NTNU – Trondheim
Norwegian University of
Science and Technology

Local bracer vibrations for Offshore Wind jackets

Thomas B. Johnsen
Kim Sandvik

Mechanical Engineering
Submission date: June 2013
Supervisor: Bjørn Haugen, IPM

Norwegian University of Science and Technology
Department of Engineering Design and Materials

Local bracer vibrations

for

Offshore Wind jackets

By:

Kim Sandvik   Thomas B. Johnsen

I. Summary

During the course of this master project, the candidates have worked with FEA of local vibrations occurring in bracers on offshore windturbine jacket. These vibrations arise as a dynamic response from wind-, wave-, and windturbine- loads. The predicted vibrations can vary depending on which analysis method is being used. The methods in question are fully coupled- and sequentially coupled analysis. The given task provided by TDA included a literature study, a study of the phenomenon in question and a recommendation of how to handle the problem.

It was necessary to be able to run both fully coupled analysis and sequentially coupled analysis to be able to study the phenomenon. The FEA codes USFOS and FEDEM was chosen for the analyses in the project. The fully coupled analysis were performed in FEDEM, while the sequentially coupled analysis were performed using a combination of both FEDEM and USFOS.

To be able to run a sequentially coupled analysis, using a combination of FEDEM and USFOS, the general properties of the jacket structure had to be defined in FEDEM without including the actual jacket structure in the FEA. This was solved by replacing the jacket structure by a full spring connected to ground. The properties of the spring included the full stiffness matrix, the diagonal damping matrix and the diagonal mass matrix. These matrices were extracted analytically from analyses made in USFOS that captured the response of the jacket structure from static and dynamic load cases. Generating these matrices was a complex and time consuming task, and created a need for scripts that could run and extract information from a large number of analyses.

Once the fully coupled- and sequential analyses were done, comparisons could be made. Prior to viewing the results, the assumption was that the sequentially coupled analysis would predict more damage than the fully coupled analysis. The results showed that the sequentially coupled analysis predicted more damage than fully coupled analysis, but the differences were extreme, and the explanation to this had to be found. After debugging input files, output files, and scripts involved in the results extraction without finding any errors, the cause would have to lie within the FEM analysis. It was reasonable to believe that the error was within the sequentially coupled analysis due to the prediction of very short lifetime. The debugging was done to find which factors contributed to the error in question. The factor contributing most to the errors was found likely to be values in the **K**, **C** and **M** matrixes affecting the rotational degree of freedom in z-direction. This results in severe twisting of the offshore wind turbine, contributing to higher axial forces in the bracers, again affecting the accumulated damage.

This report shows how important and how difficult it is to get all parameters right when doing a sequentially coupled analysis, indicating that it is a vulnerable method when generating the needed **K**, **C** and **M** matrixes analytically.

II. Sammendrag

I denne masteroppgaven har kandidatene arbeidet med FEM analyser av lokale vibrasjoner som oppstår i kryss-stag i jacetstruktur på offshore vindturbiner. Disse vibrasjonene oppstår som en dynamisk respons på grunn av vind, bølge og vindturbinlaster. De predikerte vibrasjonene kan variere avhengig av benyttet beregningsmetode. De aktuelle metodene er fullkoblet- og sekvensiell koblet analyse. Masteroppgaven gitt av TDA inkluderer et litteraturstudium, et studie av gjeldene problem og en anbefaling på hvordan håndtere problemet.

Det var nødvendig å være i stand til å utføre både fullkoblet analyse og sekvensiell analyse for å kunne studere det aktuelle fenomenet. Analyseprogrammene USFOS og FEDEM ble valgt til å utføre beregningene i prosjektet. Den fullkoblede analysemetoden ble utført i FEDEM, mens den sekvensielt koblede metoden ble utført ved å benytte både FEDEM og USFOS i kombinasjon.

