 2.2: SDOF system

From the free-body diagram the equation of motion is formulated for the system.

$$m\ddot{u} + c\dot{u} + ku = -m\ddot{u}_g(t) \tag{2.1}$$

The equation is rewritten by introducing natural angle frequency, $\omega_n^2 = k/m$, and damping ratio, $\zeta = c/(2m\omega_n)$ and knowing that the angle frequency is equivalent to $2\pi f$.

$$\ddot{u} + 4\pi\zeta f_n\dot{u} + (2\pi f_n)^2 u = -\ddot{u}_g(t) \tag{2.2}$$

For an arbitrarily ground acceleration, $\ddot{u}_g(t)$ this differential equation is not possible to solve analytically. The system can instead be solved with numerical integration for every time step defined by the time history. There are many methods in doing this and one of them is called the Central Difference Method. This method uses approximations of accelerations and velocities calculated from displacements from two time steps.

$$\dot{u}_i = \frac{u_{i+1} - u_{i-1}}{2\Delta t} \qquad \ddot{u}_i = \frac{u_{i+1} - 2u_i + u_{i-1}}{(\Delta t)^2} \tag{2.3}$$

The response equation is obtained by inserting *equation 2.3* into *equation 2.2* and do a rearrangement.

$$\left(\frac{1}{(\Delta t)^2} + \frac{2\pi\zeta f_n}{\Delta t}\right) u_{i+1} = \left(\frac{2}{(\Delta t)^2} - (2\pi f_n)^2\right) u_i - \left(\frac{1}{(\Delta t)^2} - \frac{2\pi\zeta f_n}{\Delta t}\right) u_{i-1} - \ddot{u}_{g,i} \tag{2.4}$$

The displacement for the next step is in this way calculated from the two previous steps. Velocity and acceleration can also be obtained from the displacements with *equation 2.3*.

Initially the previous displacement step is calculated from *equation 2.3* with i=0 and the boundary condition for velocity or acceleration obtained from the rewritten version of equation of motion, *equation 2.2*.

$$u_{-1} = u_0 - \Delta t \dot{u}_0 + \frac{(\Delta t)^2}{2} \ddot{u}_0 \tag{2.5}$$

Equation 2.4 is solved for frequencies of interest (normally from 0 to 50 Hz) and for a given damping ratio. The response spectrum is then obtained by plotting maximum response as a function of frequency.

There are specific requirements on Δt in order for the numerical integration to converge. The time step is normally set to 2 milliseconds for frequencies up to 50 Hz to receive accurate results, but it works as long as it fulfils the following stability requirement:

$$\Delta t < \frac{1}{\pi f_n} \tag{2.6}$$

In design it is often hard to find appropriate time histories to turn into response spectra. Therefore earthquake ground motion data have been studied to find ways to create spectra. It was concluded that earthquake magnitude, distance from source and site conditions influence the response spectra the most [5]. A generalised response spectra shape can be used for simple code design. This kind of method still needs a peak ground acceleration (PGA) to obtain the spectrum magnitude. The PGA can be found in code maps and from attenuation relationships.

Scaling is done to response spectra according to the code to account for the importance of the structure but also for the structures ductility [5]. This is done by introducing a reduction factor that represents how much it can deform after yielding. Other spectral modifications are done for structures likeliness to elongate its periods when cracking and for the reason that narrow spectral peaks should not be missed due to calculation errors.

2.2 Modelling Methods

A mathematical structural model is needed in order to perform a dynamic analysis. It is desirable to be as simple as possible due to computing power and understanding. Especially time history analysis requires many time steps and large calculations (see section Analysis Methods). There are many ways to model structures and the most common are presented here below [5]:

- Substitute
- Stick
- Detailed

The substitute model is the simplest mathematical presentation of a structure. It consists of a single degree of freedom (SDOF) model, and is normally defined with a mass, stiffness, distance and damping. For inelastic and complex problems are simple models reducing the calculations considerably. A substitute model can be used to get a rough approximation, but is likely to be sufficient enough for structures where the mass is located in a point, like a water tower.

The stick model is also a simple structural model, but with multi degree of freedom (MDOF). It has lumped masses in a few locations along a line to represent a more complex dynamic behaviour. Due to the simplification are local responses likely to be inaccurate. But this model type is sufficient for structures like shear-buildings as skyscrapers where mass can be lumped to each floor.

The detailed model is accounting for the structure's geometry and has multi degrees of freedom (MDOF). A model is normally done in 2D, but unsymmetrical structures can be modelled in 3D. Sections of members like beams, plates and shell are defined accurately. Exact geometry and inelasticity in member connections can be modelled in rigorous cases. This is normally not done due to modelling time, computing power and uncertainties about the structure. Detailed models are normally done using the Finite Element Method.

2.3 Finite Element Method

The finite element method, FEM, is a technique to analytically solve complex problems, which can be static or dynamic, with linear or nonlinear behaviour. FEM has applications in many technical fields from electronics to mechanics. In most softwares dealing with structural mechanics this method is applied. Here follows a basic description of what it is about.

The method solves problems by dividing them into small solvable elements. In structural mechanics members like beams, plates and shells are normally used to define a structure, and these are cut in small pieces, i.e. elements. For every element local matrixes are formulated with properties such as mass, stiffness and damping at each nodal degree of freedom (DOF). A linear beam element has two nodes, one at each end. But there are normally six DOFs at every node, three defining movement and three defining rotation. The element matrixes are assembled together in global matrixes for the whole structure. The size of the global matrixes represents the total number of DOFs and the matrixes describe how the DOFs are related to each other.

Global boundary conditions are described in vectors corresponding to the global DOFs. A structure can for example be fixed or have outer forces acting on it. An equation system consisting of the global property matrixes and the boundary condition vectors is then formulated and solved. In structural mechanics actions and responses such as forces and displacements are obtained. From that can stresses and strains in the members be calculated [6].

2.4 Analysis Methods

There are numerous ways to analyse a structure exposed for dynamic loads i.e. earthquakes. Methods with time history input as well as response spectrum input can be used [8, p. 17,18&22]. There are both dynamic and static methods that results in acceptable solutions. Some of them solves the problem in frequency domain and others time domain. Certain analysis methods are associated with elastic and others are inelastic behaviour. The most common techniques are shown below and explained briefly in this section [5]:

- Equivalent Static
- Pushover
- Response History
- Dynamic Pushover
- Modal
- Spectral

The equivalent static analysis method (also called equivalent lateral force method, ELF) is the simplest way to analyse a structure with dynamic loading such as a seismic one. It is an elastic method, but non-linearity such as second order effects can be included. The structures first eigenfrequency is assumed to represent the whole response since it normally accounts for 70-80% of it. Acceleration from a response spectrum is multiplied with a percentage of the structural weight to get horizontal forces. From a static calculation is the response obtained. This method is only applicable for regular symmetric structures. It is frequently used in building code for small earthquakes and simple structures where lower accuracy is needed.

The pushover analysis method is similar to the equivalent static method, since horizontal static forces are applied on the structure and a regular symmetric structure is needed. But the whole force-displacement curve is plotted by increasing intensity of the forces or the displacements. This is therefore an inelastic method and requires a set of static calculations. The force distribution on the structure corresponds to one or more modal shapes. In an adaptive version of the pushover method the distribution is changed during the analysis. For every static calculation new modal shapes are calculated. This method is therefore useful for structures that change eigenfreqencies such as concrete structures with much cracking.

The response history analysis is a time domain method for a structure with dynamic loading. This method uses numerical integration through time stepping and is therefore a natural way to get the response. Both elastic and inelastic calculations can be done. Depending on if the problem is conditionally stable (explicit) or unconditionally stable (implicit) different time-marching schemes are used. This is a very accurate method, but also demanding in computer power for big structures with many nodes.

The dynamic pushover analysis (also called incremental dynamic analysis, IDA) is a dynamic way to perform a pushover analysis. It also accounts for both nonlinearities in geometry and mechanical system in similarity with both adaptive pushover and response history analysis. The time history intensity input is scaled from zero to where the structure fails, and a response history analysis is performed for each intensity level. The inelastic behaviour of the structure is then plotted in a curve over maximum spectral acceleration and maximum story drift.

