Acknowledgement

This master thesis was carried out autumn 2011 at Epsilon Utvecklingscentrum Väst AB, Helsingborg, and the Division of Solid Mechanics at the University of Lund under supervision of M.Sc. Jimmy Carlsson at Epsilon and Ph.D. Student Anna Ask, Division of Solid Mechanics.

I wish to send a special thank to Jimmy Carlsson for his guidance and support, which has helped me during the project. I would also like to thank Magnus Johansson and Mats Kjellberg at Epsilon for having patience with my question and for their engagement during the project and Anna Ask for her feedback in the creation of the project report. I would also like to express my appreciation to Prof. Niels Saaybe Ottosen who has helped me with some derivations of the Timoshenko's beam theory in the theoretical part.

Last but not least I would like express my gratitude to my family for their support and encouragement during the project.

Helsingborg, December 2011

Henrik Andersson

Abstract

An uprating or renovation project in nuclear power plants normally requires a review of the safety analysis. A part of this analysis is the pipe stress analysis with the purpose to ensure the safe operation of piping systems by verifying their structural and pressure-retaining integrity.

Most piping analyses are today performed by use of modern calculation programs, CAE software, based on the finite element method. The results of the analyses are then evaluated against criterions in the ASME Boiler and Pressure Vessel Code Section III. The software used today, Pipestress, has for example the advantage with built in code evaluation but comes with limitations such as linear modeling of pipe supports during analyses of dynamic loads. Piping systems often include non-linear effects such as one-way motion supports or supports with directional stop and the size of the error introduced by the linear approximation is not fully known. A more general CAE software, ANSYS, also offering a piping module that contains several pipe elements. This software also allows modeling of non-linear pipe supports when performing dynamic analyses but this software has not the advantage with built in code evaluation according to ASME. However, ANSYS gives the opportunity to create and implement a code evaluation according to ASME through the scripting language APDL.

The main result of the project is a code evaluation tool to ANSYS created in APDL. The code permits evaluation of the equation in ASME that contains dynamic loads. This equation considers the internal pressure and resultant moments due to static and dynamic loads. The contributions shall then be multiplied with some stress indices that depends on the components in the system and their geometry and the requirement in the equation must be met in every nodal point in the piping system. Analyses with code evaluations have been performed on a test model in the both programs and it has been concluded that it is not possible to carry out the equation that contains dynamic loads entirely correctly according to ASME in the non-linear analysis. It has also been obtained that the same solution method and damping model is not feasible to use in the linear and non-linear analyses so the results from the analyses are not directly comparable. However, the results from the linear analyses in ANSYS seem to agree well with the results from the same loading situation in Pipestress, and it has therefore been concluded that the evaluation code seems correctly implemented.

Contents

1 Introduction

1.1 Presentation of Epsilon

Epsilon is one of Scandinavia's leading consulting companies in technology and system development with 1550 employees and more than 10,000 independent partners and specialist in various networks. Epsilon is working in industries such as energy, automotive, telecom, pharmaceutical, medical and other industries. Epsilon was founded in 1986 by Dan Olofsson and is today entirely owned by his company Danir AB. Epsilon is today one of Sweden's fastest growing consulting companies.

Calculation and simulation is one of Epsilon's cutting-edge areas. The company has a large number of analysis engineers with extensive international experience in industries such as automotive, aerospace, energy, steel and food. Within the industrial sector, Epsilon is today working towards all nuclear power plants in Sweden, where calculation standards and safety awareness is a part of the daily duties.

1.2 Background to the assignment

An uprating or renovation project in nuclear power plants normally requires a systematic review of the safety analysis. A part of this safety analysis is the pipe stress analysis. The purpose of pipe stress analysis is to ensure the safe operation of piping systems by verifying their structural and pressure-retaining integrity under the loading conditions postulated to occur during the lifetime of the piping in the nuclear power plant.

Most piping analyses are today performed by use of modern calculation programs, CAE software, based on the finite element method which is a numerical method widely used in modeling of mechanical applications. The results of the analyses are then evaluated against codes and standards. In Sweden, the general design criteria's is regulated by Strålsäkerhetsmyndigheten (SSM), and in detail in ASME Boiler and Pressure Vessel Code Section III. The code sets boundaries for the design.

A part of the pipe stress analysis is the dynamic numerical analysis. Typical dynamic loads in our piping systems are water hammers and global vibrations such as earthquake. To obtain reasonably correct boundary conditions the stiffness of pipe supports and other pipe connections is calculated and included in the piping model. The most common CAE software in Sweden for nuclear piping analyses is Pipestress. This program has its advantages with built in code evaluation, load applications and combinations based on design specifications in the nuclear industry. However, this software comes with limitations, such as linear modeling of pipe supports during analyses of dynamic loads. The size of the error introduced by this approximation is not fully known.

A more general program for structural mechanics based on the finite element method is ANSYS. This software is also capable to perform piping analyses and has no limitations in modeling of non-linear pipe supports under dynamic analyses. However, this software has for example not the advantage with built in code evaluation according to ASME.

1.3 Objective

The task in this master thesis is to develop an ASME piping code evaluation tool to ANSYS. The purpose is to explore the error introduced by the linear support approximation in Pipestress. Furthermore the assignment is to explore the capability and difference between the available pipe elements in ANSYS. The work can be described in the following tasks:

1. Consult the ASME code and investigate how Pipestress applies requirements within the Nuclear Piping Code ASME Boiler and Pressure Vessel Code Section III.
2. Explore the available pipe elements in ANSYS, formulation and capability.
3. Use ANSYS APDL to develop an ASME code evaluation tool.
4. Compare the results from ANSYS after the code evaluation with Pipestress for an identical piping system.
5. Introduce non-linear pipe supports in ANSYS and compare results with the linear approximation.

1.4 References to ASME

The code evaluation is based on rules and equations from ASME Boiler and Pressure Vessel Code Section III NC-3650 but due to copyright restrictions, this part has been excluded in the report. In cases when referring to some equations or figures including NC, these can be found in 2010 ASME Boiler and Pressure Vessel Code Section III Division 1 Subsection NC. However, the most necessary parts for the reader are given in the report.

2 ASME Boiler and Pressure Vessel Code

2.1 Background

The industrial revolution in the 1700s led to a growing use of steam boilers that generate steam with pressures above that of the atmosphere. The combination of very high pressures, faulty design of safety valves and insufficient inspections led to many boiler ruptures and explosions. As a result of this, a group of engineers met to review laws regarding the construction and operation of steam boilers. This meeting in 1911 was the basis for the formation of the American Society of Mechanical Engineers, ASME. This year, ASME set up a committee for the purpose of formulating standard rules for the construction of steam boilers and other pressure vessels. This committee is now called the Boiler and Pressure Vessel Committee.

The ASME Code committee expanded its area and today, the code covers the design and construction of power boilers, heating boilers, nuclear plant components and any pressure vessel which operates at a pressure of at least 15 psi[1].

For further details, the reader may consult Smith and Van Laan [1].

2.2 ASME Boiler and Pressure Vessel Code sections

The latest release of the ASME Boiler and Pressure Vessel Code at present is the 2010 edition and the code is currently divided into 12 sections.

The section that is relevant for this project is *Section III. Rules for Construction of Nuclear Power Plant Components.* This section provides requirements for the materials, design, fabrication, examination, testing, inspection, installation, certification, stamping, and overpressure protection of nuclear power plant components, and component and piping supports.

Section III is divided into several subsections and divisions. The code evaluation created in this project contains equations and rules from Division I, Subsection NC. This subsection contains requirements for the material, design, fabrication, examination, testing and overpressure protection of items which are intended to conform to the requirements for Class 2 construction. Class 2 components are those that are important to safety and designed for emergency core cooling, accident mitigation, containment heat removal, post accident fission product removal, and containment isolation in the nuclear power plant.

[1] 15 psi = 1.0342 bar

The rules of Subsection NC cover the requirements for assuring the structural integrity of piping components.

For further information, see http://www.asme.org/kb/standards/publications (20111228) [2].

2.3 Pipe stress analysis

The purpose of pipe stress analysis is as mentioned to ensure safe operation of piping system by verifying their structural and pressure-retaining integrity. This is accomplished by calculation of, and comparison to allowable values of variables such as stresses in the pipe wall and expansion movements. This verification is regulated by codes. Certain codes require more rigorous analyses than others, depending on the degree of hazard and consequences associated with failure. A common feature in the codes is the classification of loads into three load types which also applies in Subsection NC:

- Sustained loads, those due to forces present throughout normal operation, e.g. weight and pressure.
- Occasional loads, those due to forces present at rare intervals during operation, e.g. wind, seismic load, vibration, pipe rupture and relief valve discharge.
- Expansion loads, those due to displacements of the pipe, e.g. pipe thermal expansion, settlement, and differential anchor displacement due to seismic or thermal equipment movements.

2.3.1 Sustained loads

Sustained loads are classified as those loads caused by mechanical forces which are present throughout the normal operation of the piping system. These loads include both weight and pressure loadings.

All piping systems must be designed for weight loading. Most piping systems are usually not self-supporting, and therefore, they must be provided with supports to prevent collapse. The supports must be capable of holding the entire weight of the system, including that of the pipe, insulation, fluid, components, and the supports themselves.

2.3.2 Occasional loads

Loads which are applied to a system during only a small portion of the plant's operating life are usually classified as occasional loads.

Normally occasional loads will subject a piping system to horizontal loads as well as vertical loads, whereas sustained loads will normally be only vertical (weight). Dynamic loading is best resisted by rigid supports. However, the system flexibility must be sufficient to allow thermal growth. When thermal movements are too high to permit the use of rigid restraints, snubbers may be required. Snubbers act as rigid restraints when they are subjected to suddenly applied loads, such as vibrations, but snubbers do not resist static loads, such as weight and thermal loads.

Occasional loads include e.g. wind loads, relief valve discharge and seismic loads. Relief valves are used in piping systems to prevent that pressures builds up beyond that desired for safe operation. When the pressure setting is reached, the valve opens and allows sufficient fluid to escape from the piping system to lower the pressure. This permits a controlled discharge of fluid as a means of preventing pressure vessel ruptures. When a relief valve discharges, the fluid initiates a jet force which is transferred through the piping system and this is typically defined as a water hammer load. This force must be resisted by pipe supports if the pipe is not capable of resisting the load internally.

Safety-related piping in nuclear power plants in areas where earthquakes occurs must usually be designed to withstand seismic loads. Earthquake design criteria begin with an estimate of the earthquake potential in an area or region and this potential is partially based on the known history of previous earthquake activity in the area. Piping may be analyzed for seismic loadings through one of three methods: time history analysis, modal-response spectra analysis, or static analysis.

Time history analysis is based on a record of the postulated earthquake versus time. Data in the form of ground displacement, velocity, or acceleration is plotted for the duration of the estimated earthquake record. The time history analysis is quite accurate but is generally very expensive, since each time step requires a new calculation and therefore, the piping systems are often analyzed by modal analysis using response spectra.

Modal analysis breaks the dynamic model of the piping system into a number of modes of vibration. A more detailed description of this procedure is presented later on in the report. Response spectra are then generated to determine maximum response (acceleration, velocity, or displacement). The maximum response during the seismic event is then determined for each mode of vibration. These responses are combined to determine the total response of the system. Modal analysis is less expensive and time-consuming than time history analysis. In some cases, the dynamic load can also be approximated with an equivalent static load and the dynamic analysis is then replaced by a static analysis.

2.3.3 Expansion loads

Those loads due to displacements of the pipe, e.g. pipe thermal expansion and settlement are usually classified as expansion loads. As previously mentioned, the pipe will be more effectively restrained for deadweight and occasional loads when many restraints are added to the system. However, most pipes during operating, increases in temperature and expands. Piping which is too well restrained will not be able to expand and large forces will develop at the points of lockup, causing large stresses to develop in the pipe. The ideal restraint condition for thermal consideration is a total lack of restraint. Since this is not feasible, given other loads, some forces due to expansion will develop on restraint even in the most optimally supported system.

2.4 Service levels

Service levels are defined for nuclear safety-related piping by the ASME code, Section III. They are, in order of decreasing likelihood and increasing consequence of occurrence, levels A, B, C, and D, also known as normal, upset, emergency, and faulted, respectively. The piping codes regulate the stress levels permitted, but they do not define types of loading to be considered under each service level. The responsibility for determining the loadings to be considered under each service level rests with the plant owner. The definition of loading combinations would be recorded in the project specification or, in the case of a nuclear plant, in the safety analysis report.

2.4.1 Level A

Level A loadings refer to design conditions to which the piping system may be subjected during the performance of its specified service function. These would normally be loads due to operating pressure and weight.

2.4.2 Level B

Level B loadings include those occasional loadings which the piping system must withstand without suffering damage requiring repair. Usual examples of level B loadings would include fluid hammer and relief valve discharge. The codes allow increased stress levels for level B loadings, but not sufficient to allow damage to occur.

2.4.3 Level C

Level C loadings are normally those loadings associated with the design accidents of the plant. During level C loadings, the systems must be capable of performing their safety functions to safely shut down the plant. The piping components would be subject to inspection and repair, if necessary, before resumption of operation after a level C loading. The limits set by the codes for level C loadings are high enough to permit large deformation in areas of structural discontinuity.

2.4.4 Level D

Level D loadings are those associated with the most extreme accidents and less probable design conditions, such as a loss of coolant accident. A common level D loading in Sweden is safe shutdown earthquake (SSE), which is defined as the maximum earthquake postulated to occur at the plant site at any time. For this loading level, yet higher loads and resulting damage and deformations are postulated in the piping. As long as the components retain their ability to perform their safety function, an increase in piping stress is usually permitted.

For further information on loads and service levels, see Smith and Van Laan [1] and The American Society of Mechanical Engineers [3].

2.5 Analysis of piping designs according to NC-3650

The part of ASME Boiler and Pressure Vessel Code Section III that is relevant for the evaluation for class 2 components is NC-3650. This subsection treats the design of the complete piping system. NC-3650 is divided into the service limits Design Conditions, Level A and B, Level C and Level D. These service limits contains several equations and the definition of loading combinations and service levels decides which equations that should be carried out in the piping analysis. Common for these equations are that the contributions from the loadings shall be multiplied with some stress indices and the requirement in the equations must be met in every nodal point in the piping system. These stress indices are determined by the components and its geometry and are independent of the loading situation and service level.

The equation that contains dynamic loads in NC-3650 is (2.2). The main purpose of the project is to develop a code evaluation tool that is capable to evaluate dynamic loads with non-linear pipe supports. Due to this, (2.2) is the only equation that is relevant for this project. However, as soon will be described, there are also advantages to enable an evaluation of (2.1).

$$S_{SL} = B_1 \frac{PD_O}{2t_n} + B_2 \frac{M_A}{Z} \leq 1.5 S_h \tag{2.1}$$

$$S_{OL} = B_1 \frac{P_{max} D_0}{2t_n} + B_2 \left(\frac{M_A + M_B}{Z} \right) \leq 1.8 S_h \tag{2.2}$$

where

B_1, B_2 = primary stress indices for the specific product under investigation [FIG. NC-3673.2(b)-1]
D_0 = outside diameter of pipe
t_n = nominal wall thickness
Z = section modulus of pipe
S_h = material allowable stress at temperature consistent with the loading under consideration
M_A = resultant moment loading on cross section due to weight and other sustained loads
M_B = resultant moment loading on cross section due to occasional loads
P = internal Design Pressure
P_{max} = peak pressure

Dynamic analysis can be very time-consuming as mentioned earlier and because of this, (2.1) is also possible to evaluate using the created evaluation code. The advantage with this possibility is that the primary stress indices B_1 and B_2 quickly can be calculated and checked, and as mentioned previously, these indices only depend on the components and their geometry and not on the loading situation. Another advantage is that the moment term M_A in (2.1), also exists in (2.2). By evaluating (2.1), it becomes possible to quickly ensure that the model, loads and all the elements are correctly created or converted from Pipestress into ANSYS. After running the evaluation code in ANSYS, the obtained results should be comparable with the results from Pipestress provided identical models, loads and supports.

The primary stress indices B_1 and B_2 are defined in FIG. NC-3673.2(b)-1 for each component in the piping system. It appears that some components are not included, for example valves and flanges. These components are not included in the evaluation of the piping system but are treated separately in the component analysis and will not be considered in this project. The stress intensification factor i, also called SIF, is not used as a multiplication factor in (2.1) or (2.2). However, this factor is used in the process to calculate the stress indices B_1 and B_2 for some components, e.g. miter bends and reinforced fabricated tee, and is therefore calculated and saved for each component. This is also used as a preparation for further work.

3 Elements and supports

3.1 Modeling of the piping systems

Most piping systems can be described as irregular frames subjected to various loadings. The simplest method of estimating pipe stresses and support loads is to model the pipe as a beam. This is especially suitable in cases where the pipe travels in continuous horizontal runs, with a minimum of geometry changes.

3.2 Supports

Most piping systems are not self–supporting and must be provided with supports to prevent collapse. The supports must be capable of holding the entire weight of the system, including that of the pipe, insulation, fluid, components, and the supports themselves. The supports must also be flexible enough to permit thermal growth. The most commonly used support types is weight supports, snubbers, rigid restraints and anchors.

To obtain reasonaby correct boundary conditions the stiffness of the pipe supports is also included in the model and it is therefore of utmost importance to model the supports as they will behave in reality.