For å bli i stand til å kjøre en sekvensielt koblet analyse ved hjelp av en kombinasjon av FEDEM og USFOS, måtte de strukturelle egenskapene til jacketen defineres i FEDEM uten å inkludere den faktiske FEA-modellen. Dette ble løst ved å lage en avansert fjær koblet til jord og til vindmølletårn, med samme karakteristikk som jacketstrukturen. Fjærkarakteristikken var beskrevet ved full stivhetsmatrise, diagonal dempingsmatrise og diagonal massematrise. Disse matrisene ble generert analytisk fra responsen av statiske og dynamiske analyser utført i USFOS. Arbeidet med å generere disse matrisene var både tidkrevende og komplisert, og førte til et behov for små programmer som kunne kjøre- og hente ut resultater fra et stort antall analyser.

Da de fullkoblede- og sekvensielle analysene var utført ble de sammenlignet med hverandre. Før resultatene var klare var det antatt at sekvensielt koblet analyse ville predikere større skade enn fullkoblet analyse. Resultatene viste at sekvensielt koblet analyse predikerte mer skade enn fullkoblet analyse, men forskjellene var meget store, og årsaken til dette måtte finnes. Etter feilsøking i inputfiler, outputfiler og script som var involvert i resultathenting uten å finne noen feil, måtte feilen ligge i FEM analysen. Det var rimelig å anta at feilkilden lå i sekvensielt koblet analyse metoden på grunn av veldig kort predikert levetid. Hensikten med feilsøkingen var å finne hvilke faktorer som forårsaket til avviket. Faktoren med størst bidrag til avviket viste seg mest sannsynlig å være verdier i **K**, **C** og **M** matrisene som medvirker i rotasjonsfrihetsgraden i z-retning. Dette gir store vridninger i hele strukturen, som igjen gir høyere aksielle krefter i kryss-stagene og dermed økt akkumulert skade.

Denne rapporten viser hvor viktig og hvor utfordrende det er å generere riktige parametere til en sekvensielt koblet analyse, som igjen påviser sårbarheten til metoden dersom de nødvendige **K**, **C** og **M** matrisene genereres analytisk.

III. Foreword

This master project has introduced the candidates to a field of engineering including a whole range of interesting challenges. The idea of working with renewable energy in combination with offshore structures has been intriguing and a strong motivation during the work. The work has spanned several disciplines, and especially worth mentioning;

- Offshore loading and hydrodynamics
- Vibration studies and the interpretation of vibration signals
- Fatigue and damage estimations on signals with variable amplitude and variable mean stress.
- Learning to know new analysis software (FEDEM and USFOS)
- Learning the fundamentals of creating a sequentially coupled analysis
- Scripting in Visual Basic and batch to handle large quantities of data, FEA’s and results
- Real life challenges in analysis work; huge file sizes (too large for certain computers) and debugging of analysis and scripts

The candidates are very pleased with the implementation of the project and want to thank all parties contributing to discussions and help throughout the project.

We would like to extend our utmost appreciation to:

Bjørn Haugen

- *Extended guidance throughout the project*

Jan Erik Børja Berg

- *Provider of master thesis, TDA*

Jørgen Kvaleid

- *Technical advisor, TDA*

Jan Ivar Hansen

- *Technical advisor, TDA*

Michel Wanderdam

- *Technical advisor, TDA*

Frederik Pettersen

- *Technical advisor, TDA*

Tore Holmås

- *USFOS support and guidance*

Runar H. Refsnæs

- *FEDEM support and guidance*

Knut Morten Okstad

- *FEDEM support and guidance*

IV. TDA AS

TDA, Teknisk Data Analyse, is an engineering company established in 1978. Their offices are located at Helsfyr in Oslo, Norway.

The company is divided into three departments; Consultancy, Projects and Software, refers to *Figure 1 - Organization - TDA - ref* .