The modal analysis method is combining SDOF responses corresponding to the structure's modal shapes into a MDOF response with a transient force input. This method demands a lumped mass matrix to be able to decouple the system. It is a linear method and can be considered as a frequency domain solution. More about this method is found in the section Modal Analysis.

The spectral analysis (also called modal-spectral analysis or response spectrum analysis) uses modal properties in similarity to modal analysis but is a static method. The transient input loading is represented by a response spectrum and in this way is only the maximum responses obtained. This method is as well as the modal analysis only linear [8, p. 20] and is a frequency domain solution. More about this method is found in the section Spectral Analysis.

2.5 Modal Analysis

The responses over time of a building subjected to an earthquake can be determined using the modal analysis. A structure has many eigenmodes corresponding to different frequencies. Each eigenfrequency triggers the building into movement in a curtain way. In modal analysis are the responses from each mode up to a cut off frequency added to obtain the total response.

A frequency analysis is done on a multi degrees of freedom (MDOF) system without damping to find the natural frequencies. First is an arbitrary structure's equation of motion in free vibration formulated as *equation 2.7*, containing the mass matrix M, the stiffness matrix K, the relative acceleration vector $\ddot{u}$ and the relative displacement vector u.

$$\mathbf{M\ddot{u}} + \mathbf{Ku} = 0 \tag{2.7}$$

A harmonic solution is desired on the form $u = Acos(\omega_n t)\Phi$, where Φ holds the mode shapes and A is a constant. Two derivations in respect to time, t, results in $\ddot{u} = -\omega_n^2 Acos(\omega_n t)\Phi$. By inserting the paraphrases for displacement and acceleration as well as the angle frequency, $\omega = 2\pi f$ into *equation 2.7* acquires the homogeneous system *equation 2.8.*

$$(\mathbf{K} - (2\pi f_n)^2\mathbf{M})\mathbf{\Phi} = 0 \tag{2.8}$$

The natural frequencies are found by first rewriting *equation 2.8* into an eignenvalue problem. This problem has a trivial solution for an equation system with a determinant equivalent to zero.

$$det(\mathbf{K} - (2\pi f_n)^2\mathbf{M}) = 0 \tag{2.9}$$

The modes shapes equals the eigenvectors obtained from *equation 2.8.*

The response for each mode is found when the mode shapes and corresponding natural frequencies are known. Once more is the equation of motion formulated for a MDOF system, but this time with the damping matrix, C, and the absolute earthquake ground acceleration, $\ddot{u}_g$. I is a vector with ones and has the same size as u.

$$\mathbf{M\ddot{u}} + \mathbf{C\dot{u}} + \mathbf{Ku} = -\mathbf{MI\ddot{u}_g} \tag{2.10}$$

Equation 2.10 can then be reformulated using modal coordinates, $u = \sum_{i=1}^{N} \phi_i q_i(t) = \mathbf{\Phi q}$ and by premultiplying both sides with Φ^T.

$$\sum_{i=1}^{N} \phi_n^T \mathbf{M} \phi_n \ddot{\mathbf{q}} + \sum_{i=1}^{N} \phi_n^T \mathbf{C} \phi_n \dot{\mathbf{q}} + \sum_{i=1}^{N} \phi_n^T \mathbf{K} \phi_n \mathbf{q} = -\phi_n^T \mathbf{MI\ddot{u}_g} \tag{2.11}$$

This equation is solved for eigenfrequencies up to the cut of mode N. It is done in the time domain with a time stepping method like the Central Difference Method (described in the section Response Spectrum). Another way to solve the equation of motion is to do it in the frequency domain with a fast Fourier transform. A third way to find the modal analysis solution is to use a convolution integral like Duhamel's integral.

2.6 Spectral Analysis

Spectral analysis is using modal properties. The analysis starts with finding out the natural frequencies for the subsystem by setting up an eigenvalue problem like in the section Modal Analysis. From the mass matrix and the modal vectors is the generalized mass, $\hat{M}_i$, and the modal participation factor, Γ_i, for each mode calculated according to *equation 2.12.*

$$\hat{M}_i = \mathbf{\Phi_i^T M \Phi_i} \qquad \Gamma_i = \mathbf{\Phi_i^T MI}/\hat{M}_i \tag{2.12}$$

The corresponding accelerations are picked out from the response spectra (two horizontal and one vertical) for every eigenmode. Usually only three modes are sufficient to get 85-90% of the response, but up to 100 modes might be needed for long structures like bridges [5]. For every spectral acceleration, $S_{a,i}$ an equivalent force vector is calculated according to *equation 2.13.*

$$\mathbf{F_{max,i}} = \mathbf{M\Phi_i\Gamma_i} S_{a,i} \tag{2.13}$$

The force vectors are used in static analysis and responses are obtained for each mode. These modal responses, r_m, has to be combined to obtain the total response, r. The maximum modal responses are not likely to occur at the same time and it would therefore be an overestimation to sum up the modal responses. A common way is instead to use the square root of sum of squares (SRSS) rule.

$$r \simeq \left(\sum_{n=1}^{N} r_n^2 \right)^{1/2} \tag{2.14}$$

The SRSS rule has its limitations and should be avoided for subsystems with closely spaced natural frequencies like piping systems in nuclear power plants and multi storey buildings with unsymmetrical plan [4]. In those cases is it better to apply the complete quadratic combination (CQC) rule.

$$r \simeq \left(\sum_{i=1}^{N} \sum_{n=1}^{N} \rho_{in} r_i r_n \right)^{1/2} \tag{2.15}$$

The CQC rule introduces a correlation coefficient, ρ_{in}, which varies between zero and one depending on how close the modes are. Modes that are closely spaced have in this method bigger impact on the result than in the SRSS rule. The correlation coefficient is calculated according to *equation 2.16* when there is equal damping, ζ, at each mode [7].

$$\rho_{in} = \frac{8\zeta^2(1+\beta_{in})\beta_{in}^{3/2}}{(1-\beta_{in}^2)^2 + 4\zeta^2\beta_{in}(1+\beta_{in})^2}, with\ \beta_{in} = \frac{f_i}{f_n} \tag{2.16}$$

There are other ways to combine the modal responses as well but the SRSS rule and the CQC rule are the most used methods.

3. Method

The thesis outline is divided into four phases. It starts with a literature study to gain understanding of the topic and is followed by three models. The models are created to compare results from different types of analyses in Abaqus and Pipestress.

3.1 Initial work

The first phase in this thesis project consists of a literature study. Response spectrum theory, dynamic modelling methods and seismic analysis methods are studied. Requirements for earthquake design are also investigated to gain understanding of how dynamic problems are dealt with in practice.

In this phase, pipe support designs are mapped to find a suitable pipe system for this thesis. A pipe system with normal pipe support stiffness is used.

3.2 Abaqus Building Model

A simplified model of a building where the pipe system is located (A.B.M.) is created in the finite element program Abaqus. The actual nuclear power plant is too complex, containing tanks and other piping, and therefore a simple building geometry is applied. The design is chosen to represent a likely concrete building's behaviour due to vibration.

An acceleration time history is applied at the building model foundation and a modal analysis is made. Output time histories are collected in the points on the walls where the pipe supports are attached. From these are response spectra over the maximum accelerations created using the program MatLab.

3.3 Pipestress Pipe Model

A model of the pipe system (P.P.M.) is created in Pipestress. The system is modelled with details as valves and branches. Stiffness from the actual pipe supports are used, but not the supports' mass distribution nor damping.

A spectral analysis is performed on the pipe system, and the behaviour of the system is observed. Response spectra from the Abaqus building model are used. These are applied as modal force vectors directly on the pipes at its attachments, i.e. where the pipe ends are fixed to the walls (anchors) and where the pipe supports are located.

3.4 Abaqus Building, Support & Pipe Model

In the initial Abaqus building model the pipe system from Pipestress is inserted, which results in the Abaqus building, support & pipe model (A.B.S.P.M.). The pipe supports are inserted into the Abaqus model according to drawings. This implies a full model containing the building, pipe supports and piping.

This full model is analysed with modal analysis in Abaqus with the same ground acceleration time history used in the Abaqus building model. The result is compared to the Pipestress pipe model to find out the reliability of the Pipestress analysis. The result can also be compared to the Abaqus building model to find out if the buildings natural frequencies have changed.