3.3 Nonlinearities

Many structural problems involve non-linear behavior. Non-linear response could be caused by characteristics of a system such as large deformations and strains, material behavior or the effect of contact or other boundary condition nonlinearities. In general, the nonlinearities in a system can be summarized as

- Geometric nonlinearities – those where the stiffness depends on the displacement. Geometric nonlinearities accounts for large deformations and phenomena such as snap-through behavior.
- Material nonlinearities – those due to the stress-strain response, i.e. the constitutive relation of the material. Material nonlinearities accounts for phenomena such as plasticity of materials.
- Boundary condition nonlinearities – those where the stiffness of the structure may change when two or more parts either contact or separate from initial contacts. Boundary condition nonlinearities characterize a specific class of geometrical nonlinearity.

3.3.1 Linear and non-linear supports

Piping systems often include non-linear effects such as directional stop, one-way motion and friction. These non-linear effects can be summarized as boundary condition nonlinearities. The material in the piping analyses in this project is linear and loaded in the elastic region, i.e. any material nonlinearity is not considered. Common non-linear restraints in a piping system are

- Support with an initial gap, i.e. a directional stop, which allow a piping system to move a certain amount in either direction before the support becomes active.
- One-way support, which act only in either positive or negative direction.

A linear support has a constant stiffness, i.e. irrespective of displacement as shown in Figure 3.1 a). Now consider the support shown in Figure 3.1 b). The slope of the curve, i.e. the stiffness, seems to be linear but with some initial gap. This support is a discontinuous support since the stiffness now depends on the displacement. This discontinuity gives rise to a nonlinearity in the structure and because of this, the total system needs to be treated as a non-linear system in the analysis.

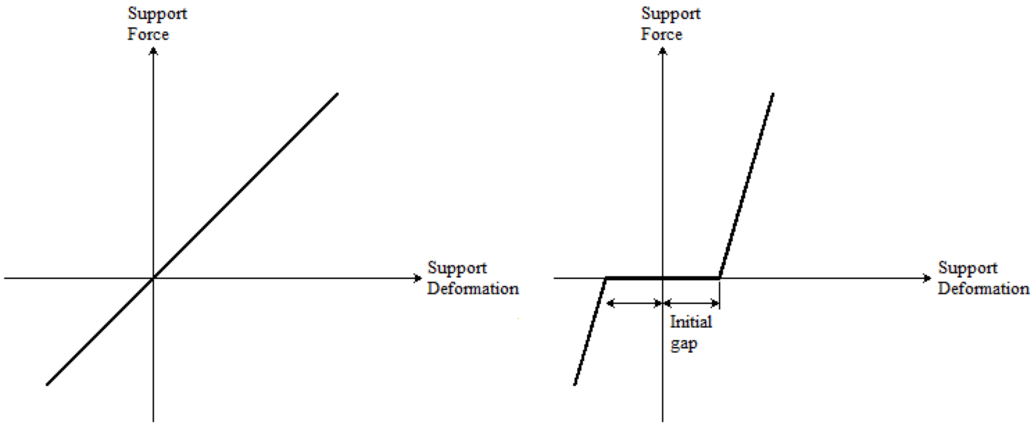


Figure 3.1: Support force as a function of support deformation. From left: a) Continuous (linear) support. b) Discontinuous (non-linear) support.

Another type of a discontinuous support, i.e. a non-linear supports, is the one-way support shown in Figure 3.2 b). A one-way support only acts in one direction and is also a discontinuous support since the response depends on the displacement.

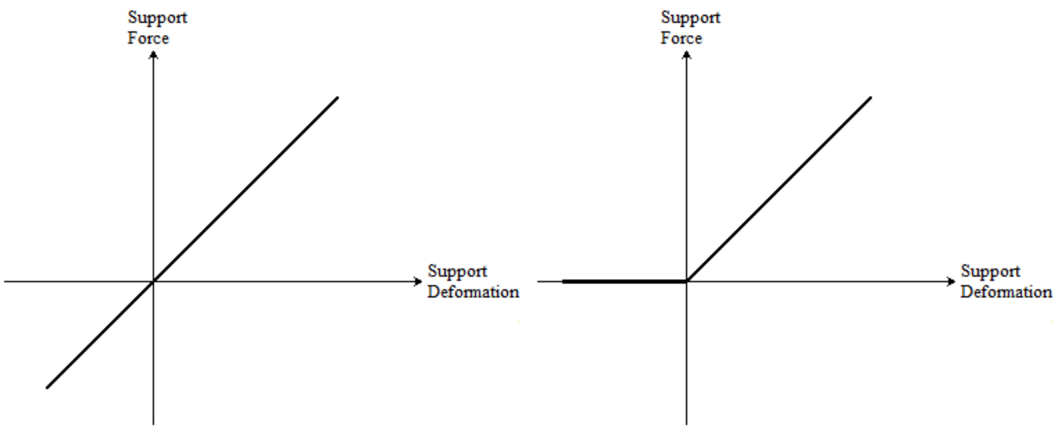


Figure 3.2: Support force as a function of support deformation. From left: a) Continuous (linear) support. b) Discontinuous (non-linear) support.

The main idea with a one-way support or a support with an initial gap is to permit thermal expansion, i.e. avoid the developing of stresses due to thermal expansion. One-way supports and supports with an initial gap exist in nuclear piping systems but these supports are assumed to behave linearly and are treated as continuous and linear supports when performing dynamic analyses in Pipestress.

4 Commercial CAE softwares and code implementation

4.1 Pipestress

The PepS package integrates Pipestress and the graphic user interface program Editpipe. Pipestress is a piping analysis program which is based on the finite element method. The program can be used to analyze chemical process piping, nuclear and conventional power generation piping systems to investigate compliance with piping codes and with other constraints on system response.

4.2 ANSYS

ANSYS is a more generally used program for structural mechanics based on the finite element method. This program also offers a piping module containing several pipe elements and has no limitations in modeling of non-linear pipe supports during dynamic analyses. However, ANSYS has not the advantage with load applications, combinations and code evaluation according to ASME.

4.2.1 Piping modeling in ANSYS

The simplest method of estimating pipe stresses and support loads is as recently mentioned to model the piping system with beam elements. For this purpose, ANSYS supplies for example the elements named PIPE16 and PIPE18.

PIPE16 and PIPE 18 are straight and curved pipes respectively. They are uniaxial elements with tension-compression, torsion and bending capabilities. The elements have six degrees of freedom at two nodes: translations in the nodal x, y, and z-directions and rotations about the nodal x, y, and z-axis. The element formulation is based on Euler-Bernoulli's beam theory.

The newer version of ANSYS (ANSYS 12.0 and further) recommends use of the newer available pipe elements, PIPE288 and ELBOW290 that will replace PIPE16 and PIPE18 respectively. PIPE 288 is based on Timoshenko's beam theory where shear-deformation effects are included. The curved pipe element ELBOW290 is based on a shell theory that not will be presented in this report.

A flexibility factor is used for some piping components according to FIG. NC-3673.2(b)-1. PIPE18 comes with the advantage that this flexibility factor is calculated by ANSYS according to FIG. NC-3673.2(b)-1 Welding elbow or pipe bend.

The flexibility factor is used to modify the element stiffness. The bending stiffness depends on Young's modulus E and on the moment of inertia I. For a hollow circle, i.e. a pipe, the moment of inertia is given by

$$I = \frac{\pi}{64}\left(D_0^4 - D_i^4\right) \tag{4.1}$$

where D_0 and D_i is the outside and inside diameter of the pipe respectively. With the flexibility factor introduced, (4.1) is then written as

$$I = \frac{\pi}{64}\left(D_0^4 - D_i^4\right)\frac{1}{C_f} \tag{4.2}$$

where C_f is the flexibility factor for the cross sectional and it yields that

$$C_f = \begin{cases} 1.0 & \text{if} \quad k = 1.0 \\ k & \text{if} \quad k > 1.0 \end{cases} \tag{4.3}$$

where k is calculated according to FIG. NC-3673.2(b)-1. For welding elbow or pipe bend, k is defined as

$$k = \frac{1.65r^2}{t_n R} \tag{4.4}$$

where

r = mean radius of pipe
t_n = nominal wall thickness of pipe
R = nominal bend radius of elbow or pipe bend

The substitution element ELBOW290 has not this advantage and this is one reason why the PIPE16 and PIPE18 are used instead of the newer recommended elements for the evaluation code. Another reason why the PIPE16 and PIPE18 elements are used is the possibilities to identify the element types that are used in the piping system, for example straight pipes, elbows, tee branches and reducers. The components shall be evaluated in various ways according to FIG. NC-3673.2(b)-1 and FIG. NC-3673.2(b)-2 and this needs to be considered. It turns out that when analyzing a piping system that is built on PIPE16 and PIPE18 elements, it is simple to identify all element types using APDL commands.

Modeling of the piping system and the loads can be done via commands in an input file. One example is the RUN command used to create a straight pipe while the commands REDUCE and TEE is creating a reducer and a branch element respectively. These commands also need inputs such as directions, lengths and nodes for the elements.

For further information about the piping modeling and the elements, the reader may consult ANSYS, Inc [4].

4.2.2 Introducing APDL

APDL stands for ANSYS Parametric Design Language, a scripting language that can be used to automate common tasks or even build up the model in terms of parameters (variables). APDL comes with a wide range of features such as repeating a command, macros, if-then-else branching, do-loops, and scalar, vector and matrix operations which is of great important for this purpose.

4.3 Code implementation according to ASME

The aim of this project is as stated to implement a code evaluation to ANSYS according to ASME using the APDL. Due to some difficulties in the work and the fact that Pipestress already is a capable to handle the most situations, some limitations are made in the work.

4.3.1 Limitations

NC-3650 includes several equations. The equations that shall be fulfilled are determined by the load combinations and the service levels. Since the purpose of the project is to enable the evaluation of dynamic loads with non-linear supports, (2.2) is the only equation in NC-3650 that is relevant to evaluate.

Equation (2.2) presented earlier is reproduced from the service level A and B in NC-3650 but also exists in the service levels C, and D. The difference in the equation between the various service levels are only the allowable stress ratio at the right hand side of the equation. As mentioned earlier, the service levels are defined in order of decreasing likelihood and increasing consequence of occurrence. An increasing service level admits higher allowable stress.

Equation (2.2) contains the stresses due to internal pressure and resultant moments in separated form due to weight and other sustained loads, and due to occasional loads. Equation (2.2) exists in two versions in NC-3650 and it is only one of them that is considered in this project. The alternative equation for piping fabricated in other defined materials is excluded from the code evaluation.

However, to obtain more realistic and correct results in the non-linear dynamic analysis, the loads may not be separated, i.e. the weight of the system needs to be included in the same transient analysis as the occasional loads. Equation (2.2) may instead be computed as

$$S_{OL} = B_1 \frac{P_{max} D_0}{2t_n} + B_2 \left(\frac{M_{AB}}{Z} \right) \leq 1.8 S_h \tag{4.5}$$

where

M_{AB} = resultant moment loading on cross section due to weight, other sustained loads and due to occasional loads

NC-3653.3 contains rules for determination of moments and section modulus. NC-3653.3(c) states that for intersections with reduced outlets where the mean radius of the branch pipe is less than half of the mean radius of the run pipe, the resultant moment for the branch leg may be calculated at the outside surface of the run pipe. This is not a requirement and is therefore not considered in the evaluation code. Figure 4.1 illustrate an intersection and the moment components. The intersection consists of two run pipes and one branch leg.

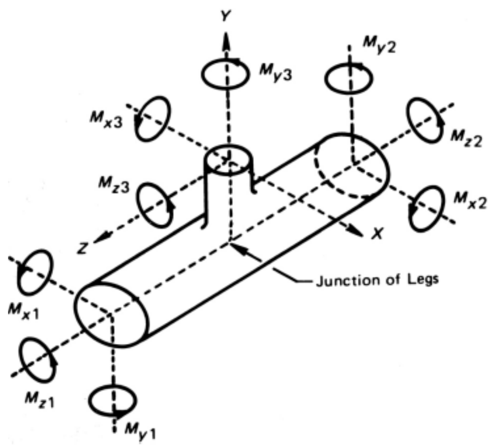


Figure 4.1: An intersection and the moment components.

All elements defined in FIG. NC-3673.2(b)-1 are available to evaluate with the created code to ANSYS, except for *threaded pipe joint or threaded flange* and *corrugated straight pipe or corrugated or creased bend.* These components are excluded since they rarely appear in Swedish nuclear power plants.

4.3.2 Programming schedule

The requirements in the equations given in NC-3650 shall be met in every nodal point in the piping system. The evaluation code requires information about the elements, loads and service level. When modeling the piping system and setting up the load cases in Editpipe, some necessary information is given that is not feasible to define in ANSYS when modeling the piping system. Because of that, some parameters and information have to be defined manually before running the evaluation code created to ANSYS. The flow diagram for the evaluation code is shown in Figure 4.2.

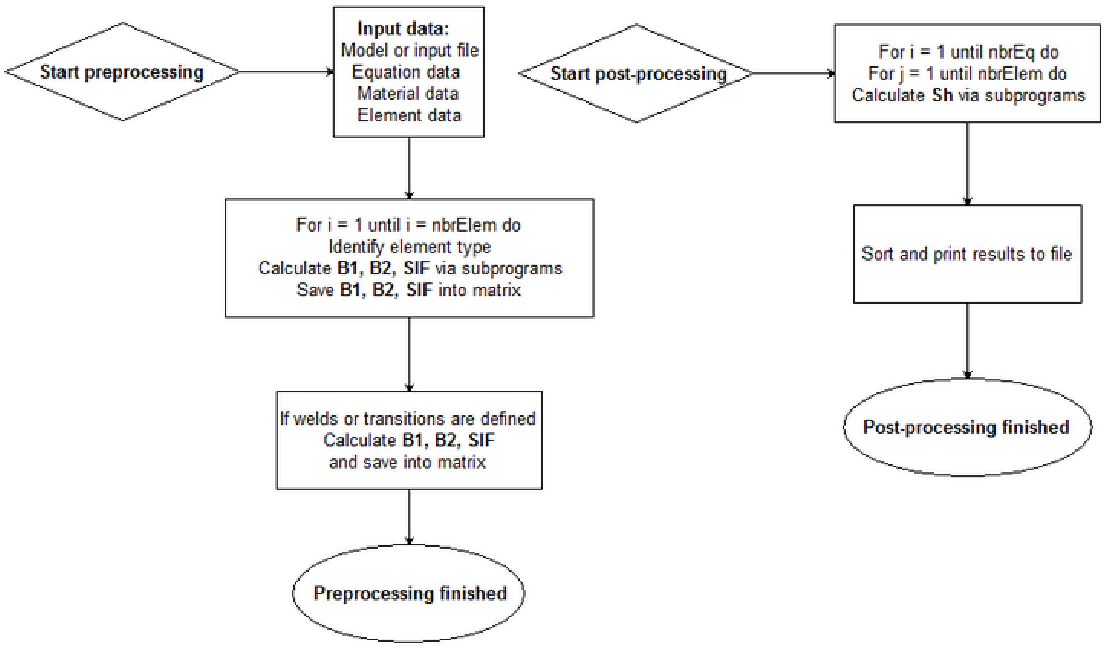


Figure 4.2: Flow diagram for the evaluation code.

The manual including all assumptions, restrictions and instructions to the created evaluation code is included in Appendix B.

5 FE formulation of beams

5.1 Beam theory

The finite element method is a numerical method used for finding approximate solutions of partial differential equations. Finite element formulations are in the most cases based on differential equations, kinematics and constitutive relations and the corresponding boundary conditions. In many cases such as beam bending, it becomes possible to introduce assumptions which will simplify the problem formulation. The two best known models for straight beams are based on the Euler-Bernoulli beam theory and the Timoshenko beam theory.

The aim of this chapter is to present the theoretical differences between the two beam elements used in ANSYS, i.e. PIPE16/PIPE18 and PIPE288. The presented theory will be restricted to the equilibrium conditions, kinematic and constitutive relations and a short static finite element formulation. For the interested reader the element stiffness for the two beam theories is derived in Appendix A for the simplest possible two dimensional beam element. The aim is to demonstrate some practical differences between the two theories but not to present all derivations needed for implementation in order to solve derived equations. For more information on beam theories and the FE-formulations, see Ottosen & Petersson [5], Timoshenko and Gere [6], and Fertis [7].

Consider an infinitely small part of a beam with the cross-sectional area $dA(x)$. The loading $q(x)$ is the force per unit length, i.e. $[N/m]$ and the material and cross-section of the beam are assumed to be symmetric about the xz-plane.

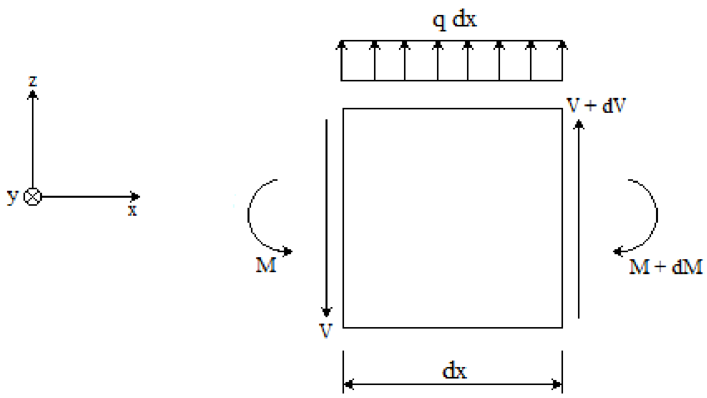


Figure 5.1: Infinitely small part of a beam.