The company has three fields of expertize, consisting of Offshore, Marine and Bridges. The project department delivers broadly defined engineering projects with competence for the full life-cycle which ranging from detailed engineering to decommissioning. The consultancy services are based upon mentoring, support and training. The software products includes software related support, training, back office and sale.

Together with OWEC, TDA has conducted detailed foundation design on offshore wind farms such as Alpha Ventus, Ormonde and Thornton Bank.

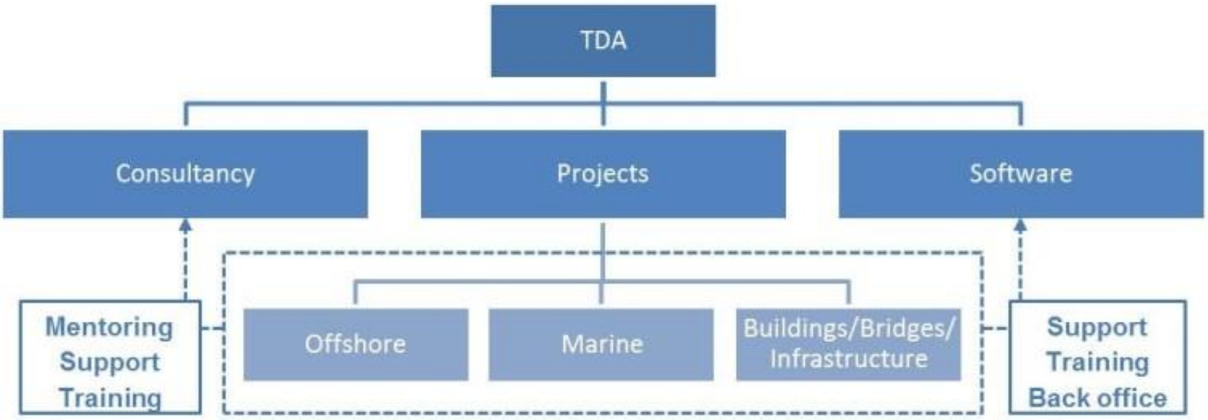


Figure 1 - Organization - TDA - ref (About us: TDA A/S Organization chart)

For more information, refer to reference (Main Page: TDA A/S - Webside).

THE NORWEGIAN UNIVERSITY
OF SCIENCE AND TECHNOLOGY
DEPARTMENT OF ENGINEERING DESIGN
AND MATERIALS

MASTER THESIS SPRING 2013
FOR
STUD.TECHN. KIM SANDVIK OG THOMAS JOHNSEN

LOCAL BRACER VIBRATIONS ON OFFSHORE WINDTURBINE JACKETS
Lokale svigninger i avstivere på offshore vindturbin jackets.

Local vibrations of braces in Offshore Wind turbine jackets - often named whiskey vibrations - his is a phenomenon that can dominate the fatigue design of such structures. It is unclear to what extent this is a real phenomenon, and it is thus of interest to get an estimate of how much, and what kind of damping there is. Moreover, it is important to understand how well the analysis captures reality, the simplifications/assumptions/mistakes made, and to what extent these assumptions influence the analysis results.

The work should include:

1. Literature study to see what information and previous work is available in this field
2. A study of the phenomenon in e.g. FEDEM and Usfos.
3. Recommendations on how such vibrations should be addressed in the design process for jackets

The thesis should include the signed problem text, and be written as a research report with summary both in English and Norwegian, conclusion, literature references, table of contents, etc. During preparation of the text, the candidate should make efforts to create a well arranged and well written report. To ease the evaluation of the thesis, it is important to cross-reference text, tables and figures. For evaluation of the work a thorough discussion of results is appreciated.

The work will be carried out in collaboration with TDA AS.

Three weeks after start of the thesis work, an A3 sheet illustrating the work is to be handed in. A template for this presentation is available on the IPM's web site under the menu "Masteroppgave" (http://www.ntnu.no/ipm/masteroppgave). This sheet should be updated one week before the Master's thesis is submitted.