4. Setup & Input

In this chapter is the setup of the three models and input to be able to run the analyses described. An explanation of how the acceleration input is created is added to give understanding of limitations and credibility.

4.1 Pipe System

4.1.1 Geometry

A pipe system with pipe supports (see section Pipe Supports) is taken from a Swedish nuclear power plant. It has common dimensions on the pipe system in relation to the pipe supports' stiffness values [2]. The system is designed to withstand earthquake loads.

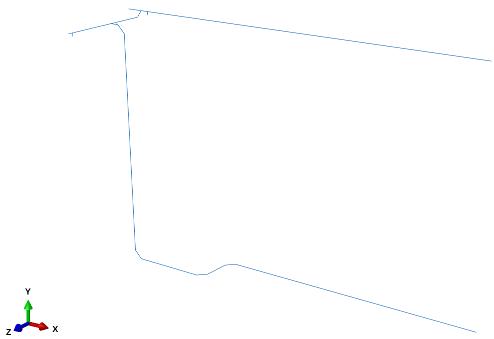


Figure 4.1: Pipe system

The pipes in the system are of two types. The majority pipe diameter in this model is 168.3 mm with wall thickness 3.4 mm. Most of the upper part of the system has a slightly smaller dimension, 129.0 mm, with wall thickness 2 mm. On the system there are three valves and two T-pieces. The pipe system has three ends with anchors and five additional pipe supports. The upper branch is for simplicity modified to look more like the ground level.

4.1.2 Material and mass

The pipes and the pipe supports are made of steel and have the same material properties according to ASCE [8, p. 10], where the modulus of elasticity is 200 GPa and the Poisson's Ratio is 0.3.

Water inside the pipes is added in form of higher density for the steel in the pipes. Mass for the 168.3 mm pipe including water is 13.8 kg/m and for the 129.0 mm pipe it is 6.26 kg/m. There are two branches and four valves. Three of the valves have an actuator with mass 25 kg. Each valve is divided in two 12 kg point masses along the valves.

Lumped mass matrixes are used for the pipe system. Both a lumped mass matrix as well as a consistent mass matrix may be used under the condition that a lumped mass matrix has equivalent total mass with the same centre of gravity as a consistent mass matrix.

4.1.3 Damping

For analysis in both Pipestress and Abaqus direct modal damping are used with damping ratio according to ASCE [8, p. 11]. The damping ratio for steel is 3%.

4.1.4 Elements

The pipe system is modelled in both Pipestress and Abaqus with equivalent beam elements [3]. These have six degrees of freedom per node, where three are displacements and three are rotations. Different element formulation exists for straight and bended elements. In Abaqus are the straight elements used called PIPE31.

4.2 Pipe Supports

4.2.1 Geometry

The system has five pipe supports of three different types, A-C. All the pipe supports are consoles made out of standard steel cross sections. These are all fix fastened on to the buildings walls. All the pipe supports are modelled with simplified stiff plates for the pipe guides (where the pipe and support are connected). The plates reach up to the centre of the pipe and constrains allow for pipe movement along the pipe. In the calculations are no clearance and friction accounted at the guide shoe.

The pipe support A consists of a 510 mm long UPE 120 beam welded onto a steel plate, 410x410x20 mm. The plate is fixed at the wall with expanders. The geometry is shown in *figure 4.2*.

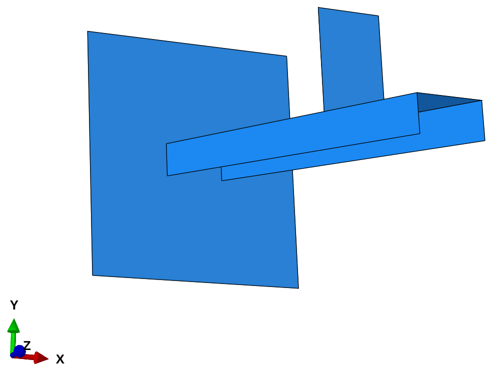


Figure 4.2: Pipe support A

The pipe support B consists of two HEB 100 beams. One is pointing 1100 mm straight out of the wall and the other is welded underneath in a 45 degree angle to support the first one. Each beam has a steel plate, 410x410x20 mm, welded on the end to be mounted with bolts to the wall. The geometry is visualized in *figure 4.3*.

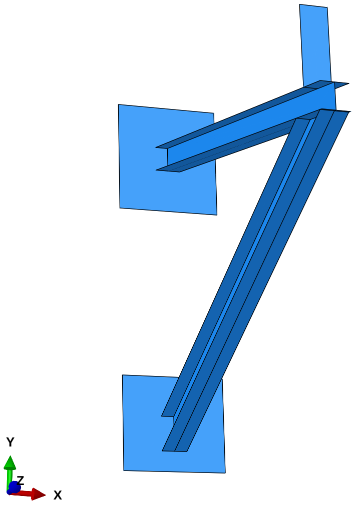


Figure 4.3: Pipe support B

The pipe support C is a complex construction consisting of 10 pieces of HEB 100 beams, two small wall mounting plates, 410x410x20 mm, and two big wall mounting plates, 410x890x20 mm. The geometry is visualized in *figure 4.4*. The purpose of this structure is to support the pipe as it makes a turn from horizontal to vertical direction. This construction is only used once in the building model.

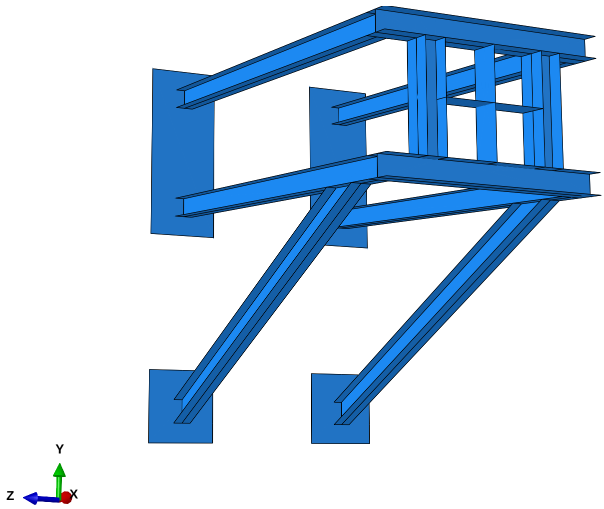


Figure 4.4: Pipe support C

4.2.2 Material

The pipe supports have the same material properties as steel pipes, where the modulus of elasticity is 200 GPa and the Poisson's Ratio is 0.3.

4.2.3 Damping

For the supports in the full Abaqus model (A.B.S.P.M.) direct modal damping is used with damping ratio 3% [8, p. 11].

4.2.4 Elements

The pipe supports are modelled in Abaqus with a quad shaped shell element. The mid section in each cross section is modelled and the thickness is used to get the right moment of inertia. The size of the elements was chosen to give a good result without using unnecessary computation power.

4.2.5 Stiffness

In Pipestress are stiffness values for each support and perpendicular direction of the pipe (x is in the pipe direction) from Abaqus used. In Abaqus are unit loads applied on the supports' pipe guides when the supports' back plates are fixed. The stiffness values for each support are calculated from the resulting displacements. The stiffness in each global direction are summarized in *table 4.1*.

Support	k_x *[kN/mm]*	k_y *[kN/mm]*	k_z *[kN/mm]*
A	(0.753)	2.30	2.22
B	(0.0739)	18.5	1.45
C	(26.9)	11.8	1.50

Table 4.1: Stiffness for pipe supports

4.3 Building

4.3.1 Geometry

A simplified model for the building is created instead of the complex geometry of the real nuclear power plant. Only three walls are used to be able to visualize the piping system. The floors and walls are connected with constrains that allow rotations. This is a realistic assumption which results in lower eigenfrequencies. The building has been chosen to fit the pipe system and to have a lowest eigenfrequency at 7 Hz. That is what has been used in the design of the San Onofre nuclear power plant in California [4, p. 28].