The stress components normal to the x-axis give rise to a bending moment M, a vertical shear force V, and a normal force N defined by

$$M = \int_A z\sigma_{xx} dA \ ; \qquad V = \int_A \tau_{xz} dA \ ; \qquad N = \int_A \sigma_{xx} dA \tag{5.1}$$

where M is the moment about the y-axis. No resulting forces act in the x-direction according to Figure 5.1 so horizontal equilibrium implies that

$$N = 0 \tag{5.2}$$

Forces and moments caused by σ_{xx} and τ_{xz} are shown in Figure 5.1. Vertical equilibrium requires that

$$\frac{dV}{dx} = -q \tag{5.3}$$

and a moment equilibrium for the infinitely small part of the beam in Figure 5.1 requires that

$$\frac{dM}{dx} = V \tag{5.4}$$

since qdx and dV are infinitesimal.

5.1.1 Euler-Bernoulli theory

An essential assumption in beam theories is the kinematic behavior during deformation. The definition of Bernoulli's assumptions is that plane sections normal to the beam axis remain plane and normal to the beam axis during the deformation.

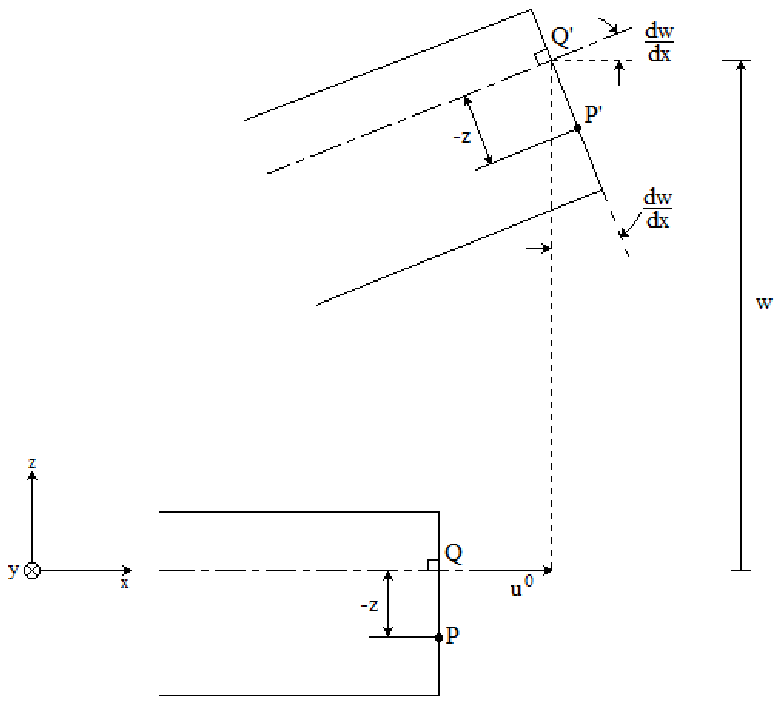


Figure 5.2: Deformation of Bernoulli beam.

The beam in Figure 5.2 is illustrated before and after the deformation with a plane section normal to the x-axis. Due to the deformation, the points P and Q move to the positions P' and Q'. Following Bernoulli's assumption, the plane defined by P'Q' is still normal to the beam axis after deformation. Since P is located below the x-axis, the distance between P and Q is given by $-z$ and since no shear deformation effects, the distance between P' and Q' is still given by $-z$. Assuming that the slope, i.e. the change in the beam axis rotation dw/dx is small, the displacements u_x and u_z in the x- and z-directions of point P are given by

$$u_x = u^0 - z\frac{dw}{dx} \ ; \qquad u_z = w \tag{5.5}$$

where u^0 is the displacement in the x-direction of the point Q and w is the deflection. It is also assumed that

$$u_y = 0 \tag{5.6}$$

and that u^0 and w only depend on x, i.e.

$$u^0 = u^0(x) \ ; \qquad w = w(x) \tag{5.7}$$

With the normal strain in the x-direction given by

$$\epsilon_{xx} = \frac{\partial u_x}{\partial x} \tag{5.8}$$

and the kinematic assumptions in (5.5), the normal strain now becomes

$$\epsilon_{xx} = \frac{du^0}{dx} - z\frac{d^2w}{dx^2} \tag{5.9}$$

Since u^0 only depends on x it becomes evident that the only non-zero strain component is ϵ_{xx}.

Linear elasticity for isotropic material and a uniaxial stress state is assumed. The constitutive relation between the stress and strain is then given by Hooke's law

$$\sigma_{xx} = E\epsilon_{xx} \tag{5.10}$$

Insertion of (5.9) into (5.10) and using (5.1 a), the bending moment M is determined from

$$M = \frac{du^0}{dx}\int_A EzdA - \frac{d^2w}{dx^2}\int_A Ez^2dA \tag{5.11}$$

If E is constant within the cross-section and the vertical location of the x-axis is chosen so that

$$\int_A zdA = 0$$

the bending moment in (5.11) then becomes

$$M = -EI\frac{d^2w}{dx^2} \ ; \ I = \int_A z^2dA \tag{5.12}$$

where I is the moment of inertia about the y-axis and the term EI is called the bending stiffness.

In the same manner the normal force N given by (5.1 c) becomes

$$N = \frac{du^0}{dx}\int_A EdA \tag{5.13}$$

but since $N = 0$ it holds that

$$\frac{du^0}{dx} = 0 \tag{5.14}$$

From (5.12) and (5.13) it is observed that in situations when $N \neq 0$, the bending moment M is controlled by the term d^2w/dx^2 and the normal force N by du^0/dx. The bending and elongation of the beam are therefore uncoupled phenomena and can be treated separately. The FE formulation of a beam can directly be combined with the FE formulation of an elastic bar in cases when $N \neq 0$.

Since the shear strain γ_{xz} is assumed to be zero, the shear force V can be eliminated from (5.3) and (5.4) to obtain

$$\frac{d^2M}{dx^2} + q = 0 \tag{5.15}$$

Use of (5.12) in (5.15) then results in

$$\frac{d^2}{dx^2}\left(EI\frac{d^2w}{dx^2}\right) - q = 0 \tag{5.16}$$

It appears that when the deflection w has been determined from this differential equation, all quantities of interest can be derived.

The kinematic assumption with neglected shear strain γ_{xz} in the Euler-Bernoulli beam theory is usually satisfactory for long slender beams where the ratio L/h $> 5 - 10$, where L is the beam length and h the beam height. For higher or shorter beams, theories which include the effect of a non-zero shear strain γ_{xz} can be used, for instance Timoshenko beam theory, presented in the upcoming chapter.

As in other finite element formulations, it is desirable to obtain the weak form of the equilibrium condition. The weak form of the differential equation given by (5.15) is obtained by multiplying by an arbitrary weight function $v(x)$ and integrating over the region. This results in

$$\int_a^b v\frac{d^2M}{dx^2}dx + \int_a^b vqdx = 0$$

assuming that the beam is extending from a to b.

Integration by parts then results in

$$\int_a^b \frac{d^2v}{dx^2}Mdx = \left[\frac{dv}{dx}M\right]_a^b - [vV]_a^b - \int_a^b vqdx \tag{5.17}$$

The moment M and the shear force V appears in the boundary terms of this weak form and they are natural boundary conditions.

The FE formulation can then be derived through the weak for of the equilibrium condition (5.17). Since the deflection w is the unknown function, an approximation for w can be written as

$$w = \mathbf{Na} \tag{5.18}$$

where

$$\mathbf{N} = \begin{bmatrix} N_1 & N_2 & \dots & N_n \end{bmatrix} \; ; \qquad \mathbf{a} = \begin{bmatrix} u_1 \\ u_2 \\ \vdots \\ u_n \end{bmatrix} \tag{5.19}$$

and n is the number of unknowns of the entire beam. From (5.18) it follows that

$$\frac{d^2w}{dx^2} = \mathbf{Ba} \quad \text{where} \quad \mathbf{B} = \frac{d^2N}{dx^2} \tag{5.20}$$

i.e

$$\mathbf{B} = \begin{bmatrix} \frac{d^2N_1}{dx^2} & \frac{d^2N_2}{dx^2} & \dots & \frac{d^2N_n}{dx^2} \end{bmatrix} \tag{5.21}$$

Using the Galerkin method for the arbitrary weight function v then gives that (5.17) can be written as

$$\int_a^b \mathbf{B}^T M dx = \left[\frac{d\mathbf{N}^T}{dx} M \right]_a^b - [\mathbf{N}^T V]_a^b - \int_a^b \mathbf{N}^T q dx \tag{5.22}$$

From (5.12) and (5.20) it follows that

$$M = -EI\mathbf{Ba} \tag{5.23}$$

and insertion into (5.22) then results in

$$\left(\int_a^b \mathbf{B}^T EI\mathbf{B} dx \right) \mathbf{a} = [\mathbf{N}^T V]_a^b - \left[\frac{d\mathbf{N}^T}{dx} M \right]_a^b + \int_a^b \mathbf{N}^T q dx \tag{5.24}$$

if assuming that E varies over the cross-section.

This FE formulation can be written in a more compact form and for one element, the following matrices are defined:

$$\mathbf{K}^e = \int_L \mathbf{B}^{eT} EI \mathbf{B}^e dx$$

$$\mathbf{f}_b^e = \left[\mathbf{N}^{eT} V\right]_L - \left[\frac{d\mathbf{N}^{eT}}{dx} M\right]_L \tag{5.25}$$

$$\mathbf{f}_l^e = \int_L \mathbf{N}^{eT} q dx$$

where L is the length of the beam, $\mathbf{K}^e$ is the element stiffness matrix, $\mathbf{a}^e$ the element displacement vector, $\mathbf{f}_b^e$ the element boundary vector and $\mathbf{f}_l^e$ the element load vector.

5.1.2 Timoshenko theory

In Euler-Bernoulli's beam theory, it has been concluded that shear deformations are neglected, and that plane sections remain plane and normal to the longitudinal axis. In the Timoshenko beam theory, plane sections still remain plane but are no longer normal to the longitudinal axis. The difference between the normal to the longitudinal axis and the plane section rotation is the shear deformation. These relations are shown in Figure 5.3.

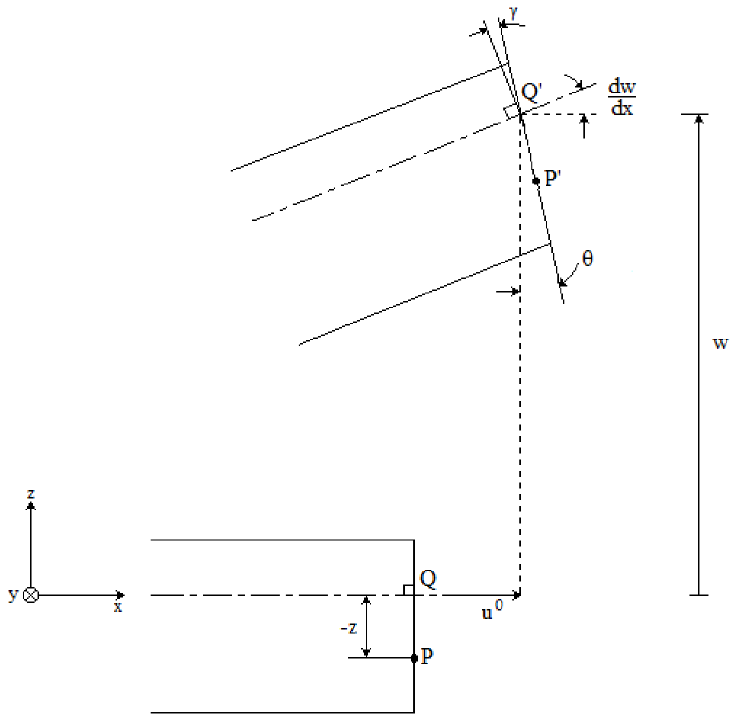


Figure 5.3: Deformation of Timoshenko beam.

In similarity to the Euler-Bernoulli beam theory and with consideration to Figure 5.3, the displacements u_x and u_z in the x- and z-directions of point P are given by

$$u_x = u^0 - z\theta; \qquad u_z = w \tag{5.26}$$

The normal strain in the x-direction is still given by (5.8) and now becomes

$$\epsilon_{xx} = \frac{du^0}{dx} - z\frac{d\theta}{dx} \tag{5.27}$$

It should also be observed that the shear strain now affects on the deformation and is according to Figure 5.3 given as

$$\gamma_{xz} = \gamma = \frac{dw}{dx} - \theta \tag{5.28}$$

The constitutive relation between the normal stress and the normal strain is given by (5.10). Timoshenko's beam theory also includes the effect of the transverse shear stress on the deformation and the relation between the shear stress τ_{xz} and the shear strain γ_{xz} also needs to be defined. Again a linear isotropic material is assumed and the relation is defined by

$$\tau_{xz} = G\gamma_{xz} \tag{5.29}$$

where G is the shear modulus.

The relation given by (5.27) inserted into (5.10) and with (5.28) inserted into (5.29) then gives

$$\sigma_{xx} = E\left(\frac{du_0}{dx} - z\frac{d\theta}{dx}\right) \tag{5.30}$$

and

$$\tau_{xz} = G\left(\frac{dw}{dx} - \theta\right) \tag{5.31}$$

From (5.30) it holds that the normal stress σ_{xx} varies linearly in z while the shear stress τ_{xz} in (5.31) is constant in z. The assumption with a constant shear stress is not correct. A correct distribution of the shear stress allows too become zero at the top and the bottom of the beam due to the equilibrium and a shear deflection constant K must be imposed which depends on the cross-section.

Equation (5.31) now becomes

$$\tau_{xz} = GK\left(\frac{dw}{dx} - \theta\right) \tag{5.32}$$

Considering the infinitely small part of a beam in Figure 5.1, and state that the expressions in (5.1) and (5.2) still holds since no kinematic assumptions are made. With (5.30) inserted into (5.1 a) the moment M then becomes

$$M = \int_A E\left(\frac{du_0}{dx} - z\frac{d\theta}{dx}\right) z dA \tag{5.33}$$

Following the same procedure as for the Euler-Bernoulli beam, the moment will then be written as

$$M = -EI\frac{d\theta}{dx} \tag{5.34}$$

Now it may be noted that the moment according to the Timoshenko beam theory is dependent on the change in the cross-section rotation $d\theta/dx$ and no longer to the change in the beam axis rotation dw/dx as in the Euler-Bernoulli beam theory. In the Timoshenko theory, the shear strain is not assumed to be zero and with (5.32) inserted into (5.1 b) and integrating over the cross-sectional area A, the vertical force then becomes

$$V = GAK\left(\frac{dw}{dx} - \theta\right) \tag{5.35}$$

Equation (5.34) and (5.35) inserted into the equilibrium equations given by (5.3) and (5.4) results in

$$EI\frac{d^2\theta}{dx} + GAK\left(\frac{dw}{dx} - \theta\right) = 0 \tag{5.36}$$

and

$$GAK\frac{d}{dx}\left(\frac{dw}{dx} - \theta\right) + q = 0 \tag{5.37}$$

After some calculations it holds that

$$EI\frac{d^3\theta}{dx^3} = q \tag{5.38}$$

and

$$EI\frac{d^4w}{dx^4} + \frac{EI}{GAK}\frac{d^2}{dx^2}q = q \tag{5.39}$$

Now it may be noted that if q is constant or linear, (5.39) then will result in

$$EI\frac{d^4w}{dx^4} = q \tag{5.40}$$

i.e. the same differential equation as for the Euler-Bernoulli theory stated in (5.16). However, in the Timoshenko beam theory the shear strain is not assumed to be zero and it is not possible to derive the shape functions by following the same manner as in the Euler-Bernoulli theory. The shape functions needs to be derived in another way but this procedure will not be presented in this project.

The difference between the Timoshenko and the Euler-Bernoulli beam theory is as presented that Timoshenko's beam theory includes the effect of the transverse shear stress on the deformation. These two models are also known as the shear-deformable and shear-indeformable model respectively.

In dynamic analysis, transverse shear can have a significant effect on higher modes and this is one reason why the Timoshenko beam theory can be used advantageously instead of the Euler-Bernoulli beam theory.

6 Dynamic effects and time integration

6.1 Dynamic loads and equation of motion

A dynamic load is a load that varies with time and the response must be determined with a dynamic analysis. A dynamic analysis can be performed to verify the response of the system due to resonance. Resonance occurs when the system's natural frequencies coincides with the frequency of the applied load.

A linear system can be described by the equation of motion defined as

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{C}\dot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{f}(t) \tag{6.1}$$

where $\ddot{\mathbf{u}}$ is the acceleration vector, $\dot{\mathbf{u}}$ the velocity vector and $\mathbf{u}$ the displacement vector. The structure is described by the mass matrix $\mathbf{M}$, the viscous damping matrix $\mathbf{C}$, and the stiffness matrix $\mathbf{K}$. The external load vector $\mathbf{f}(t)$ is in dynamic analyses assumed to be a given function of time.

6.2 Solution methods

The solution of the linear dynamic equilibrium equation (6.1) is in practical analysis obtained by two methods:

- Direct integration, where (6.1) are integrated using a numerical step-by-step procedure. No transformation of the equations into different form is carried out.
- Mode superposition, where the equilibrium equations are transformed into a form in which the step-by-step solution is less costly. In general the object is to obtain a good approximation to the actual exact response. In a mode superposition analysis only a few modes may need to be considered.

When damping is small, the resonant frequency is approximately equal to the natural frequency of the system. At these frequencies even a small external load can cause large amplitude oscillations. Due to this is it usually sufficient to restrict the mode superposition analysis to cover the frequencies of the external load, i.e. where resonance can occur.

Pipestress is using the mode superposition method while direct time integration is used by the linear and non-linear Newmark method in ANSYS. Therefore, the purpose of this chapter is to present the basis of the analysis methods that are used in the both programs.

6.2.1 Newmark's method for linear and non-linear systems

Newmark's method is a numerical integration method used to solve differential equations. In this project, this method is used in ANSYS in order to solve the equation of motion in (6.1).

The integration procedure is to compute the state vector $(\mathbf{u}_{n+1}, \dot{\mathbf{u}}_{n+1})$ at time $t_{n+1} = t_n + h$, given the state vector $(\mathbf{u}_n, \dot{\mathbf{u}}_n)$ at the previous time t_n and the external load vector at both these times, i.e. $\mathbf{f}_n$ and $\mathbf{f}_{n+1}$. The procedure can be established in two steps.

The first procedure is to express the increments $\mathbf{u}(t)$ and $\dot{\mathbf{u}}(t)$ in terms of integrals of the acceleration $\ddot{\mathbf{u}}$ over the time interval $[t_n, t_{n+1}]$. The state $(\mathbf{u}_{n+1}, \dot{\mathbf{u}}_{n+1})$ can be derived as

$$\begin{aligned} \dot{\mathbf{u}}_{n+1} &= \dot{\mathbf{u}}_n + \int_{t_n}^{t_{n+1}} \ddot{\mathbf{u}}(\tau) d\tau \\ \mathbf{u}_{n+1} &= \mathbf{u}_n + h\dot{\mathbf{u}}_n + \int_{t_n}^{t_{n+1}} (t_{n+1} - \tau)\, \ddot{\mathbf{u}}(\tau) d\tau \end{aligned} \tag{6.2}$$

where the time step $h = t_{n+1} - t_n$. The acceleration in (6.2) will not be known but the integrals can be evaluated approximately by

$$\begin{aligned} &\int_{t_n}^{t_{n+1}} \ddot{\mathbf{u}}(\tau) d\tau \simeq (1-\gamma) h\ddot{\mathbf{u}}_n + \gamma h \ddot{\mathbf{u}}_{n+1} \\ &\int_{t_n}^{t_{n+1}} (t_{n+1} - \tau)\, \ddot{\mathbf{u}}(\tau) d\tau \simeq \left(\tfrac{1}{2} - \beta\right) h^2 \ddot{\mathbf{u}}_n + \beta h^2 \ddot{\mathbf{u}}_{n+1} \end{aligned} \tag{6.3}$$

where the parameters γ and β should be chosen as $\gamma = \frac{1}{2}$ and $\beta = \frac{1}{4}$ for the most stable algorithm.

With (6.3) substituted into (6.2) the following relations are obtained between the displacement, velocity and acceleration vectors

$$\begin{aligned} \dot{\mathbf{u}}_{n+1} &= \dot{\mathbf{u}}_n + (1-\gamma) h\ddot{\mathbf{u}}_n + \gamma h \ddot{\mathbf{u}}_{n+1} \\ \mathbf{u}_{n+1} &= \mathbf{u}_n + h\dot{\mathbf{u}}_n + \left(\tfrac{1}{2} - \beta\right) h^2 \ddot{\mathbf{u}}_n + \beta h^2 \ddot{\mathbf{u}}_{n+1} \end{aligned} \tag{6.4}$$

The second procedure is to compute the acceleration by use of the equation of motion. Substitution of (6.4 a) and (6.4 b) into the linear equation of motion (6.1) gives

$$\begin{aligned}\left(\mathbf{M}+\gamma h\mathbf{C}+\beta h^2\mathbf{K}\right)\ddot{\mathbf{u}}_{n+1} &= \mathbf{f}_{n+1}-\mathbf{C}\left(\dot{\mathbf{u}}_n+(1-\gamma)h\ddot{\mathbf{u}}_n\right)\\ &\quad -\mathbf{K}\left(\mathbf{u}_n+h\dot{\mathbf{u}}_n+\left(\tfrac{1}{2}-\beta\right)h^2\ddot{\mathbf{u}}_n\right)\end{aligned} \tag{6.5}$$

where the acceleration vector $\ddot{\mathbf{u}}_{n+1}$ can be computed at time t_{n+1}. The velocity and displacement can then be computed by (6.4).

However, the computation of a time step in Newmark's algorithm is made by a prediction step followed by a correction step. In the linear analyses, this is of minor importance but in the non-linear analyses, the predictor acts as the starting point for iterations. Preliminary values of velocity $\dot{\mathbf{u}}^*_{n+1}$ and displacement $\mathbf{u}^*_{n+1}$ are evaluated from (6.4), without the as yet unknown acceleration $\ddot{\mathbf{u}}_{n+1}$:

$$\begin{aligned}\dot{\mathbf{u}}^*_{n+1} &= \dot{\mathbf{u}}_n+(1-\gamma)h\ddot{\mathbf{u}}_n\\ \mathbf{u}^*_{n+1} &= \mathbf{u}_n+h\dot{\mathbf{u}}_n+\left(\tfrac{1}{2}-\beta\right)h^2\ddot{\mathbf{u}}_n\end{aligned} \tag{6.6}$$

With these predicted values and by defining the modified mass matrix as

$$\mathbf{M}_* = \mathbf{M}+\gamma h\mathbf{C}+\beta h^2\mathbf{K} \tag{6.7}$$

the equation of motion (6.5) then takes the simplified form

$$\mathbf{M}_*\ddot{\mathbf{u}}_{n+1} = \mathbf{f}_{n+1}-\mathbf{C}\dot{\mathbf{u}}^*_{n+1}-\mathbf{K}\mathbf{u}^*_{n+1} \tag{6.8}$$

where the acceleration $\ddot{\mathbf{u}}_{n+1}$ can be computed. A correction step for the predicted velocity and displacement is then made by

$$\begin{aligned}\dot{\mathbf{u}}_{n+1} &= \dot{\mathbf{u}}^*_{n+1}+\gamma h\ddot{\mathbf{u}}_{n+1}\\ \mathbf{u}_{n+1} &= \mathbf{u}^*_{n+1}+\beta h^2\ddot{\mathbf{u}}_{n+1}\end{aligned} \tag{6.9}$$

Newmark's method can also be extended to non-linear dynamic analyses. This requires that Newton iterations must be performed at each time step in order to satisfy equilibrium. The idea of Newton iterations is briefly to linearize a function about a point.

The equation of motion in a non-linear system can be expressed as

$$\mathbf{M}\ddot{\mathbf{u}}_{n+1}+\mathbf{g}\left(\mathbf{u}_{n+1},\dot{\mathbf{u}}_{n+1}\right) = \mathbf{f}_{n+1} \tag{6.10}$$

where the solution at t_{n+1} is obtained by Newton iterations on the residual

$$\mathbf{r} = \mathbf{f}_{n+1} - \mathbf{M}\ddot{\mathbf{u}}_{n+1} - \mathbf{g}\left(\mathbf{u}_{n+1}, \dot{\mathbf{u}}_{n+1}\right) \tag{6.11}$$

i.e. the difference between the external and internal forces.

For non-linear systems it is necessary to assemble the tangential stiffness matrix and the structural damping matrix at the end of each time step and these matrices are defined as

$$\mathbf{K}_T = \partial\mathbf{g}/\partial\mathbf{u}\ ; \qquad \mathbf{C}_T = \partial\mathbf{g}/\partial\dot{\mathbf{u}} \tag{6.12}$$

The non-linear Newmark algorithm uses a modified tangent stiffness matrix defined as

$$\mathbf{K}_* = \mathbf{K}_T + \frac{\gamma}{\beta h}\mathbf{C}_T + \frac{1}{\beta h^2}\mathbf{M} \tag{6.13}$$

that is obtained as the the total derivative of the residual vector $\mathbf{r}$ with respect to the displacement vector $\mathbf{u}$. The residual is calculated in the iterations as

$$\mathbf{K}_*\delta\mathbf{u} = \mathbf{r} \tag{6.14}$$

where $\delta\mathbf{u}$ is the displacement increment in each iteration.

The linear and non-linear Newmark algorithm is summarized as pseudo code in Table 6.1 and for more information about Newmark's method, the reader may consult Krenk [8].

Table 6.1: The linear and non-linear Newmark algorithm in pseudo code.

Linear Newmark algorithm	Non-linear Newmark algorithm
System matrices $\mathbf{K}, \mathbf{C}, \mathbf{M}$ $\mathbf{M}_* = \mathbf{M} + \gamma h \mathbf{C} + \beta h^2 \mathbf{K}$ *Initial conditions* $\mathbf{u}_0, \dot{\mathbf{u}}_0$ $\ddot{\mathbf{u}}_0 = \mathbf{M}^{-1}(\mathbf{f}_0 - \mathbf{C}\dot{\mathbf{u}}_0 - \mathbf{K}\mathbf{u}_0)$ *for time step* $n = 1, 2, \ldots, n_{max}$ *Determine new external load* $\mathbf{f}_{n+1}$ *Prediction step:* $\dot{\mathbf{u}}^*_{n+1} = \dot{\mathbf{u}}_n + (1-\gamma) h \ddot{\mathbf{u}}_n$ $\mathbf{u}^*_{n+1} = \mathbf{u}_n + h\dot{\mathbf{u}}_n + \left(\frac{1}{2} - \beta\right) h^2 \ddot{\mathbf{u}}_n$ *Correction step:* $\ddot{\mathbf{u}}_{n+1} = \mathbf{M}_*^{-1}\left(\mathbf{f}_{n+1} - \mathbf{C}\dot{\mathbf{u}}^*_{n+1} - \mathbf{K}\mathbf{u}^*_{n+1}\right)$ $\dot{\mathbf{u}}_{n+1} = \dot{\mathbf{u}}^*_{n+1} + \gamma h \ddot{\mathbf{u}}_{n+1}$ $\mathbf{u}_{n+1} = \mathbf{u}^*_{n+1} + \beta h^2 \ddot{\mathbf{u}}_{n+1}$ *Accept quantities:* $\ddot{\mathbf{u}}_n = \ddot{\mathbf{u}}_{n+1}, \quad \dot{\mathbf{u}}_n = \dot{\mathbf{u}}_{n+1}, \quad \mathbf{u}_n = \mathbf{u}_{n+1}$ *end time step loop*	*Initial conditions* $\mathbf{u}_0, \dot{\mathbf{u}}_0$ $\ddot{\mathbf{u}}_0 = \mathbf{M}^{-1}(\mathbf{f}_0 - \mathbf{g}(\mathbf{u}_0, \dot{\mathbf{u}}_0))$ *for time step* $n = 1, 2, \ldots, n_{max}$ *Determine new external load* $\mathbf{f}_{n+1}$ *Prediction step:* $\ddot{\mathbf{u}}_{n+1} = \ddot{\mathbf{u}}_n$ $\dot{\mathbf{u}}_{n+1} = \dot{\mathbf{u}}_n + h\ddot{\mathbf{u}}_n$ $\mathbf{u}_{n+1} = \mathbf{u}_n + h\dot{\mathbf{u}}_n + \frac{1}{2} h^2 \ddot{\mathbf{u}}_n$ *Equilibrium iteration* $i = 1, 2, \ldots,$ *until* $\lvert\mathbf{r}\rvert_{norm} < epsilon$ *Residual calculation* $\mathbf{r} = \mathbf{f}_{n+1} - \mathbf{M}\ddot{\mathbf{u}}_{n+1} - \mathbf{g}(\mathbf{u}_{n+1}, \dot{\mathbf{u}}_{n+1})$ *Matrices and increment correction:* $\mathbf{K}_T = \partial \mathbf{g}/\partial \mathbf{u},\ \mathbf{C}_T = \partial \mathbf{g}/\partial \dot{\mathbf{u}}$ $\mathbf{K}_* = \mathbf{K}_T + \frac{\gamma}{\beta h}\mathbf{C}_T + \frac{1}{\beta h^2}\mathbf{M}$ $\delta\mathbf{u} = \mathbf{K}_*^{-1}\mathbf{r}$ $\mathbf{u}_{n+1} = \mathbf{u}_{n+1} + \delta\mathbf{u}$ $\dot{\mathbf{u}}_{n+1} = \dot{\mathbf{u}}_{n+1} + \frac{\gamma}{\beta h}\delta\mathbf{u}$ $\ddot{\mathbf{u}}_{n+1} = \ddot{\mathbf{u}}_{n+1} + \frac{1}{\beta h^2}\delta\mathbf{u}$ *end iteration loop* *Accept quantities:* $\ddot{\mathbf{u}}_n = \ddot{\mathbf{u}}_{n+1}, \quad \dot{\mathbf{u}}_n = \dot{\mathbf{u}}_{n+1}, \quad \mathbf{u}_n = \mathbf{u}_{n+1}$ *end time step loop*

6.2.2 Mode superposition method

The basic idea in the mode superposition method is the transformation of the equilibrium equations into a more effective form for solution, using

$$\mathbf{u} = \mathbf{P}\mathbf{x}(t) \tag{6.15}$$

where $\mathbf{u}$ is the displacements, $\mathbf{P}$ is an nxn nonsingular transformation matrix and $\mathbf{x}(t)$ is the generalized displacements.

Use of (6.15) into the equation of motion (6.1) and pre-multiplying of $\mathbf{P}^T$ gives

$$\tilde{\mathbf{M}}\ddot{\mathbf{x}} + \tilde{\mathbf{C}}\dot{\mathbf{x}} + \tilde{\mathbf{K}}\mathbf{x} = \tilde{\mathbf{f}} \tag{6.16}$$

where

$$\tilde{\mathbf{M}} = \mathbf{P}^T\mathbf{M}\mathbf{P} \;; \qquad \tilde{\mathbf{C}} = \mathbf{P}^T\mathbf{C}\mathbf{P} \;; \qquad \tilde{\mathbf{K}} = \mathbf{P}^T\mathbf{K}\mathbf{P} \;; \qquad \tilde{\mathbf{f}} = \mathbf{P}^T\mathbf{f} \tag{6.17}$$

The objective is to come up with a $\mathbf{P}$-matrix which when used in the transformation gives a $\tilde{\mathbf{M}}$-, $\tilde{\mathbf{C}}$- and $\tilde{\mathbf{K}}$-matrix that are all diagonal. An effective transformation matrix $\mathbf{P}$ is established using the displacement solutions of the free vibration equilibrium equation. With damping neglected and without forces applied, the free vibration equilibrium equation becomes

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{0} \tag{6.18}$$

An assumed solution to (6.18) is

$$\mathbf{u} = \boldsymbol{\phi} \sin \omega \left(t - t_0\right) \tag{6.19}$$

where ω is the natural frequency and $\boldsymbol{\phi}$ the mode shape vector. With (6.19) inserted into (6.18), the generalized eigenproblem is then obtained as

$$\mathbf{K}\boldsymbol{\phi} = \omega^2\mathbf{M}\boldsymbol{\phi} \tag{6.20}$$

with the n eigensolutions $(\omega_1^2, \phi_1), (\omega_2^2, \phi_2), \ldots, (\omega_n^2, \phi_n)$. Each of these eigenpairs satisfies (6.20) and it also holds that

$$\boldsymbol{\phi}_i^T\mathbf{M}\boldsymbol{\phi}_j = \begin{cases} 1 & \text{if} \quad i = j \\ 0 & \text{if} \quad i \neq j \end{cases} \tag{6.21}$$

Defining

$$\boldsymbol{\Phi} = \left[\begin{array}{cccc} \boldsymbol{\phi}_1 & \boldsymbol{\phi}_2 & \ldots & \boldsymbol{\phi}_n \end{array}\right] \;; \qquad \boldsymbol{\Omega}^2 = \begin{bmatrix} \omega_1^2 & & & \\ & \omega_2^2 & & \\ & & \ddots & \\ & & & \omega_n^2 \end{bmatrix} \tag{6.22}$$

the eigenproblem in (6.20) can then be written in the following form

$$\mathbf{K\Phi} = \mathbf{M\Phi\Omega}^2 \tag{6.23}$$

From (6.21) and (6.23) it follows that

$$\mathbf{\Phi}^T\mathbf{M\Phi} = \mathbf{I}\ ; \qquad \mathbf{\Phi}^T\mathbf{K\Phi} = \mathbf{\Omega}^2 \tag{6.24}$$

The objective is as mentioned to come up with a $\mathbf{P}$-matrix which when used in the transformation gives a $\tilde{\mathbf{M}}$-, $\tilde{\mathbf{C}}$- and $\tilde{\mathbf{K}}$-matrix that are all diagonal. From (6.24) it turns out that $\mathbf{\Phi}$ is such a matrix and with $\mathbf{\Phi} = \mathbf{P}$, (6.15) then becomes

$$\mathbf{u} = \mathbf{\Phi x}(t) \tag{6.25}$$

where $\mathbf{\Phi}$ stores the eigenvectors of the generalized eigenproblem. Use of (6.25) into the equation of motion (6.1) and pre-multiplying of $\mathbf{\Phi}^T$ gives

$$\ddot{\mathbf{x}}(t) + \mathbf{\Phi}^T\mathbf{C\Phi}\dot{\mathbf{x}}(t) + \mathbf{\Omega}^2\mathbf{x}(t) = \mathbf{\Phi}^T\mathbf{f}(t) \tag{6.26}$$

If the damping is proportional it holds that

$$\boldsymbol{\phi}_i^T\mathbf{C}\boldsymbol{\phi}_j = 2\omega_i\xi_i\delta_{ij} \tag{6.27}$$

where δ_{ij} is the Kronecker delta which takes the value of 1 if $i = j$, otherwise 0. ξ_i is a modal damping parameter for mode i called the damping ratio.