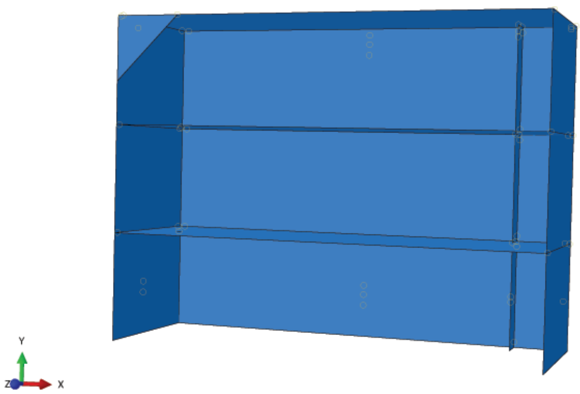


Figure 4.5: Building model

The simplification of the building may lead to other forces on the pipe system than used in design. However, the main purpose of the thesis is to compare the pipe system and its supports using different methods and therefore the exact form of the building is not of importance.

4.3.2 Material

The Concrete is modelled as noncracked, but should be modelled as cracked or noncracked depending on the stress level. The modulus of elasticity, E, for concrete is chosen according to ASCE [8, p. 10-11], and depends on the weight in pcf and the strength in psi. The concrete of the building is presumed to be of the European quality C30 [9], which implies that the 28 day compressive strength, f_c', is 30 MPa (4351 psi). The weight of concrete, w_c, is assumed to be 2400 kg/m^3 (150 pcf).

$$E = w_c^{1.5} * 33 * \sqrt{f_c'} = 28\ MPa \tag{4.1}$$

The Poisson's Ratio, ν, for concrete is 0.17 according to ASCE [8, p. 10].

4.3.3 Damping

Direct modal damping is used for the analysis in Abaqus with damping ratios chosen according to ASCE [8, p. 11]. The damping ratio for concrete is 5%.

4.3.4 Elements

The same quad shaped shell element is used for the building as for the pipe supports. The mid sections in the walls, floors and roof are modelled and the thickness values are used to get the right moment of inertia. The size of the elements was chosen to give a good result without using unnecessary computation power.

4.4 Time History

The earthquake applied on the building structure is taken from the former Swedish Nuclear Power Inspectorate's (SKI) project Seismic Safety [10]. The SKI report includes acceleration time histories for two qualification levels, 10^{-5} and 10^{-7}, where the first one with smaller accelerations is used, *figure 4.6*. It corresponds to an earthquake likely to occur once in hundred thousand years at a Swedish NPP. It contains ground accelerations for two vertical and one horizontal direction. These are applied in the three orthogonal directions that correspond to the structures principle axes [8, p. 18].

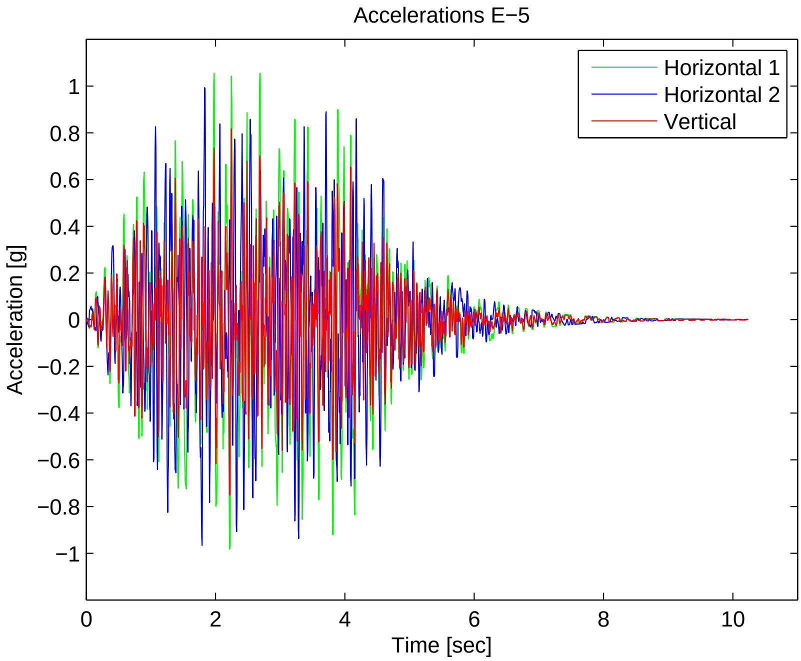


Figure 4.6: SKI's acceleration time histories for the three principle axis

The time histories have 10 seconds duration and 0.005 seconds time steps. Values of maximum acceleration over maximum velocity are bigger than 1.2 and indicate a short source distance and hard soil. A site is likely to experience many different types of seismic loading depending of epicentre and wave propagation. In proper code design are three to five records with different durations and shapes for each direction demanded [5].

Here follows a summation of how SKI acceleration history was derived to give understanding for limitations of this analysis.

There is a lack of earthquake records in Sweden. SKI has with the assistance of the power plant owners Vattenfall, Sydkraft and OKG in 1992 designed earthquake ground motion for Swedish plants [10].

Investigations of the geological conditions at the nuclear power plant sites were carried out as well as likely locations and characteristics of earthquake faults. The earthquake energy parameter moment magnitude (more exact than for example the Richter scale) was described as well as a likely focal distance (distance from site to underground earthquake source).

Response spectrum scaling for magnitude is normally bad due to inelasticity in the ground [5]. Therefore Japanese earthquakes with similar moment magnitude and focal distance were used to develop response spectra instead of scaling up small Swedish earthquakes. Scaling was however done to fit Swedish geological conditions (hard rock) as well as modification for frequency content and probability limits [10].

Earthquake time histories were synthetically derived in the project from the response spectra as calculated above. This was done with a random phase distribution in combination with an empirically based intensity time function [10, p. 17].

4.5 Response spectra

The accelerations over time for each orthogonal direction are gathered in eight points in the Abaqus Building Model, which makes total 24 time histories. Of these eight points three are where the pipe system is fixed to the walls (anchors) and five where the pipe supports are located. A representative middle point on each pipe support's back plate is chosen for the time histories.

The time histories are transformed into response spectra (see section Response Spectrum), which results in 24 response spectra. For simplicity usually the maximum response for a whole room/floor and direction is used in design [8, p. 22&40], but not in this study.

In response spectra the peak responses are broaden 15% according to the ASCE regulation [8, p. 32]. Peak broadening is to include uncertainties when generating structure response. The input spectra can be found in Appendix A.1.2.

5. Results

The results from the Abaqus and the Pipestress analyses consist of eigenmodes and support forces. These depend closely of each other and this is further discussed in chapter 6.

5.1 Eigenmodes

The eigenmodes and frequencies for the Abaqus Building Model (A.B.M.), the Pipestress Pipe Model (P.P.M.) and the Abaqus Building, Support & Pipe Model (A.B.S.P.M.) are found in Appendix. In *table 5.1* are the natural frequencies sorted by shape.

	A.B.M.		*P.P.M.*		*A.B.S.P.M.*	
Shape	*freq. [Hz]*	*mode*	*freq. [Hz]*	*mode*	*freq. [Hz]*	*mode*
1	-	-	5.36	1	-	-
2	-	-	6.39, ≈9.14	2,5	6.57	1
3	6.83	1	-	-	6.82	2
4	-	-	17.38	4	7.13	3
5	-	-	-	-	8.87	4
6	9.84	2	-	-	9.84	5
7	-	-	7.06	3	10.00	6
8	10.19	3	-	-	10.19	7
9	-	-	10.00,10.57	6,7	11.46,11.65	8,9
10	12.55	4	-	-	12.51	10
11	-	-	≈11.37	8	12.78	11
12	13.77	5	-	-	13.72	12
13	-	-	-	-	13.81	13
14	-	-	14.51	9	14.84	14
15	15.56	6	-	-	15.56	15
16	-	-	≈14.93	10	15.59	16

Table 5.1: Eigenfrequencies from the three models sorted by shape

It can be concluded that the mode shapes associated with the building are independent from those associated with the pipe system and pipe supports. Shapes 3, 6, 8, 10, 12 and 15 are natural frequencies only triggering the building into oscillation, whereas shapes 1, 2, 4, 5, 7, 9, 11, 13, 14 and 16 are associated with the piping.

It can be seen from the A.B.M. and the A.B.S.P.M. that the eigenfrequencies of the building hardly change when the pipe system and pipe supports are added.

The pipe eigenmodes are over all different in the Pipestress Pipe Model and the Abaqus Building, Support & Pipe Model. The P.P.M.'s first mode cannot be found in the A.B.S.P.M. The shapes 5 and 13 do not have any representation in the Pipestress Pipe Model, even if they are associated with the pipe in the Abaqus Building, Support & Pipe Model.