For n individual equations (6.26) then takes the form

$$\ddot{x}_i(t) + 2\omega_i\xi_i\dot{x}_i(t) + \omega_i^2x_i(t) = f_i(t) \tag{6.28}$$

where

$$f_i(t) = \boldsymbol{\phi}_i^T\mathbf{f}(t) \tag{6.29}$$

If solving the uncoupled equations in (6.28) with for example the Newmark method for all n equations, the exact same solution will be obtained as for the coupled equations in (6.1) provided use of same damping parameters and same time steps.

The effect of the mode superposition method is that not all of the n uncoupled equations need to be solved and in general it is not necessary to solve them all. In mode superposition analysis, the only frequencies of the system that needs to be considered are those that are truly excited by the dynamic loads.

If the n equations are reduced to p uncoupled equations, the total dynamic response is then obtained by adding the contributions from all p modes as

$$\mathbf{u}^p = \sum_{i=1}^{p} \boldsymbol{\phi}_i x_i(t) \tag{6.30}$$

where $\mathbf{u}^p$ is the approximated solution i.e.

$$\mathbf{u}^p \doteq \mathbf{u} \tag{6.31}$$

For further details in the mode superposition method, the reader may consult Kirsch [9].

6.3 Damping models

Determining a structure's true physical damping is very complex. To obtain a simplified model it is therefore often assumed that the damping of a structure can be represented by viscous damping. This damping shall correspond to the energy losses due to e.g. friction and gaps in the piping system. In this project, the damping models that are used are modal damping and Rayleigh damping. Pipestress uses modal damping in the mode superposition method, and Rayleigh damping is used in the direct integration in ANSYS. Some inconvenience has been obtained in the comparison of the results of the analyses after using the two different damping models and solution methods. Therefore, the two damping models and the obstacles are presented below. For further details, the reader may consult Buchholdt [10] and Craig Jr. and Kurdila [11].

6.3.1 Modal damping

Modal damping is the damping typically specified in seismic analysis codes and standards. The modal damping is defined by the damping ratio

$$\xi = \frac{C}{C_c} \tag{6.32}$$

where C_c is the critical damping. A typical value of the damping ratio ξ to be assumed for piping systems are 2 – 10%. This damping ratio is used in the uncoupled equations given by (6.28).

In the discussion of mode superposition it was concluded that the main idea of the method was to come up with a set of uncoupled equations. In the case of non-linear structures this form of transformation is not permissible, because for such structures the natural frequencies and mode shapes vary with each incremental/iterative modification of the tangent stiffness matrix.

Since the modal damping only is feasible to use when solving the dynamic equilibrium equations using the mode superposition method, an alternative damping model needs to be used in the direct integration method.

6.3.2 Rayleigh damping

The traditional damping model used in direct integration algorithms is Rayleigh damping defined by

$$\mathbf{C} = \alpha\mathbf{M} + \beta\mathbf{K} \tag{6.33}$$

which is proportional to a linear combination of the mass matrix $\mathbf{M}$ and the stiffness matrix $\mathbf{K}$. Since these matrices already are established for the system, the only unknown parameters to determine in order to form the damping matrix $\mathbf{C}$ is α and β which can be obtained by

$$\alpha = \xi\frac{\omega_i\omega_j}{\omega_i + \omega_j}\ ; \qquad \beta = \frac{2\xi}{\omega_i + \omega_j} \tag{6.34}$$

where

α = uniform mass damping multiplier
β = uniform stiffness damping multiplier
ξ = constant damping ratio
ω_i = natural circular frequency of mode i
ω_j = natural circular frequency of mode j

The difficult part is to determine the α and β parameters that depends on the frequencies ω_i and ω_j of the two modes i and j when the damping ratio ξ is constant. Modal damping is constant for all frequencies while Rayleigh damping varies with the frequencies. A conservative approach would be to enforce a Rayleigh damping curve that matches a prescribed modal damping for the highest and lowest modes of the structure. However, to obtain a closely comparable Rayleigh damping to modal damping is not such a simple assumption and this will not be considered in this project.

7 Results

7.1 Test model

The main result of this project is the evaluation code and the accompanying manual. The manual is included in Appendix B but the code is excluded from the report. Some analyses are done on a test model in Pipestress and ANSYS in order to verify the code and investigate the influence of non-linear pipe supports.

The test model and the node numbering are shown in Figure 7.1 and Figure 7.2. The system is constrained in all degrees of freedom in node 1, 26, 14 and 2020 in the both models. The model contains straight pipes, bends, branch connections, reducers, valves and flanges. The supports numbered according to Figure 7.2 are listed in Table 7.1. The dynamic load in the system is a water hammer load applied with a amplitude and frequency illustrated in Figure 7.5 - 7.7, with the node numbering according to the model in Figure 7.1.

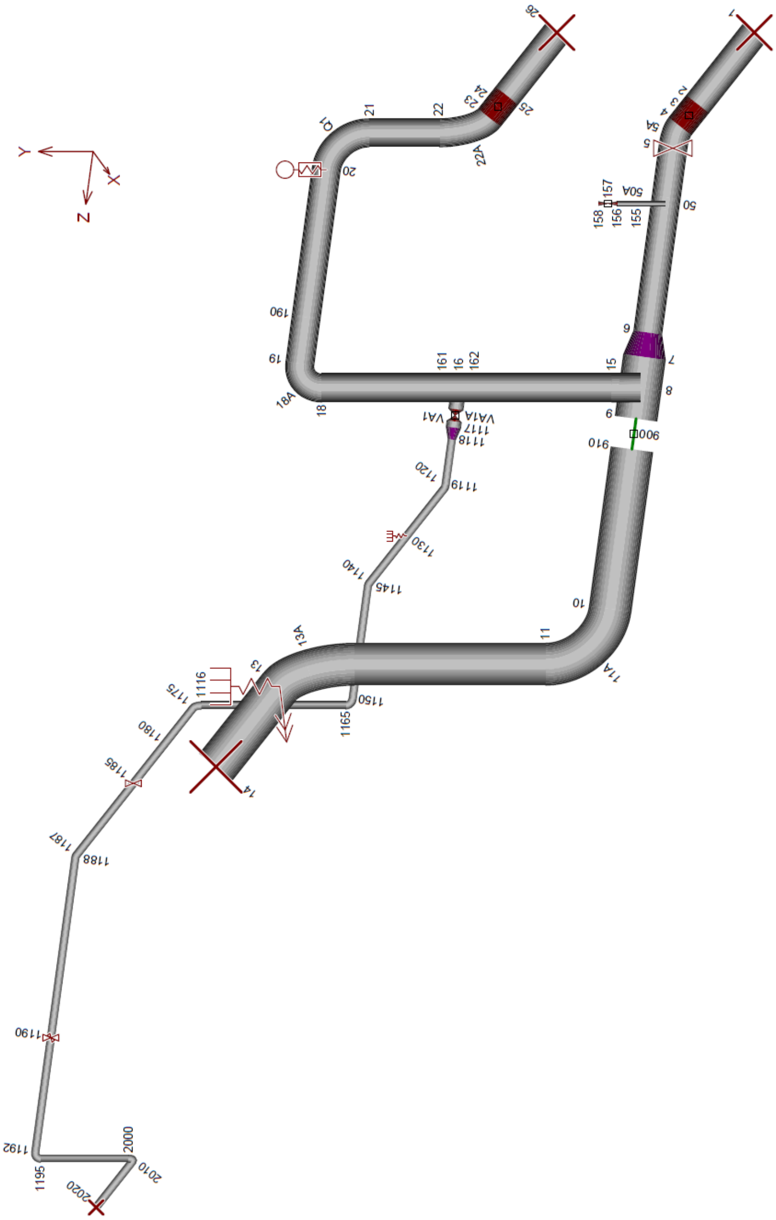


Figure 7.1: Test model in Pipestress.

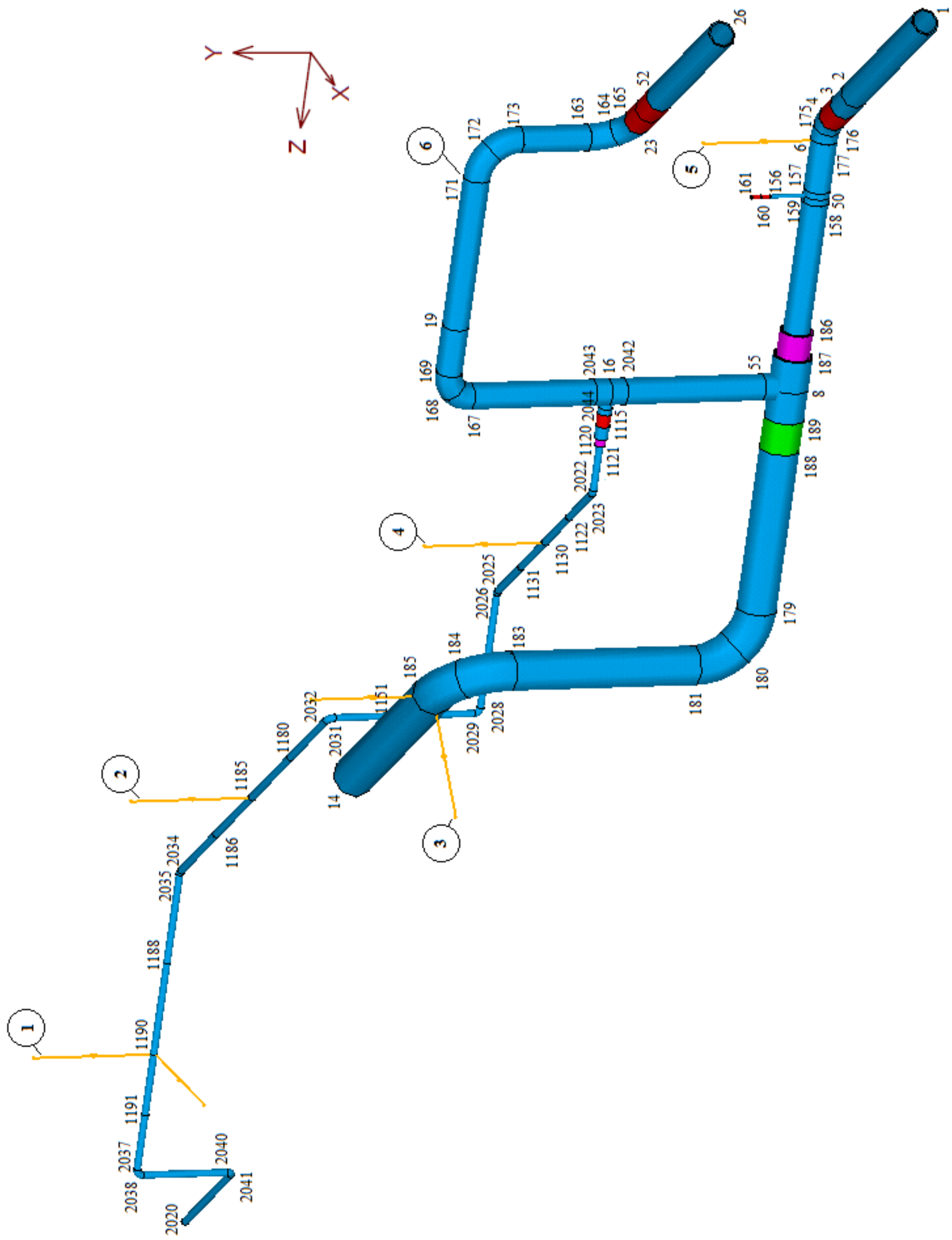


Figure 7.2: Test model in ANSYS.

Table 7.1: Support list.

Support number	Support type	Support direction		
		x	y	z
1	Guide support	1	1	
2	Guide support		1	
3	Constant spring		1	
3	Rotational support	0.707		0.707
4	Constant spring		1	
5	Guide support		1	
6	Constant force		1	

For the dynamic analyses, the Rayleigh damping parameters α and β are calculated with respect to the frequency content of the applied water hammer load. The dynamic loads are illustrated in Figure 7.5 – 7.7 and vary with time and in the frequency range 2-10 Hz. The parameters α and β are calculated through (6.34) with the natural frequencies $\omega_i = 4\pi$ rad/s and $\omega_j = 20\pi$ rad/s and with the damping ratio $\xi = 0.05$, obtained to match the results using the mode superposition method with 5 % modal damping.

Support number 1, 2 and 5 are modified in the non-linear analysis. Support number 1 is given an initial gap in the x- and y-direction as shown in Figure 7.3. The guide supports number 2 and 5 are modified to only act in the negative y-direction, i.e. zero stiffness in the positive y-direction as shown in Figure 7.4.

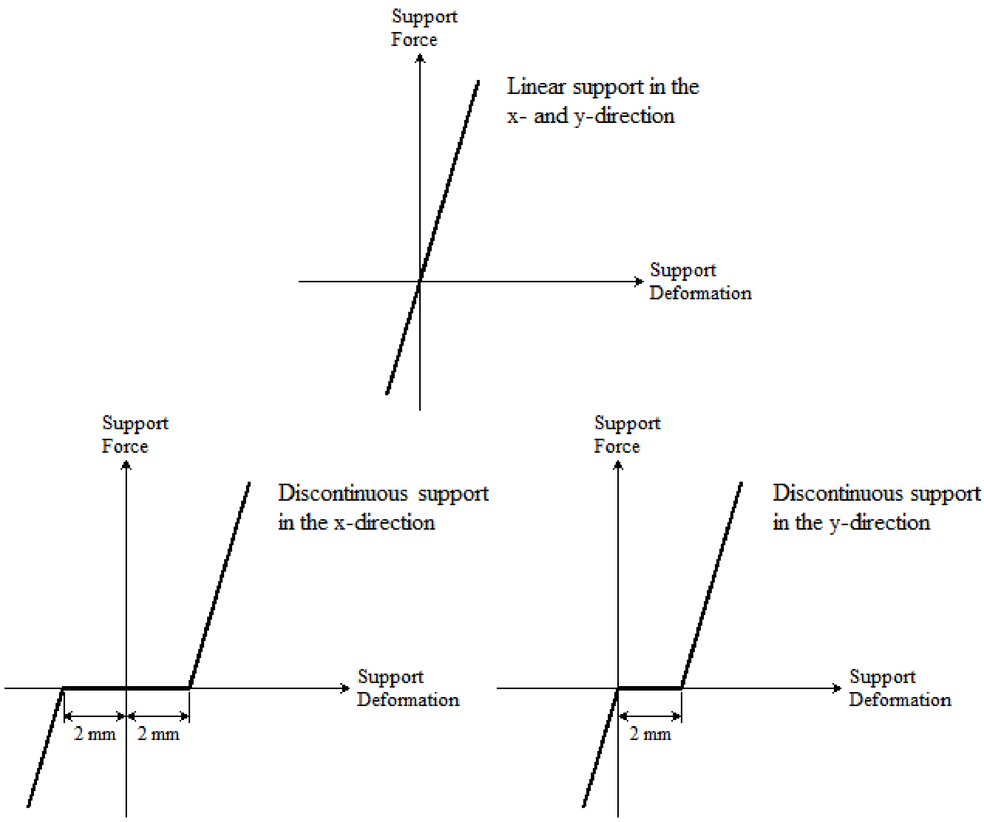


Figure 7.3: The behavior of support number 1 in the linear and non-linear analysis.

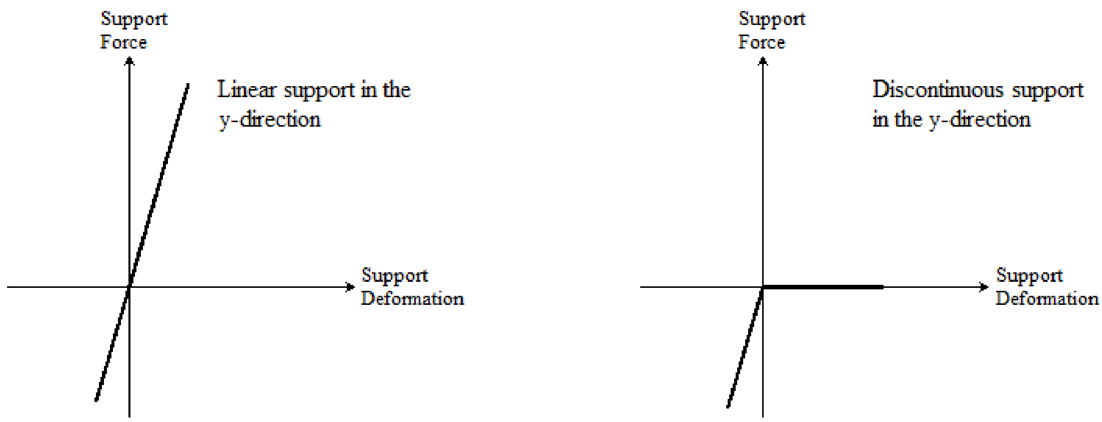


Figure 7.4: The behavior of support number 2 and 5 in the linear and non-linear analysis.

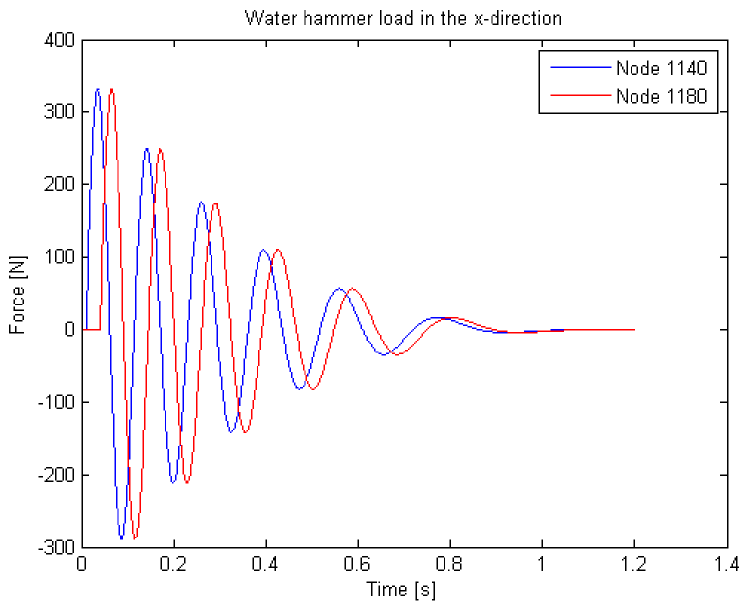


Figure 7.5: The water hammer load in the x - direction.

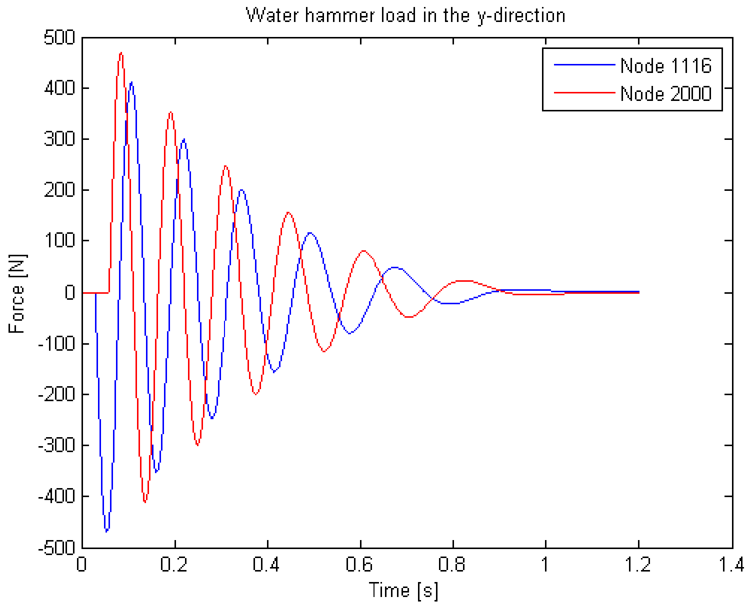


Figure 7.6: The water hammer load in the y - direction.

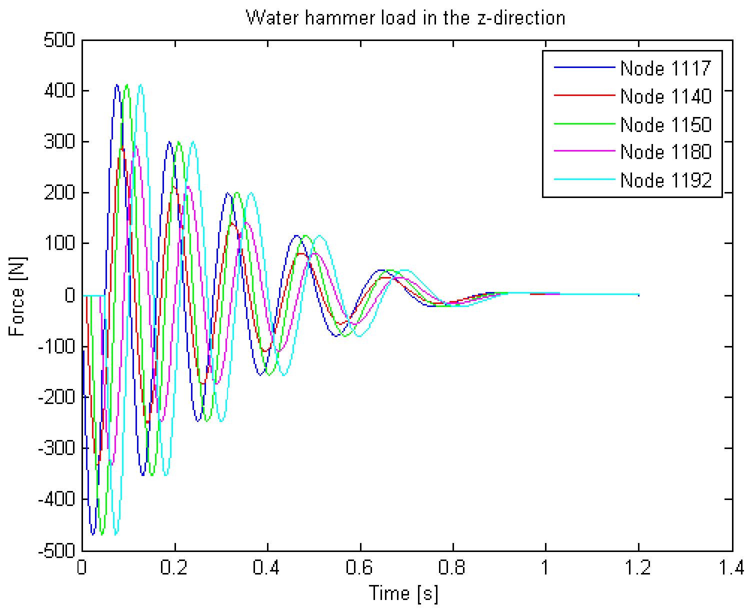


Figure 7.7: The water hammer load in the z - direction.

7.2 Linear analyses in Pipestress and ANSYS

A typical presentation of the results in Pipestress is shown in Figure 7.8. This output contains a report of the 20 highest stress points. The evaluation code created to ANSYS presents the results in the same manner. The nodal results for the components are printed and sorted for all points but the presented output file is cut to fit this report.

Equation (2.1) is evaluated in Pipestress and ANSYS for the model presented above. The loads are taken as the weight of the system and an internal pressure.

The loads in (2.2) are still the weight of the system and a changed internal pressure. The dynamic part is taken as the presented water hammer load.

LOADING CASE NO. 301 COMBINATION ANALYSIS - PD+DW, Eq8

PR CASE 10 CASES 21
METH ALGADD FACT 1.0
THERMAL FORCES AND MOMENTS MULTIPLIED BY EC/EH

EQUATION 8 SUSTAINED LOAD

- - - - - HIGHEST STRESS POINTS - - - - -

RANK	--POINTS-- FROM	AT	ELEMENT	CORROSION TYPE	PERCENT	WALL THICK. MM	OUTER DIAM. MM	SECTION MODULUS MM**3	B1	PRESS. STRESS N/MM**2	B2	MOMENT STRESS N/MM**2	STRESS RATIO	
1	MID	6	REDUCER	NOM.	0.000	12.700	219.000	0.40141E+06	1.000	29.61	1.000	5.50	0.243	
2	MID	7	REDUCER	NOM.	0.000	17.450	324.000	0.12225E+07	1.000	32.34	1.000	2.03	0.238	
3	1175	1116	SR ELBOW	NOM.	0.000	5.540	60.320	0.11979E+05	0.048	0.42	2.526	26.66	0.167	
4	25	26	TANGENT	NOM.	0.000	12.700	219.000	0.40141E+06	0.500	14.81	1.000	9.18	0.166	
5	1116	1175	SR ELBOW	NOM.	0.000	5.540	60.320	0.11979E+05	0.048	0.42	2.526	25.74	0.161	
6	q1	20	LR ELBOW	NOM.	0.000	12.700	219.000	0.40141E+06	0.500	14.81	1.000	6.48	0.147	
7	7	8	WELDING TEE	NOM.	0.000	17.450	324.000	0.12879E+07	0.500	16.17	2.129	4.55	0.144	RUN
	15	8	WELDING TEE	NOM.	0.000	12.700	219.000							BRANCH
8	50B	6	TANGENT	NOM.	0.000	12.700	219.000	0.40141E+06	0.500	14.81	1.000	5.50	0.141	
9	26	25	TANGENT	NOM.	0.000	12.700	219.000	0.40141E+06	0.500	14.81	1.000	5.46	0.140	
10	9	8	WELDING TEE	NOM.	0.000	17.450	324.000	0.12879E+07	0.500	16.17	2.129	4.07	0.140	RUN
11	1180	1185	TANGENT	NOM.	0.000	5.540	60.320	0.11979E+05	0.500	4.39	1.000	17.70	0.136	
12	1120	1130	TANGENT	NOM.	0.000	5.540	60.320	0.11979E+05	0.500	4.39	1.000	16.91	0.131	
13	1150	1165	SR ELBOW	NOM.	0.000	5.540	60.320	0.11979E+05	0.048	0.42	2.526	20.33	0.128	
14	18A	18	SR ELBOW	NOM.	0.000	12.700	219.000	0.40141E+06	0.500	14.81	1.000	3.60	0.128	
15	13A	13	LR ELBOW	NOM.	0.000	17.450	324.000	0.12225E+07	0.500	16.17	1.000	2.07	0.126	
16	8	7	TANGENT	NOM.	0.000	17.450	324.000	0.12225E+07	0.500	16.17	1.000	2.03	0.126	
17	10	910	TANGENT	NOM.	0.000	17.450	324.000	0.12225E+07	0.500	16.17	1.000	1.95	0.126	
18	22A	22	LR ELBOW	NOM.	0.000	12.700	219.000	0.40141E+06	0.500	14.81	1.000	3.19	0.125	
19	190	19	TANGENT	NOM.	0.000	12.700	219.000	0.40141E+06	0.500	14.81	1.000	3.11	0.124	

Figure 7.8: Equation (2.1) evaluated in Pipestress.

EQUATIONS OUTPUT

EQUATION 8

RANK	FROM	AT	ELEMENT	WALL THICK. [mm]	OUTER. DIAM. [mm]	SECTION MODULUS [mm^3]	B1	PRESS.STRESS [N/mm^2]	B2	MOM.STRESS [N/mm^2]	STRESS RATIO	
1	186	187	218	12.70	219.00	401514.76	1.000	29.398	1.000	6.821	0.243	
2	187	186	218	17.45	324.00	1222815.22	1.000	32.108	1.000	2.002	0.228	
3	2032	2031	244	5.54	60.32	11982.16	0.048	0.419	2.526	27.009	0.159	
4	52	26	19	12.70	219.00	401514.76	0.500	14.699	1.000	8.885	0.158	
5	2031	2032	244	5.54	60.32	11982.16	0.048	0.419	2.526	26.090	0.154	
6	19	171	14	12.70	219.00	401514.76	0.500	14.699	1.000	6.507	0.142	
7	187	8	20	17.45	324.00	1287917.23	0.500	16.054	2.129	5.000	0.141	Run
8	158	186	7	12.70	219.00	401514.76	0.500	14.699	1.000	6.096	0.139	
9	189	8	21	17.45	324.00	1287917.23	0.500	16.054	2.129	4.599	0.138	Run
10	26	52	19	12.70	219.00	401514.76	0.500	14.699	1.000	5.384	0.135	
11	1186	1185	233	5.54	60.32	11982.16	0.500	4.394	1.000	17.945	0.130	
12	1180	1185	232	5.54	60.32	11982.16	0.500	4.394	1.000	17.945	0.130	
13	1131	1130	226	5.54	60.32	11982.16	0.500	4.394	1.000	17.361	0.126	
14	1122	1130	225	5.54	60.32	11982.16	0.500	4.394	1.000	17.361	0.126	
15	2043	167	12	12.70	219.00	401514.76	0.500	14.699	1.000	3.673	0.123	
16	2028	2029	243	5.54	60.32	11982.16	0.048	0.419	2.526	20.597	0.122	
17	179	188	8	17.45	324.00	1222815.22	0.500	16.054	1.000	2.127	0.122	
18	172	171	210	12.70	219.00	401514.76	0.046	1.341	2.550	16.592	0.120	
19	19	169	13	12.70	219.00	401514.76	0.500	14.699	1.000	3.171	0.120	
20	173	163	15	12.70	219.00	401514.76	0.500	14.699	1.000	3.161	0.120	
21	2029	2028	243	5.54	60.32	11982.16	0.048	0.419	2.526	19.975	0.119	
22	2	1	1	12.70	219.00	401514.76	0.500	14.699	1.000	2.925	0.118	
23	2042	16	248	12.70	219.00	424514.04	0.500	14.699	1.419	2.700	0.117	Run
24	6	177	5	12.70	219.00	401514.76	0.500	14.699	1.000	2.681	0.116	
25	183	181	9	17.45	324.00	1222815.22	0.500	16.054	1.000	1.326	0.116	
26	157	6	6	12.70	219.00	401514.76	0.500	14.699	1.000	2.669	0.116	
27	177	6	5	12.70	219.00	401514.76	0.500	14.699	1.000	2.669	0.116	
28	2043	16	249	12.70	219.00	424514.04	0.500	14.699	1.419	2.668	0.116	Run
29	185	14	10	17.45	324.00	1222815.22	0.500	16.054	1.000	1.165	0.115	
30	55	8	22	12.70	219.00	424514.04	0.500	14.699	1.703	2.479	0.115	Branch
31	163	173	15	12.70	219.00	401514.76	0.500	14.699	1.000	2.279	0.114	
32	167	2043	12	12.70	219.00	401514.76	0.500	14.699	1.000	2.159	0.113	
33	165	23	16	12.70	219.00	401514.76	0.500	14.699	1.000	2.128	0.113	
34	23	165	16	12.70	219.00	401514.76	0.500	14.699	1.000	2.125	0.113	
35	1	2	1	12.70	219.00	401514.76	0.500	14.699	1.000	2.065	0.112	
36	188	179	8	17.45	324.00	1222815.22	0.500	16.054	1.000	0.618	0.112	
37	181	183	9	17.45	324.00	1222815.22	0.500	16.054	1.000	0.575	0.111	
38	14	185	10	17.45	324.00	1222815.22	0.500	16.054	1.000	0.565	0.111	
39	55	2042	11	12.70	219.00	401514.76	0.500	14.699	1.000	1.860	0.111	
40	171	19	14	12.70	219.00	401514.76	0.500	14.699	1.000	1.469	0.108	

Figure 7.9: Equation (2.1) evaluated in ANSYS.

LOADING CASE NO. 305 COMB.STRESS ANALYSIS - PO+DW+WH (dynamic),EQ9

PR CASE 1 CASES MA= 21 MB= 32 MC= 0
METH UNSIGN FACT 1.0 1.0
THERMAL FORCES AND MOMENTS MULTIPLIED BY EC/EH

EQUATION 9a OCCASIONAL LOAD - LEVEL A&B

- - - - - HIGHEST STRESS POINTS - - - - -

RANK	--POINTS-- FROM	AT	ELEMENT	CORROSION TYPE	PERCENT	WALL THICK. MM	OUTER DIAM. MM	SECTION MODULUS MM**3	B1	PRESS. STRESS N/MM**2	B2	MOMENT STRESS N/MM**2	STRESS RATIO
1	1195	1192	SR ELBOW	NOM.	0.000	5.540	60.320	0.11979E+05	0.048	0.34	2.526	190.24	0.978
2	1192	1195	SR ELBOW	NOM.	0.000	5.540	60.320	0.11979E+05	0.048	0.34	2.526	183.06	0.941
3	1165	1150	SR ELBOW	NOM.	0.000	5.540	60.320	0.11979E+05	0.048	0.34	2.526	181.46	0.933
4	1150	1165	SR ELBOW	NOM.	0.000	5.540	60.320	0.11979E+05	0.048	0.34	2.526	180.09	0.926
5	2000	2010	SR ELBOW	NOM.	0.000	5.540	60.320	0.11979E+05	0.048	0.34	2.526	179.31	0.922
6	2010	2000	SR ELBOW	NOM.	0.000	5.540	60.320	0.11979E+05	0.048	0.34	2.526	175.88	0.904
7	1175	1116	SR ELBOW	NOM.	0.000	5.540	60.320	0.11979E+05	0.048	0.34	2.526	134.37	0.691
8	1116	1175	SR ELBOW	NOM.	0.000	5.540	60.320	0.11979E+05	0.048	0.34	2.526	133.99	0.689
9	1120	1119	SR ELBOW	NOM.	0.000	5.540	60.320	0.11979E+05	0.048	0.34	2.526	124.38	0.640
10	1119	1120	SR ELBOW	NOM.	0.000	5.540	60.320	0.11979E+05	0.048	0.34	2.526	121.66	0.626
11	1140	1145	SR ELBOW	NOM.	0.000	5.540	60.320	0.11979E+05	0.048	0.34	2.526	115.62	0.595
12	1145	1140	SR ELBOW	NOM.	0.000	5.540	60.320	0.11979E+05	0.048	0.34	2.526	114.71	0.590
13	2010	2020	TANGENT	NOM.	0.000	5.540	60.320	0.11979E+05	0.500	3.60	1.000	94.28	0.502
14	1187	1188	SR ELBOW	NOM.	0.000	5.540	60.320	0.11979E+05	0.048	0.34	2.526	90.15	0.464
15	1188	1187	SR ELBOW	NOM.	0.000	5.540	60.320	0.11979E+05	0.048	0.34	2.526	89.52	0.461
16	1120	1130	TANGENT	NOM.	0.000	5.540	60.320	0.11979E+05	0.500	3.60	1.000	79.87	0.428
17	1190	1192	TANGENT	NOM.	0.000	5.540	60.320	0.11979E+05	0.500	3.60	1.000	75.32	0.405
18	1130	1120	TANGENT	NOM.	0.000	5.540	60.320	0.11979E+05	0.750	5.39	1.500	72.25	0.398
19	2000	1195	TANGENT	NOM.	0.000	5.540	60.320	0.11979E+05	0.500	3.60	1.000	72.47	0.390
20	1145	1150	TANGENT	NOM.	0.000	5.540	60.320	0.11979E+05	0.500	3.60	1.000	71.84	0.387

Figure 7.10: Equation (2.2) evaluated in Pipestress.