Other modelling methods are tested to find the reason for the differences in pipe eigenmodes. More about this is found in the discussion. A pipe system model in Abaqus looking exactly like the one in Pipestress (P.P.M.) was created (Abaqus Pipe Model, A.P.M.), i.e. a model with pipe support stiffness values from Abaqus. Eigenmodes from this model is presented in *table 5.2*. The result from another model with no pipe bends (perpendicular changes in pipe direction) in Pipestress (P.P.M. bendless) is also presented in same table. Only the shape numbers associated with the pipe system is included in the table.

	A.P.M.		*P.P.M. bendless*	
Shape	*freq. [Hz]*	*mode*	*freq. [Hz]*	*mode*
1	-	-	-	-
2	6.57	1	6.71	1
4	7.12	2	7.15	2
5	8.85	3	9.42	3
7	10.07	4	9.61	4
9	11.47,11.66	5,6	11.58,11.71	5,6
11	12.65	7	12.65	7
13	14.28	8	14.28	9
14	14.82	9	13.94	8
16	15.74	10	14.82	10

Table 5.2: Eigenfrequencies from two alternative ways to model the pipe system

The bends in the Abaqus Pipe Model are modelled in four different ways. The first way is with one bended element per curve. The second is to use small straight elements and the third has one straight element between the bends endpoints. The last way is to have perpendicular bends (bendless). All these methods results in similar natural frequencies in Abaqus. In *table 5.2* is the first method presented.

5.2 Support Forces

The maximum support forces at the pipe supports in the Pipestress Pipe Model and Abaqus Building, Support & Pipe Model are presented in *table 5.3* and *table 5.4*. The forces in the X-direction are all zero since the pipe is allowed to move in that direction without friction.

Support	*X-Force [kN]*	*Y-Force [kN]*	*Z-Force [kN]*
A1	0	1.0	0.4
B1	0	1.2	0.9
C1	0	4.1	1.0
A2	0	0.5	4.2
B2	0	0.9	4.2

Table 5.3: Support forces in Pipestress Pipe Model

Support	*X-Force [kN]*	*Y-Force [kN]*	*Z-Force [kN]*
A1	0	3.4	3.7
B1	0	3.4	6.7
C1	0	12.4	5.5
A2	0	3.1	14.2
B2	0	3.7	15.0

Table 5.4: Support forces in Abaqus Building, Support & Pipe Model

It can be noticed that the Abaqus Building, Support & Pipe Model has in average 4.9 times greater support forces than the Pipestress Pipe Model. The difference cannot be connected to a specific support.

The supports in the top of the structure have the highest support forces even if the pipe diameter is smaller there. The reason is increased seismic loading higher up in the building.

6. Discussion

The eigenmodes of the building in the Abaqus Building Model (A.B.M.) are also found in the Abaqus Building, Support & Pipe Model (A.B.S.P.M.). The values and shapes are agreeing and therefore it can be concluded that the building system is independent of the pipe system and the pipe supports. According to ASCE, coupled analysis is required if the mass of a subsystem is bigger than 1% of its primary system, or if the subsystem change the response more than 10% [8, p. 15].

The eigenmodes as well as the support forces depend on a number of factors. By having the pipe system modelled in Abaqus in the same way as in Pipestress it is possible to compare the mode shapes. The mode shapes from the Abaqus Pipe Model (A.P.M.) agree with the A.B.S.P.M. In the A.P.M. only support stiffness is included, whereas the A.B.S.P.M. also accounts for mass distribution. The conclusion is that pipe support modelling technique has a minor impact on the lower eigenmodes of the pipe system.

The question of why the eigenfrequencies differs between the Abaqus and Pipestress still needs to be answered. A model in Pipestress is therefore made with perpendicular curves (P.P.M. bendless). Since there are only five bends and long straight pipelines, this is likely to have a small impact, as with different bend modelling in Abaqus. However when the bend radius decreases to zero (bendless) in Pipestress, the mode shapes and frequencies radically change and become similar to A.P.M. It is remarkable, that bends have that impact on the frequencies in Pipestress.

Comparisons (which are not presented in this thesis) between Abaqus and Pipestress with less number of bends resulted in smaller differences for the eigenfrequencies. It means that large pipe systems increase the difference between eigenfrequencies in Abaqus and Pipestress to the extent that some disappear.

The different eigenmodes may depend on the eigensolver or the way the bends are modelled in Pipestress. Four different ways to model the bends are done in Abaqus, which all result in similar values. Pipestress is created to agree with code and may therefore have a different element formulation of the bends.

The support forces are much bigger in the modal analysis in Abaqus than in the spectral analysis in Pipestress, which indicates that a Pipestress analysis can be non-conservative. Reason for this difference may depend on the way pipe bend elements are defined in the Pipestress model. Another explanation may be that Pipestress use Modal analysis and Abaqus Spectral analysis.

How big impact each eigenmode has on the force has not been presented. The first mode normally stands for the biggest part of the response. Modal participation factors can be calculated and may differ between the analyses in Abaqus and Pipestress.

The time history used in these analyses are not sufficient for design of a pipe system. Normally will three to five histories be sufficient. In a design response spectrum many time histories with different duration and shape are combined. One time history was used for this research but should be avoided in design.

7. Suggestions for Further Research

There are many ways to continue this research. Here are suggestions for topics:

- Analysis method's impact on pipe design
- Pipe support type's call for model detail
- Pipestress modelling method

The analysis method's impact on pipe design can be investigated by modelling the pipe system in Abaqus and perform both modal and spectral analysis. This research is then independent of differences in software. Both time histories and response spectra are needed as input and therefore a building in similarity to the one created in this report may be used.

The pipe support type's call for model detail can be investigated by expanding this analysis to more types of supports. It is interesting to find out if any type of support demand more model detail. This study can be based mainly on the natural frequencies and mode shapes.

The Pipestress modelling method could be further investigated. The difference in eigenmodes can be studied by looking at how the elements (i.e. bends) are modelled. The eigensolver used in Pipestress can also be examined.

Bibliography

[1] American Nuclear Society (1983). *ANSI/ANS-52.1: Nuclear Safety Criteria for the Design of Stationary Boiling Water Reactor Plants.* U.S.A.

[2] De svenska kärnkraftsverkens branschgemensamma beräkningsgrupp (1996). *Studie av styvhetskriterium för upphängningar i dynamiskt belastade rörsystem.* Sweden

[3] Burman D. & Ehrenborg C. (2009). *Dynamic behaviour of viscous dampers in a piping system.* Sweden

[4] Chopra, A. K. (2007). *Dynamics of Structures, Theory and Applications to Earthquake Engineering.* Pearson Education. University of California. U.S.A.

[5] Elnashai, A. S. & Di Sarno, L. (2008). *Fundamentals of earthquake engineering.* John Wiley & Sons. United Kingdom

[6] Ottosen, N. (1992). *Introduction to the finite element method.* Sweden

[7] Der Kiureghian, A. (1981). *A Response Spectrum Method for Random Vibration Analysis of MDOF Systems*, Earthquake Engineering & Structural Dynamics, Vol. 9, No. 5, p. 419-435.

[8] American Society of Civil Engineers (1998). *4-98, Sesmic Analysis of Safety-Related Nuclear Structures and Commentary.* U.S.A.

[9] Swedish Standards Institute (2009). *Eurocode 8 - Design of structures for earthquake resistance.* Sweden

[10] Swedish Nuclear Power Inspectorate (SKI) (1992). *Seismic safety - Characterization of seismic ground motions for probabilistic safety analyses of nuclear facilities in Sweden.* Sweden

Appendix - Figures

A.1 Abaqus Building Model

A.1.1 Eigenmodes

In this section are the 10 first eigenmodes with eigenfrequencies presented from the Abaqus Building Model.

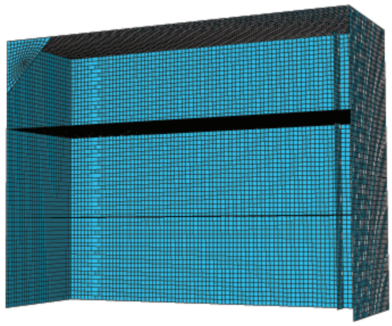


Figure A.1: Abaqus Building Model

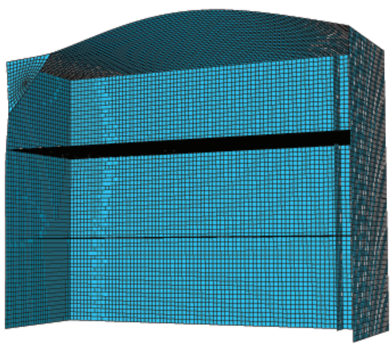


Figure A.2: First eigenmode at 6.83 Hz for the Abaqus Building Model

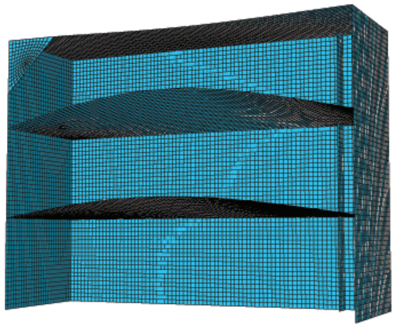


Figure A.3: Second eigenmode at 9.84 Hz for the Abaqus Building Model

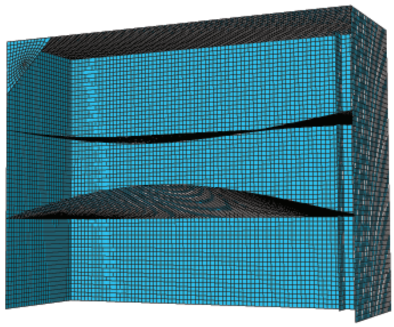


Figure A.4: Third eigenmode at 10.19 Hz for the Abaqus Building Model

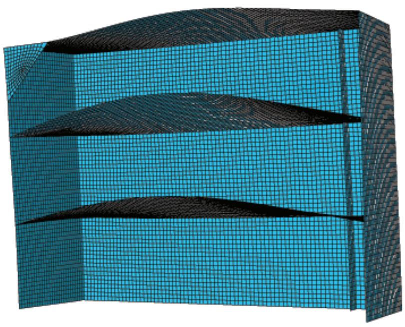


Figure A.5: Fourth eigenmode at 12.55 Hz for the Abaqus Building Model

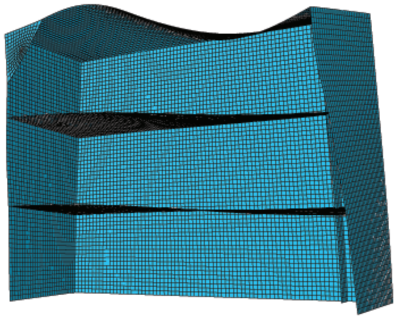


Figure A.6: Fifth eigenmode at 13.77 Hz for the Abaqus Building Model

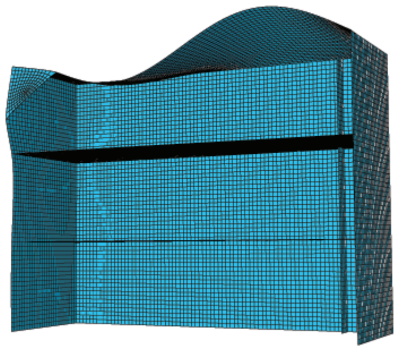


Figure A.7: Sixt eigenmode at 15.63 Hz for the Abaqus Building Model

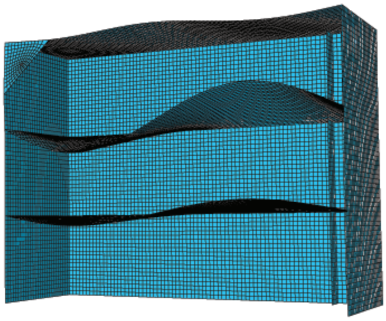


Figure A.8: Seventh eigenmode at 24.72 Hz for the Abaqus Building Model

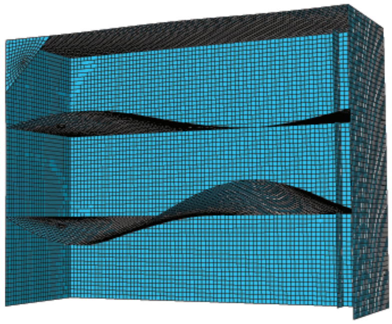


Figure A.9: Eight eigenmode at 25.08 Hz for the Abaqus Building Model

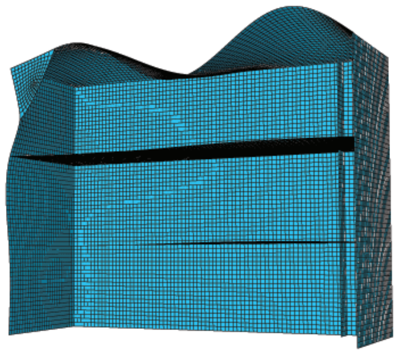


Figure A.10: Ninth eigenmode at 26.42 Hz for the Abaqus Building Model

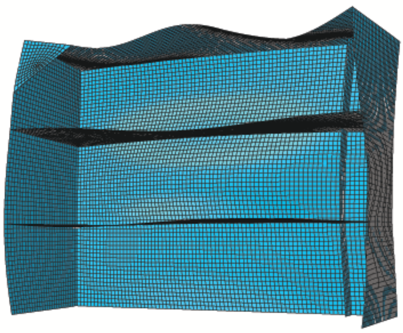


Figure A.11: Tenth eigenmode at 30.55 Hz for the Abaqus Building Model

A.1.2 Accelerations

In this section are the 24 response spectra presented that belongs to eight different locations with three directions each. The spectra are created in MatLab from acceleration time histories from the pipe support midpoints and the anchors of the Abaqus Building Model.

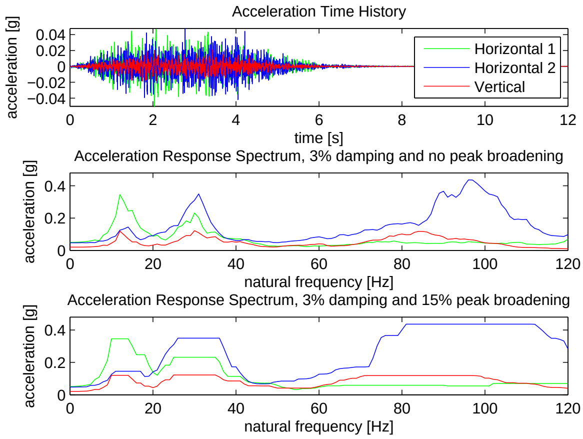


Figure A.12: Three acceleration diagrams for the three principle axis belonging to support A1. First the acceleration time history, then response spectra with no peak broadening and finaly response spectra with 15% peak broadening

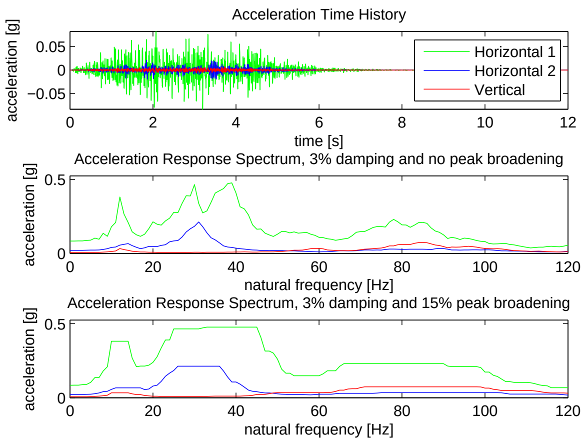


Figure A.13: Three acceleration diagrams for the three principle axis belonging to support B1. First the acceleration time history, then response spectra with no peak broadening and finaly response spectra with 15% peak broadening