Equation 9a Level A&B

RANK	FROM	AT	ELEMENT	WALL THICK. [mm]	OUTER. DIAM. [mm]	SECTION MODULUS [mm^3]	B1	PRESS.STRESS [N/mm^2]	B2	MOM.STRESS [N/mm^2]	STRESS RATIO
1	2032	2031	244	5.54	60.32	11982.16	0.048	0.343	2.526	181.033	0.879
2	2031	2032	244	5.54	60.32	11982.16	0.048	0.343	2.526	180.859	0.878
3	2026	2025	242	5.54	60.32	11982.16	0.048	0.343	2.526	130.473	0.634
4	2025	2026	242	5.54	60.32	11982.16	0.048	0.343	2.526	129.912	0.631
5	2035	2034	245	5.54	60.32	11982.16	0.048	0.343	2.526	115.153	0.560
6	2034	2035	245	5.54	60.32	11982.16	0.048	0.343	2.526	113.472	0.551
7	2023	2022	241	5.54	60.32	11982.16	0.048	0.343	2.526	104.031	0.506
8	2022	2023	241	5.54	60.32	11982.16	0.048	0.343	2.526	101.821	0.495
9	2038	2037	246	5.54	60.32	11982.16	0.048	0.343	2.526	99.651	0.484
10	2041	2040	247	5.54	60.32	11982.16	0.048	0.343	2.526	97.182	0.473
11	2037	2038	246	5.54	60.32	11982.16	0.048	0.343	2.526	96.777	0.471
12	2040	2041	247	5.54	60.32	11982.16	0.048	0.343	2.526	96.629	0.470
13	2029	2028	243	5.54	60.32	11982.16	0.048	0.343	2.526	81.908	0.399
14	1180	2032	231	5.54	60.32	11982.16	0.500	3.595	1.000	72.776	0.370
15	2028	2029	243	5.54	60.32	11982.16	0.048	0.343	2.526	75.911	0.369
16	1151	2031	230	5.54	60.32	11982.16	0.500	3.595	1.000	72.184	0.367
17	2041	2020	240	5.54	60.32	11982.16	0.500	3.595	1.000	62.948	0.322
18	1122	2023	224	5.54	60.32	11982.16	0.750	5.393	1.500	60.359	0.319
19	1186	1185	233	5.54	60.32	11982.16	0.500	3.595	1.000	56.674	0.292
20	1180	1185	232	5.54	60.32	11982.16	0.500	3.595	1.000	56.588	0.292
21	1122	1130	225	5.54	60.32	11982.16	0.500	3.595	1.000	52.152	0.270
22	1131	1130	226	5.54	60.32	11982.16	0.500	3.595	1.000	52.139	0.270
23	2028	2026	228	5.54	60.32	11982.16	0.500	3.595	1.000	52.093	0.270
24	1131	2025	227	5.54	60.32	11982.16	0.500	3.595	1.000	52.009	0.269
25	2034	1186	234	5.54	60.32	11982.16	0.500	3.595	1.000	51.364	0.266
26	1185	1186	233	5.54	60.32	11982.16	0.500	3.595	1.000	51.179	0.265
27	1186	2034	234	5.54	60.32	11982.16	0.500	3.595	1.000	46.513	0.243
28	1188	2035	235	5.54	60.32	11982.16	0.500	3.595	1.000	46.170	0.241
29	1130	1131	226	5.54	60.32	11982.16	0.500	3.595	1.000	45.384	0.237
30	2025	1131	227	5.54	60.32	11982.16	0.500	3.595	1.000	45.365	0.237
31	2031	1151	230	5.54	60.32	11982.16	0.500	3.595	1.000	42.718	0.224
32	2029	1151	229	5.54	60.32	11982.16	0.500	3.595	1.000	42.543	0.224
33	1121	1120	223	5.54	60.32	11982.16	0.500	3.595	1.000	41.784	0.220
34	1121	2022	222	5.54	60.32	11982.16	0.500	3.595	1.000	41.174	0.217
35	2022	1121	222	5.54	60.32	11982.16	0.500	3.595	1.000	41.047	0.216
36	186	187	218	12.70	219.00	401514.76	1.000	27.181	1.000	7.981	0.210
37	1191	2037	238	5.54	60.32	11982.16	0.500	3.595	1.000	39.550	0.209
38	1191	1190	237	5.54	60.32	11982.16	0.500	3.595	1.000	39.416	0.208
39	1188	1190	236	5.54	60.32	11982.16	0.500	3.595	1.000	38.991	0.206
40	2040	2038	239	5.54	60.32	11982.16	0.500	3.595	1.000	38.703	0.205

Figure 7.11: Equation (2.2) with linear supports evaluated in ANSYS.

7.3 Non-linear analysis in ANSYS

The evaluation of the dynamic load in the non-linear analysis is done according to the modified version of (2.2), given by (4.5).

Equation 9a Level A&B

RANK	FROM	AT	ELEMENT	WALL THICK. [mm]	OUTER. DIAM. [mm]	SECTION MODULUS [mm^3]	B1	PRESS.STRESS [N/mm^2]	B2	MOM.STRESS [N/mm^2]	STRESS RATIO
1	2032	2031	244	5.54	60.32	11982.16	0.048	0.343	2.526	159.903	0.776
2	2031	2032	244	5.54	60.32	11982.16	0.048	0.343	2.526	156.744	0.761
3	2038	2037	246	5.54	60.32	11982.16	0.048	0.343	2.526	132.857	0.645
4	2037	2038	246	5.54	60.32	11982.16	0.048	0.343	2.526	128.446	0.624
5	2029	2028	243	5.54	60.32	11982.16	0.048	0.343	2.526	101.791	0.495
6	2028	2029	243	5.54	60.32	11982.16	0.048	0.343	2.526	100.470	0.488
7	2040	2041	247	5.54	60.32	11982.16	0.048	0.343	2.526	96.783	0.471
8	2041	2040	247	5.54	60.32	11982.16	0.048	0.343	2.526	95.202	0.463
9	2026	2025	242	5.54	60.32	11982.16	0.048	0.343	2.526	93.206	0.453
10	2025	2026	242	5.54	60.32	11982.16	0.048	0.343	2.526	92.553	0.450
11	1180	1185	232	5.54	60.32	11982.16	0.500	3.595	1.000	88.257	0.445
12	1186	1185	233	5.54	60.32	11982.16	0.500	3.595	1.000	87.867	0.443
13	2035	2034	245	5.54	60.32	11982.16	0.048	0.343	2.526	85.049	0.414
14	2034	2035	245	5.54	60.32	11982.16	0.048	0.343	2.526	84.919	0.413
15	2041	2020	240	5.54	60.32	11982.16	0.500	3.595	1.000	69.099	0.352
16	2023	2022	241	5.54	60.32	11982.16	0.048	0.343	2.526	69.752	0.340
17	2022	2023	241	5.54	60.32	11982.16	0.048	0.343	2.526	68.068	0.331
18	1151	2031	230	5.54	60.32	11982.16	0.500	3.595	1.000	63.976	0.327
19	1180	2032	231	5.54	60.32	11982.16	0.500	3.595	1.000	62.893	0.322
20	1122	1130	225	5.54	60.32	11982.16	0.500	3.595	1.000	62.447	0.320
21	1131	1130	226	5.54	60.32	11982.16	0.500	3.595	1.000	62.409	0.320
22	1191	2037	238	5.54	60.32	11982.16	0.500	3.595	1.000	52.695	0.273
23	2040	2038	239	5.54	60.32	11982.16	0.500	3.595	1.000	51.567	0.267
24	186	187	218	12.70	219.00	401514.76	1.000	27.181	1.000	14.735	0.250
25	2031	1151	230	5.54	60.32	11982.16	0.500	3.595	1.000	44.251	0.232
26	2029	1151	229	5.54	60.32	11982.16	0.500	3.595	1.000	44.115	0.231
27	1185	1180	232	5.54	60.32	11982.16	0.500	3.595	1.000	43.399	0.228
28	1122	2023	224	5.54	60.32	11982.16	0.750	5.393	1.500	40.335	0.222
29	1191	1190	237	5.54	60.32	11982.16	0.500	3.595	1.000	41.238	0.217
30	2026	2028	228	5.54	60.32	11982.16	0.500	3.595	1.000	40.261	0.212
31	172	171	210	12.70	219.00	401514.76	0.046	1.240	2.550	34.352	0.212
32	1151	2029	229	5.54	60.32	11982.16	0.500	3.595	1.000	39.541	0.209
33	1188	1190	236	5.54	60.32	11982.16	0.500	3.595	1.000	39.373	0.208
34	2020	2041	240	5.54	60.32	11982.16	0.500	3.595	1.000	38.616	0.205
35	187	186	218	17.45	324.00	1222815.22	1.000	29.686	1.000	4.315	0.203
36	2034	1186	234	5.54	60.32	11982.16	0.500	3.595	1.000	37.922	0.201
37	2038	2040	239	5.54	60.32	11982.16	0.500	3.595	1.000	37.502	0.199
38	1185	1186	233	5.54	60.32	11982.16	0.500	3.595	1.000	37.400	0.199
39	1130	1131	226	5.54	60.32	11982.16	0.500	3.595	1.000	37.293	0.198
40	2025	1131	227	5.54	60.32	11982.16	0.500	3.595	1.000	37.216	0.198

Figure 7.12: Equation (2.2) with non-linear supports.

8 Discussion and future work

8.1 Discussion of the results

The main purpose of this master thesis was to create a code evaluation tool to ANSYS. This part is successfully made and a user manual is created including all assumptions, restrictions and instructions to the evaluation code and can be found in Appendix B. The code on more than 100 pages is not included in the report.

The results obtained from the linear analyses in ANSYS seem to agree well with the results from the same loading situations in Pipestress. The stress indices B_1 and B_2 takes the same values for the components in the both models. The sequence of the stress ratio seems to agree well between the two models and also the stresses seem to be comparable for each nodal point. It can therefore be concluded that the models are quite similar and that the evaluation code seems correctly implemented. The small differences in the results in the linear dynamic analyses are probably caused by the differences in the damping.

Some tests are also done to obtain the influence of non-linear pipe supports during dynamic loads but this type of analysis is quite hard to perform and the following obstacles have been obtained:

- Equation (2.2) for dynamic loads according to NC-3650 is not possible to carry out entirely correctly.
- The mode superposition method is not feasible to use in the non-linear analysis and due to this, an alternative model of damping has to be defined.

The results obtained from the non-linear dynamic analysis are not directly comparable with the linear analysis because the loads are not separable as in the linear analysis. This will introduce some errors since in the case of a one-way supports, the weight of the system will act as an opposing load to the dynamic load that will lift the pipe from the support. The stresses in the test model seem to decrease in the non-linear analysis and also the sequence of the stress ratios seem to differ from the linear analysis. The linear analysis may also be evaluated according to the modified form of (2.2) given by (4.5) to obtain more comparable results and find out what impact the non-linear supports actually have.

8.2 Suggestions for future work

It is hard to determine the influence of non-linear pipe supports from the test model in this project. It is not possible to predict the influence on a piping system with different configuration and loads and the analysis has to be performed on every unique system. The issue with the damping model that was encountered in the project is also a mission that needs to be examined for every new piping system and loading situation. To obtain a Rayleigh damping ratio that gives comparable results between the analysis using modal damping and the analysis using Rayleigh damping is not an evident task in all cases.

However, the common used stress indices B_1 and B_2 and the stress intensification factor SIF are calculated in the evaluation code. This gives the opportunity to further develop the code evaluation for several equations given in NC-3650.

References

[1] Smith, Paul R., Van Laan, Thomas J., *Piping and Pipe Support Systems*, McGraw-Hill, the United States of America, 1987.

[2] ASME (2010). *Boiler and Pressure Vessel Code - 2010 Edition*, http://www.asme.org/kb/standards/publications [20111228]

[3] The American Society of Mechanical Engineers, *ASME Boiler and Pressure Vessel Code III Division 1 – Subsection NC*, the United States of America, 2000.

[4] ANSYS, Inc. Theory Reference, Vol 13, 2010.

[5] Ottosen N.S, Petersson H., *Introduction to the Finite Element Method*, Prentice Hall, Europe, 1992.

[6] Timoshenko S.P, Gere J.M., *Mechanics of materials*, Van Nostrand Reinhold Company, the United States of America, 1972.

[7] Fertis Demeter G. *Mechanical and Structural Vibrations*, John Wiley & Sons, Inc. New York, 1995

[8] Krenk S., *Non-linear Modeling and Analysis of Solids and Structures*, Cambridge University Press, United Kingdom, 2009.

[9] Kirsch U., *Reanalysis of Structures: A Unified Approach for Linear, Nonlinear, Static and Dynamic Systems*, Springer, The Netherlands, 2008.

[10] Buchholdt H.A, *Structural dynamics for engineers*, Thomas Telford Publications, Great Britain, 1997.

[11] Craig Jr. Roy R., Kurdila Andrew J., *Fundamentals of Structural Dynamics, 2nd Edition*, John Wiley & Sons, Inc., New Jersey, 2006.

A Continued FE-formulation of beams

The stiffness matrices are derived for the simplest possible two dimensional beam element. For three dimensional beams elements, the derivations also needs to been done in one more plane.

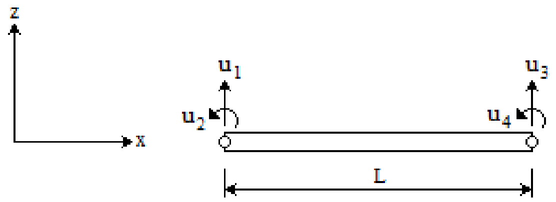


Figure A.1: Simplest possible beam element.

A.1 Euler-Bernoulli FE formulation

An approximation for the two dimensional beam element shown in Figure A.1 is chosen for the deflection w as

$$w = \alpha_1 + \alpha_2 x + \alpha_3 x^2 + \alpha_4 x^3 \tag{A.1}$$

where α_1, α_2, α_3 and α_4 are certain parameters and the unknown quantities is u_1, u_2, u_3 and u_4 shown in Figure A.1. Two unknowns are related to each nodal point, the deflection w and the change in the beam axis rotation dw/dx.

By using the **C**-matrix method it follows that (A.1) is written as

$$w = \bar{\mathbf{N}}\boldsymbol{\alpha} \tag{A.2}$$

where

$$\bar{\mathbf{N}} = \begin{bmatrix} 1 & x & x^2 & x^3 \end{bmatrix} \; ; \qquad \boldsymbol{\alpha} = \begin{bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \end{bmatrix} \tag{A.3}$$

According to Figure A.1 the conditions for w at the nodal points is expressed by

$$w_{x=0} = u_1 \; ; \qquad \left(\frac{dw}{dx}\right)_{x=0} = u_2 \; ; \qquad w_{x=L} = u_3 \; ; \qquad \left(\frac{dw}{dx}\right)_{x=L} = u_4 \tag{A.4}$$

Use of these conditions in (A.2) results in

$$\mathbf{a}^e = \mathbf{C}\boldsymbol{\alpha} \tag{A.5}$$

where

$$\mathbf{a}^e = \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} \quad ; \qquad \mathbf{C} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ 0 & L & L^2 & L^3 \\ 0 & 1 & 2L & 3L^2 \end{bmatrix} \tag{A.6}$$

From (A.5) it follows that

$$\boldsymbol{\alpha} = \mathbf{C}^{-1}\mathbf{a}^e \tag{A.7}$$

and use of this result in (A.2) gives

$$w = \mathbf{N}^e\mathbf{a}^e \tag{A.8}$$

where

$$\mathbf{N}^e = \bar{\mathbf{N}}\mathbf{C}^{-1} = \begin{bmatrix} N_1^e & N_2^e & N_3^e & N_4^e \end{bmatrix} \tag{A.9}$$

From this expression the element shape functions is derived as

$$\begin{aligned} N_1^e &= 1 - 3\frac{x^2}{L^2} + 2\frac{x^3}{L^3} \;; \qquad & N_3^e &= \frac{x^2}{L^2}\left(3 - 2\frac{x}{L}\right) \\ N_2^e &= x\left(1 - 2\frac{x}{L} + \frac{x^2}{L^2}\right) \;; & N_4^e &= \frac{x^2}{L}\left(\frac{x}{L} - 1\right) \end{aligned} \tag{A.10}$$

From (5.25) and (5.21) it follows that the element stiffness matrix for the two dimensional beam element shown in Figure A.1 is given by

$$\mathbf{K}^e = \int_0^L EI \begin{bmatrix} B_1^eB_1^e & \dots & \dots & B_1^eB_4^e \\ B_2^eB_1^e & \ddots & \dots & B_2^eB_4^e \\ B_3^eB_1^e & \dots & \ddots & B_3^eB_4^e \\ B_4^eB_1^e & \dots & \dots & B_4^eB_4^e \end{bmatrix} dx \tag{A.11}$$

which after some calculations becomes

$$\mathbf{K}^e = \begin{bmatrix} 12EI/L^3 & 6EI/L^2 & -12EI/L^3 & 6EI/L^2 \\ & 4EI/L & -6EI/L^2 & 2EI/L \\ & & 12EI/L^3 & -6EI/L^2 \\ sym. & & & 4EI/L \end{bmatrix} \tag{A.12}$$

The presented element stiffness matrix given by (A.12) for the Euler-Bernoulli theory holds for the two dimensional beam element shown in Figure A.1.