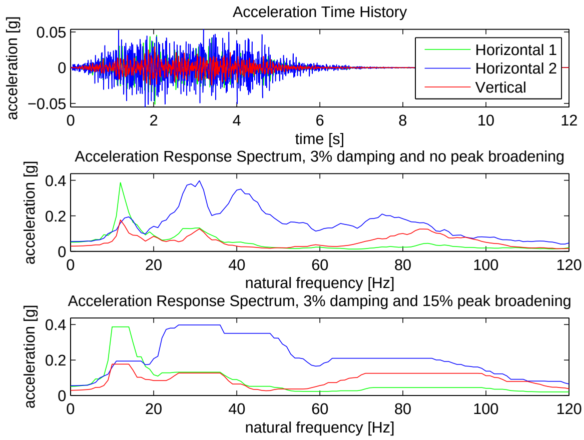


Figure A.14: Three acceleration diagrams for the three principle axis belonging to support C1. First the acceleration time history, then response spectra with no peak broadening and finaly response spectra with 15% peak broadening

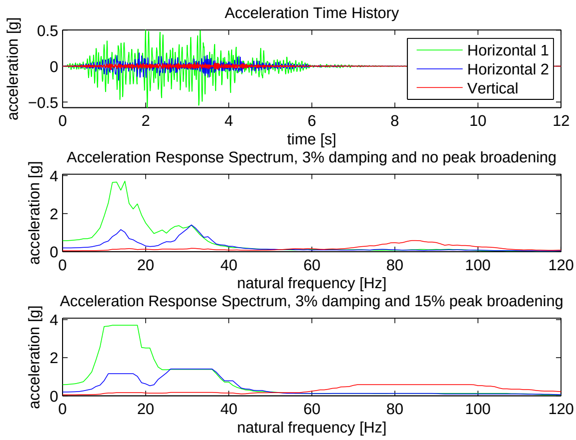


Figure A.15: Three acceleration diagrams for the three principle axis belonging to support A2. First the acceleration time history, then response spectra with no peak broadening and finaly response spectra with 15% peak broadening

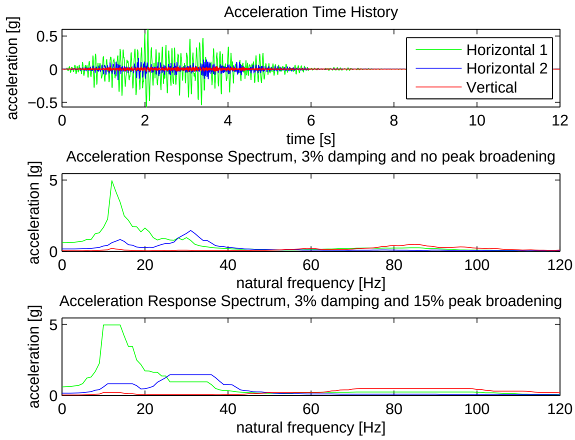


Figure A.16: Three acceleration diagrams for the three principle axis belonging to support B2. First the acceleration time history, then response spectra with no peak broadening and finaly response spectra with 15% peak broadening

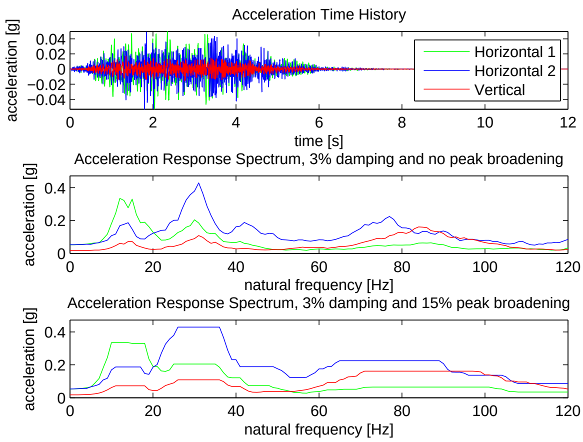


Figure A.17: Three acceleration diagrams for the three principle axis belonging to FIX1. First the acceleration time history, then response spectra with no peak broadening and finaly response spectra with 15% peak broadening

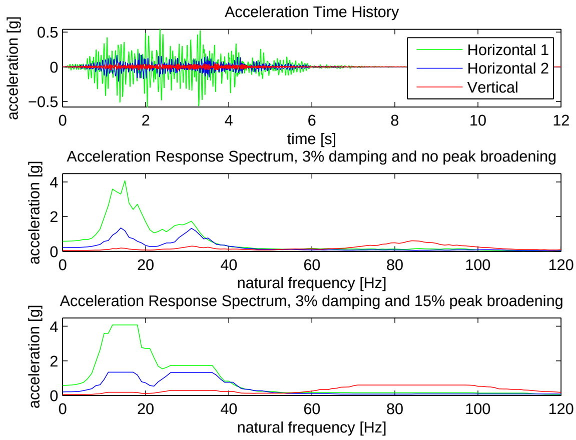


Figure A.18: Three acceleration diagrams for the three principle axis belonging to FIX2. First the acceleration time history, then response spectra with no peak broadening and finaly response spectra with 15% peak broadening

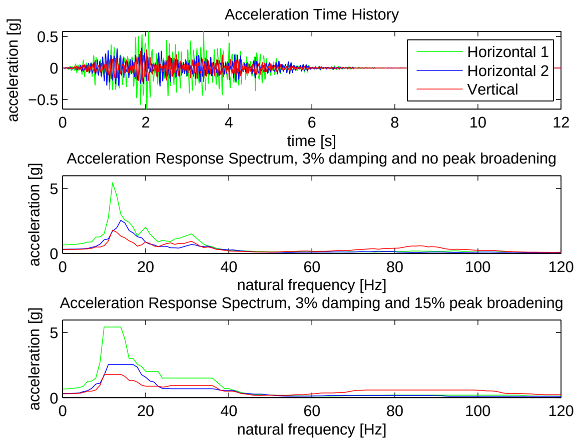


Figure A.19: Three acceleration diagrams for the three principle axis belonging to FIX3. First the acceleration time history, then response spectra with no peak broadening and finaly response spectra with 15% peak broadening

A.2 Pipestress Pipe Model

A.2.1 Eigenmodes

In this section are the seven first eigenmodes with eigenfrequencies presented from the Pipestress Pipe Model.