A.2 Timoshenko FE formulation

In similarity to the Euler-Bernoulli beam theory, an approximation for the two dimensional beam element shown in Figure A.1 is chosen for the deflection w as

$$w = C_1 x^3 + C_2 x^2 + C_3 x + C_4 \tag{A.13}$$

Since the shear deformation also effects, an approximation needs for the cross-section rotation θ as

$$\theta = C_5 x^2 + C_6 x + C_7 \tag{A.14}$$

Equation (A.13) and (A.14) are the homogenous solutions of (5.38) and (5.40). Equation (A.13) and (A.14) contains seven unknown parameters but as concluded earlier, there are only two unknowns related to each nodal point, i.e. some relations need to be derived. In the Euler-Bernoulli theory, the weak form of the differential equations was obtained for the FE-formulation. For the Timoshenko theory, it is not so easy to derive this in the same manner. Instead some relations between the parameters can be derived and after some calculations it holds that

$$\begin{aligned} \theta &= 3\alpha_1 x^2 + 2\alpha_2 x + 6\beta\alpha_1 + \alpha_3 \\ w &= \alpha_1 x^3 + \alpha_2 x^2 + \alpha_3 x + \alpha_4 \end{aligned} \tag{A.15}$$

or written in matrix form

$$\begin{bmatrix} w \\ \theta \end{bmatrix} = \begin{bmatrix} x^3 & x^2 & x & 1 \\ 3\left(x^2 + 2\beta\right) & 2x & 1 & 0 \end{bmatrix} \begin{bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \end{bmatrix} \tag{A.16}$$

where $\alpha_1 = C_4$, $\alpha_2 = C_5$, $\alpha_3 = C_6$, $\alpha_4 = C_7$ and $\beta = EI/GAK$

The conditions for w and θ at the nodal point is expressed by

$$w_{x=0} = u_1 \ ; \qquad \theta_{x=0} = u_2 \ ; \qquad w_{x=L} = u_3 \ ; \qquad \theta_{x=L} = u_4 \tag{A.17}$$

and following the same method as for the Euler-Bernoulli beam, the $\mathbf{C}$-matrix now becomes

$$\mathbf{C} = \begin{bmatrix} 0 & 0 & 0 & 1 \\ 6\beta & 0 & 1 & 0 \\ L^3 & L^2 & L & 1 \\ 3\left(L^2 + 2\beta\right) & 2L & 1 & 0 \end{bmatrix} \tag{A.18}$$

It also holds that

$$\mathbf{P} = \mathbf{B}\boldsymbol{\alpha} \tag{A.19}$$

where

$$\mathbf{P} = \begin{bmatrix} P_1 \\ P_2 \\ P_3 \\ P_4 \end{bmatrix} = \begin{bmatrix} -V(0) \\ M(0) \\ V(L) \\ -M(L) \end{bmatrix} = \begin{bmatrix} 6GAK\beta & 0 & 0 & 0 \\ 0 & -2EI & 0 & 0 \\ -6GAK\beta & 0 & 0 & 0 \\ 6EIL & 2EI & 0 & 0 \end{bmatrix} \begin{bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \end{bmatrix}$$

The element stiffness is then given by

$$\mathbf{K}^e = \mathbf{B}\mathbf{C}^{-1} \tag{A.20}$$

and the stiffness matrix for the Timoshenko beam according to Figure A.1 now becomes

$$\mathbf{K}^e = \frac{EI}{L^3(1+\mu)} \begin{bmatrix} 12 & 6L & -12 & 6L \\ & 4L^2\left(1+\frac{\mu}{4}\right) & -6L & 2L^2\left(1-\frac{\mu}{2}\right) \\ & & 12 & -6L \\ sym. & & & 4L^2\left(1+\frac{\mu}{4}\right) \end{bmatrix} \tag{A.21}$$

where

$$\mu = \frac{12EI}{L^2GAK}$$

It may be noted that if μ becomes zero, then

$$[\mathbf{K}^e]_{Timoshenko} = [\mathbf{K}^e]_{Bernoulli}$$

B User manual for the evaluation code

In this chapter is the user manual for the evaluation code presented.

NC-3650 code evaluation to ANSYS

Version 1.0

December 2011

Preface

The code is a result of my master thesis written in autumn 2011. The code is used for evaluating results from ANSYS according to ASME Boiler and Pressure Vessel Code Section III Division 1 Subsection NC-3650. The advantage of the code is the possibility to evaluate dynamic loads with use of non-linear pipe supports.

Helsingborg, December, 2011

Henrik Andersson

Introduction

The code is capable to evaluate eq. (8) and eq. (9a) in NC-3650. The code is divided into two main parts, one *preprocessing* part and one *post-processing* part. All files and commands are including instructions, assumptions and examples.

The piping system needs to be modelled with *PIPE16* and *PIPE18* elements in ANSYS. The element identification is done via *KEYOPT(4)* and following needs to yield:

KEYOPT(4) = 0 for straight pipes

KEYOPT(4) = 1 for valves

KEYOPT(4) = 2 for reducers

KEYOPT(4) = 3 for flanges

KEYOPT(4) = 5 for mitered bends

KEYOPT(4) = 6 for tee branches

The run directory must include the result files (.rst) from the analyses, the program files (.mac) and the input files (.txt). The element and node numbering can be arbitrary but higher numbering than necessary is not recommended due to the time consumption and the matrix sizes. The results from the load cases must be separated in load sets. All components in FIG. NC-3673.2(b)-1 and FIG. NC-3673.2(b)-2 are available to use and evaluate except for *Threaded pipe joint or threaded flange* and *Corrugated straight pipe or corrugated or creased bend* in FIG. NC-3673.2(b)-1.

Table B.1: List of the necessary files.

Input files (.txt)	Program files (.mac)
constants	man_elements
equations	postproc
materials	preproc
reducers	subprog1
teelem	subprog2
welds	subprog3
	subprog4
	subprog6
	subprog61
	subprog62
	subprog63
	subprog64
	subprogeq8
	subprogeq9

Instructions

1. Ensure that the run directory contains all the listed files in Table B.1.

2. Load the input file *xx.inp* or the model *xx.db*. Read in results from an optional set.

3. Open *equations.txt* and follow the instructions under Inputs, Equations.

4. Open *materials.txt* and follow the instructions under Inputs, Materials.

5. If the model contains any tee elements or reducers, open *teelem.txt* and *reducers.txt* and follow the instructions under Inputs, Tee elements and Inputs, Reducers. If the model do not includes any tee elements or reducers, make sure that there are not more than 4 rows in *teelem.txt* and *reducers.txt* and that these rows begins with “!”.

6. To define welds and transitions, open *welds.txt* and follow the instructions under Inputs, Welds and transitions. If the model do not includes any welds or transitions, make sure that there are not more than 4 rows in welds.txt and that these rows begins with “!”.

7. Run the preprocessing part by typing **preproc** in the ANSYS command prompt. Wait for the message *“Prepost finished! Type postproc for post-processing.”*.

8. To define B_1, B_2 or SIF manually, follow the instructions under Inputs, Manually given constants. The manually inputs have to be done after running the preprocessing part.

9. When the steps 1-7 for preprocessing are done, type **postproc** in the command prompt. Wait for the message *“Post-processing finished!”*. The results are printed to *results.txt* in the run directory.

Inputs

Equations

Name of the input file: *equations.txt*

Purpose

Define equations to evaluate, name of the result files and load cases separated in load sets. Supported equations to evaluate are eq. (8) and eq. (9a) in NC-3650.

Description

The file is divided into 12 columns. The columns define in which result file and load sets the separated results are stored.

Assumptions in eq. (8)

All load cases must be included and separated in one result file.

Inputs: Equation number, temperature load set, pressure load set, MA load set, MA result file.

Temperature load set: Load set where the Design temperature to decide S_h is defined.

Pressure load set: Load set where the internal Design Pressure P_0 is defined.

MA load set: Load set where the moments M_A due to weight and other sustained loads are defined.

MA result file: Name of the result file containing all the load sets. The length of the name is not allowed to exceed 7 characters.

Assumptions in eq. (9)

Eq. (9a) can be evaluated with linear or non-linear pipe supports. When non-linear pipe supports are used, the moments M_A and M_B shall not be separated, i.e. the analysis must be performed with the gravity included in the same load case as the occasional loads.

Linear supports:

Inputs: Equation number, temperature load set, pressure load set, MA load set, MB from load set, MB to load set, level of service limit, MA result file, MB result file.

Temperature load set: Load set where the temperature to decide S_h and S_y are defined. This load set must be included in the result file for MA.

Pressure load set: Load set where the peak pressure P_{max} is defined. This load set must be included in the MA result file.

MA load set: Load set where the moments due to weight and other sustained loads are defined.

MB from load set: First load set number in MB result file that contains the occasional load.

MB to load set: Last load set number in MB result file that contains the occasional load (all load sets between MB from and MB to will be used to get maximum resulting moment).

Level: Level of service limit. *1 = level A and B*, *2 = level C*, *3 = level D*. Level of service limit is used to decide allowable stress.

MA result file: The name of the result file containing the load sets for MA, temperature and pressure. The length of the name is not allowed to exceed 7 characters.

MB result file: The name of the result file containing the load sets for MB. The length of the name is not allowed to exceed 7 characters. MB result file can be the same as MA result file.

Non-linear pipe-supports:

Inputs: Equation number, temperature load set, pressure load set, MA+MB from load set, MA+MB to load set, level of service limit, MA+MB result file.

Temperature load set: Load set where the temperature to decide S_h and S_y are defined.

Pressure load set: Load set where the peak pressure P_{max} is defined.

MA+MB from load set: First load set number in MAB result file that contains the occasional load (all load sets between MB from and MB to will be used to get maximum resulting moment).

MA+MB to load set: Last load set number in MAB result file that contains the occasional load.

Level: Level of service limit. *1 = level A and B*, *2 = level C*, *3 = level D*. Level of service limit is used to decide allowable stress.

MAB result file: The name of the result file containing the load sets for MA+MB, temperature and pressure. The length of the name is not allowed to exceed 7 characters.

Example:

[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]	[10]	[11]	[12]
Eq. nbr	Temp.	P	MA	MB fr	MB to	MAB fr	MAB to	Level	fileMA	fileMB	fileMAB
9	1	1	2	3	85			1	result1	result2	
9	1	1				24	8492	2			result3
8	1	1	2						result4		

Materials

Name of the input file: *materials.txt*

Purpose

Define materials that are assigned in the model. The code will use this to compute the allowable stress S_h for each element with the temperature under consideration.

Description

The file is divided into 5 columns. The columns define the material numbers that are assigned in the model and the known S_y and S_u for a temperature range.

Assumptions

Material number needs to be assigned in the model. The code will interpolate Sy and Su from the given temperatures. That means if the temperature under loading is $255°C$, Sy and Su needs to be defined at for example 200 and $300°C$ to interpolate the correct values at $255°C$. The room temperature at $20°C$ shall always be included. All parameters are given in N, mm and $°C$.

Inputs: Material number, type, temperature, Sy, Su.

Material number: The assigned material number in the model.

Type: The type of the material. Austenitic = 0, Ferritic = 1. This will be used to calculate S_h.

Temp: Temperature where S_y and S_u are known. The room temperature at $20°C$ shall always be included.

Sy = material yield strength at temperature consistent.

Su = material ultimate tensile strength at temperature consistent.

Example:

[1]	[2]	[3]	[4]	[5]
Material number	Type	Temp.	Sy	Su
1	0	20	172	483
1	0	100	145	467
1	0	250	114	426
1	0	350	105	425
1	0	450	97.5	413
2	1	20	310	665
2	1	100	246	562
2	1	150	226	511
2	1	200	215	486
2	1	300	201	475
2	1	400	179	462

Tee elements

Name of the input file: *teelem.txt*

Purpose

Define tee elements in the model. The code will use the additional parameters to calculate B_1, B_2 and SIF for the defined elements.

Description

The file is divided into 9 columns. The columns define element numbers, element types and the necessary element parameters.

Assumptions

The shape of the tee elements must be "T" not "Y". An error will occur if two intersecting tee elements exist, for example

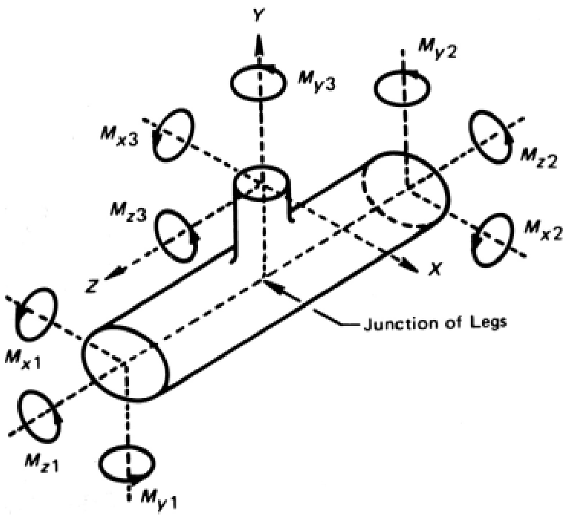
```
       |   |                 |
       |   |                 |
 ---------------   or  --------------
                             |
                             |
```

To avoid this, the tee elements can be modeled close to each other but separated without any elements having common nodes. Use of the command

`CP, NSET, Lab, NODE1, NODE2, ... ,`

can then be used to connect the nodes again so that they still get the common degree of freedom.

For reduced outlets where $r_m^{'}/R_m < 0.5$, the resultant moment of the branch pipe may be taken at the outside surface of the run pipe. The code will not consider this and the resultant moments for all intersections are taken at the junction point.



Give parameters only for one element in each tee branch, the code will find the two common elements. The evaluation code is supporting all defined tee element types in FIG. NC-3673.2(b)-1 and FIG. NC-3673.2(b)-1. All parameters shall be given in *mm.* and *degrees.*

Types:
1 = Welding tee per ASME B16.9
2 = Reinforced fabricated tee
3 = Branch connection or unreinforced fabricated tee
4 = Fillet welded and partial penetration welded branch connections

Type 1:
Inputs: Element number, type

Type 2:
Inputs: Element number, type, T_e

Assumptions in FIG. NC-3673.2(b)-1:
T_n = Thickness of the modelled run element in the tee
r = Mean radius of the modelled run element
T_b' = Thickness of the modelled branch element in the tee
$T_r = T_h$
T_e = Pad or saddle thickness

Type 3 and 4:
FIG. NC-3673.2(b)-2

FIG. NC-3673.2(b)-2(a) = 1
Inputs: Element number, type, figure, r2, L1, theta

FIG. NC-3673.2(b)-2(b) = 2
Inputs: Element number, type, figure,r2, L1

FIG. NC-3673.2(b)-2(c) = 3
Inputs: Element number, type, figure, rp, r2, L1, theta, y

FIG. NC-3673.2(b)-2(d) = 4
Inputs: Element number, type, figure, r2

Assumptions in FIG. NC-3673.2(b)-2:

$T_b^{'}$ = Thickness of the connecting pipe to branch element
T_b = Thickness of the modelled branch element in the tee
$r_m^{'}$ = Mean radius of the connecting pipe to branch element
R_m = Mean radius of the modelled run element in the tee
T_r = Thickness of the modelled run element in the tee
r_p = Outside radius of the modelled branch element in (a), (b) and (d), manually given in (c)

Example:

[1] Element number	[2] Type	[3] Figure	[4] te	[5] rp	[6] r2	[7] L1	[8] theta	[9] y
17	2		5					
28	3	3		200	5	400	45	100
5	1							
44	4	1			10	350	30	
51	3	2			7.5	325		
68	3	4			15			

Reducers

Name of the input file: *reducers.txt*

Purpose

Define reducer elements in the model. The code will use the additional parameter to calculate B_1, B_2 and SIF for the defined elements.

Description

The file is divided into 2 columns. The columns define element number and the cone angle alfa.

Assumptions

The cone angle alfa should be given in positive *degrees*.

Example:

[1]	[2]
Element number	alfa
43	45
11	30

Welds and transitions

Name of the input file: *welds.txt*

Purpose

Define welds and transitions in the model. The code will use the additional parameters to calculate B_1, B_2 and SIF for the defined nodes.

Description

The file is divided into 3 columns. The columns define node numbers, types of welds or transitions and necessary parameters. The code will save B_1, B_2 and SIF for the common elements but not write over the previous values. When evaluating the equations, the code computes the stress ratio in the nodal point with and without the weld/transition. The highest stress ratio determines which indices that will be saved. This can be B_1 and B_2 for the element or B_1 and B_2 for the weld/transition but not B_1 for the element and B_2 for the weld/transition and neither vice versa.

Assumptions

Supported welds and transitions are *Girth butt weld*, *Circumferential fillet socket welded or socket welded joints*, *Brazed joints*, and *30 deg tapered transitions*.

Types:
1 = Girth butt weld
2 = Circumferential fillet welded or socket welded joints
3 = Brazed joint
4 = 30 deg tapered transition

Type1:
Inputs: Node number

Type 2:
Inputs: Node number, Cx (See FIG. NC-4427-1 sketches (c-1), (c-2), and (c-3))

Type 3:
Inputs: Node number

Type 4:
Inputs: Node number

Example:

[1] Node number	[2] Type	[3] Cx
55	2	5
19	3	
182	4	
177	1	

Manually given constants

Name of the input file: *constants.txt*

Purpose

Define B_1, B_2 and SIF manually.

Description

The file is divided into 5 columns. The columns define node numbers and the manually given parameters.

Assumptions

If column 5 is left blank, the parameters will be replaced for all elements that are connected to the defined node. If column 5 contains an element number, the parameters will only replace the old parameters in the given node for the defined element. If some column is left blank, 0 will be inserted for that parameter. Old parameters will be replaced.

Example:

[1] Node number	[2] B1	[3] B2	[4] SIF	[5] Elem. nbr
18	1	1	1	10
27	2	2	1.3	
230	1	0.55	1	