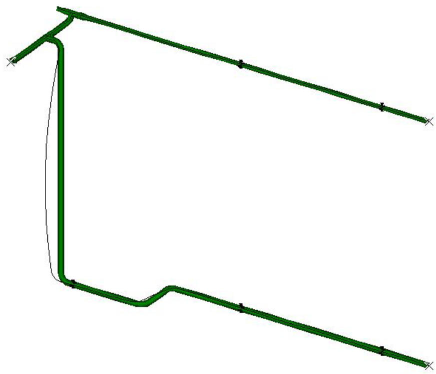


Figure A.20: First eigenmode at 5.34 Hz for the Pipestress Pipe Model

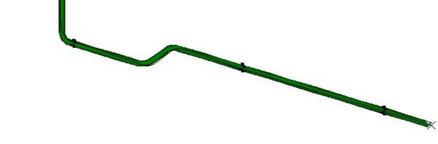


Figure A.21: Second eigenmode at 6.39 Hz for the Pipestress Pipe Model

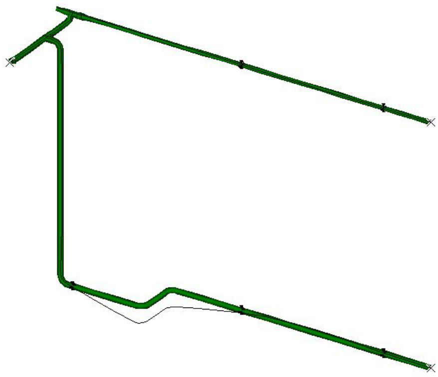


Figure A.22: Third eigenmode at 7.06 Hz for the Pipestress Pipe Model

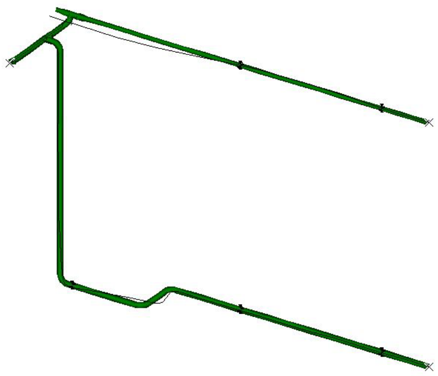


Figure A.23: Fourth eigenmode at 7.38 Hz for the Pipestress Pipe Model

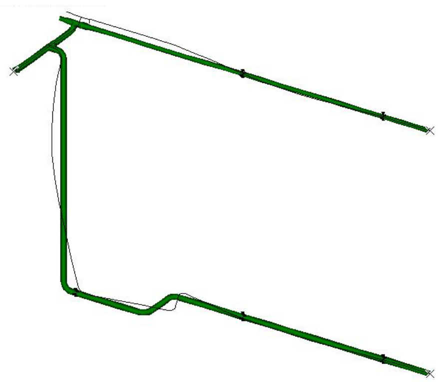


Figure A.24: Fifth eigenmode at 9.14 Hz for the Pipestress Pipe Model

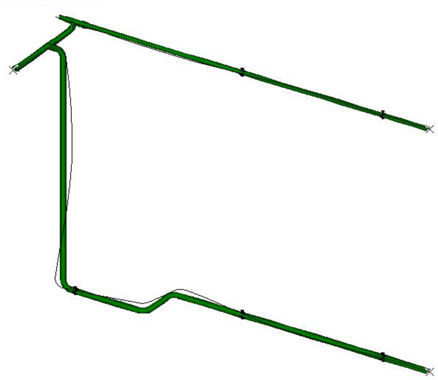


Figure A.25: Sixth eigenmode at 10.00 Hz for the Pipestress Pipe Model

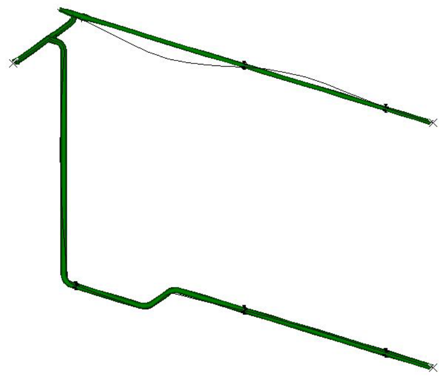


Figure A.26: Seventh eigenmode at 10.57 Hz for the Pipestress Pipe Model

A.3 Abaqus Building, Support & Pipe Model

A.3.1 Eigenmodes

In this section are the 10 first eigenmodes with eigenfrequencies presented from the Abaqus Building, Support & Pipe Model.

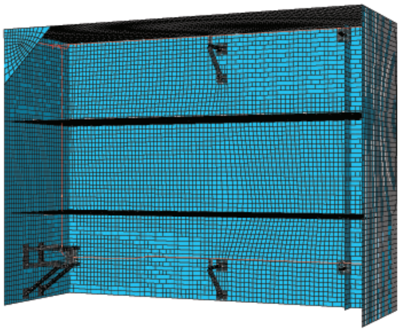


Figure A.27: Abaqus Building, Support & Pipe Model

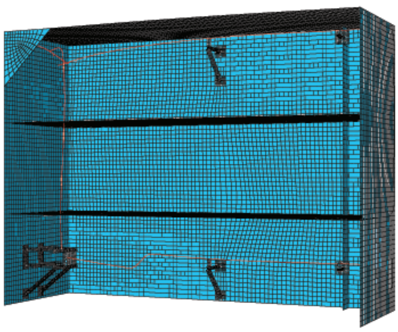


Figure A.28: First eigenmode at 6.57 Hz for the Abaqus Building, Support & Pipe Model

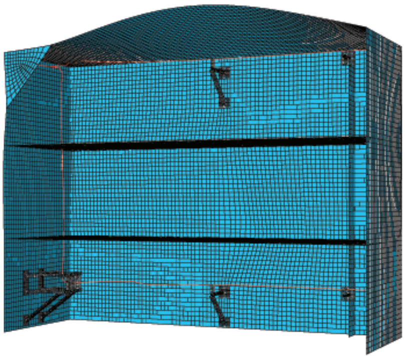


Figure A.29: Second eigenmode at 6.82 Hz for the Abaqus Building, Support & Pipe Model

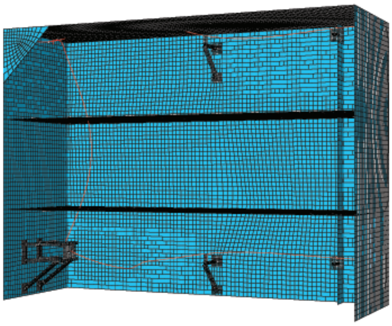


Figure A.30: Third eigenmode at 7.13 Hz for the Abaqus Building, Support & Pipe Model

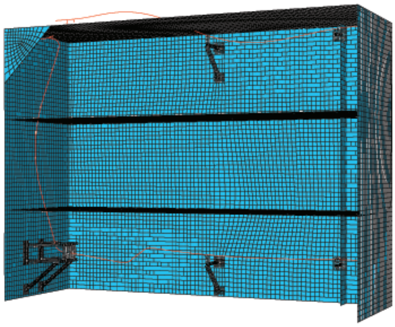


Figure A.31: Fourth eigenmode at 8.87 Hz for the Abaqus Building, Support & Pipe Model

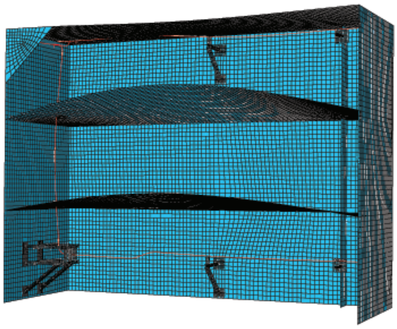


Figure A.32: Fifth eigenmode at 9.84 Hz for the Abaqus Building, Support & Pipe Model

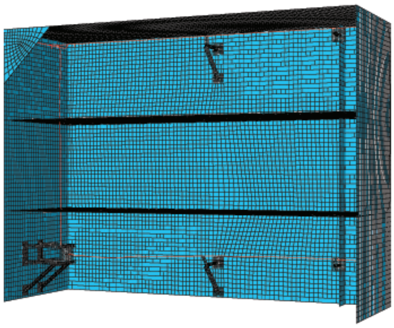


Figure A.33: Sixth eigenmode at 10.00 Hz for the Abaqus Building, Support & Pipe Model

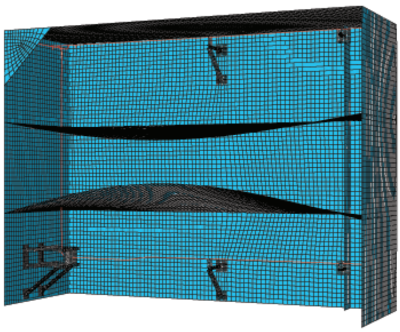


Figure A.34: Seventh eigenmode at 10.17 Hz for the Abaqus Building, Support & Pipe Model

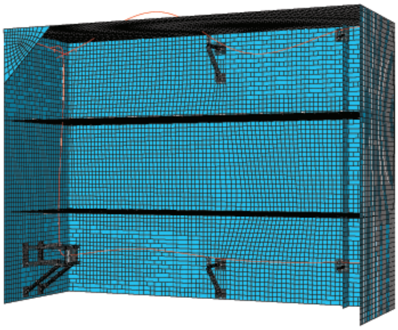


Figure A.35: Eight eigenmode at 11.46 Hz for the Abaqus Building, Support & Pipe Model

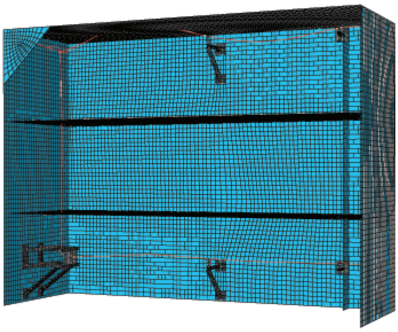


Figure A.36: Ninth eigenmode at 11.65 Hz for the Abaqus Building, Support & Pipe Model

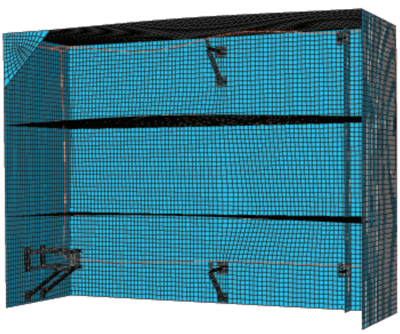


Figure A.37: Tenth eigenmode at 12.51 Hz for the Abaqus Building, Support & Pipe Model