The thesis shall be submitted electronically via DAIM, NTNU's system for Digital Archiving and Submission of Master's thesis.

The contact person is Jan Erik Børja Berg (TDA AS)

Torgeir Welo

Torgeir Welo
Head of Division

NTNU
Norges teknisk-
naturvitenskapelige universitet
Institutt for produktutvikling
og materialer

Bjørn Haugen

Bjørn Haugen
Professor/Supervisor

VI. TABLE OF CONTENTS

6 ANALYSIS RESULTS 94

7 DISCUSSION 113

VII. DIAGRAM LIST

VIII. TABLE LIST

1 Introduction

1.1 Scope

This master thesis revolves around the occurrence and magnitude of local out of plane vibrations in offshore jacket structures carrying a wind turbine. These vibrations are more prone to occur when doing sequentially coupled analysis (ref chapter 2.3.5).

1.2 Previous knowledge and work

No pre-project was done prior to this master thesis, and the candidates had no special knowledge about offshore structures or wind turbines before starting on this project. It was therefore necessary to do a study of basic offshore loading and hydrodynamics theory in the beginning of the project.

1.3 The Task

The task provided by TDA AS describes the phenomenon of out of plane vibrations observed in bracers when doing FEM analyses on an offshore wind turbine jacket. The vibrations in question are in the range from 2-5 Hz, and the amplitude range is 0.1-1% of the bracer diameter. TDA addresses the following needs on the topic:

1) The need for a literature study on the issue of damping high frequency vibrations.

2) A study of the problem to check the validity of the predicted vibrations. Are the vibrations real or are they not?

3) A recommended practice describing how to handle the vibrations in question.

1.4 The task revised

Two methods for analysing the complete windturbine are investigated. The two methods are fully coupled analysis and sequentially coupled analysis.

Information provided by TDA AS in the beginning of April indicated that the issue of large high frequency vibrations might be related to the sequentially analysis method. At the approximate same time it became clear that sending the jacket structure to 3rd party was not possible. The consequence of the former was change of focus, while the latter demanded the use of FEDEM for running sequentially coupled analysis. This was a turning point in the project, and the literature study is therefore inclined towards the topic of sequentially coupled analysis. This was done to gain knowledge of the method, along with knowledge of how to recreate the phenomenon observed by TDA AS.

Literature was found (Muskulus, The full-height lattice tower concept, 2012) supporting the indicated phenomenon of large high frequency vibrations in sequentially coupled analysis. To have a reasonable reference for comparing results, a fully coupled analysis also had to be done.

Being able to run a sequentially coupled analysis along with a fully coupled analysis with two arbitrary software programs was considered important and has been the main task in the project. Much time has been spent on learning the analysis tools (USFOS and FEDEM), along with creating the needed tools for being able to run and collect results from many analyses in an effective way. Much work was also put in to the development of a method to generate the stiffness-, mass- and damping matrix needed for the sequentially coupled analysis method, and is also a theme discussed in this report.

Simplified FEA models were considered, and tested, but it was of interest to capture the full effect of the structure’s response as a whole. This was done because it was believed that the full response had an effect on the out of plane vibrations on the bracers studied. Making a less refined mesh was also preformed, but this analysis model was not used based on an advice from TDA. The analysis model of the jacket structure is therefore kept as is with the exception of minor changes to make the model work within USFOS.

1.5 About this report

This report will focus on illuminating the differences between fully coupled- and sequentially coupled analysis, and discuss the sources of error that made the difference between the two methods. The required background for understanding the project work and the set-up of the necessary analysis, including the scripts used will be presented.

The methods being used will be described along with how results were compared between the two analysis methods. Finally results from the analyses will be presented and discussed.

2 Background information

This chapter presents background information on aspects that will be helpful when reading the report. Important aspects will be explained further later in the report.

2.1 Jacket supported offshore wind turbine

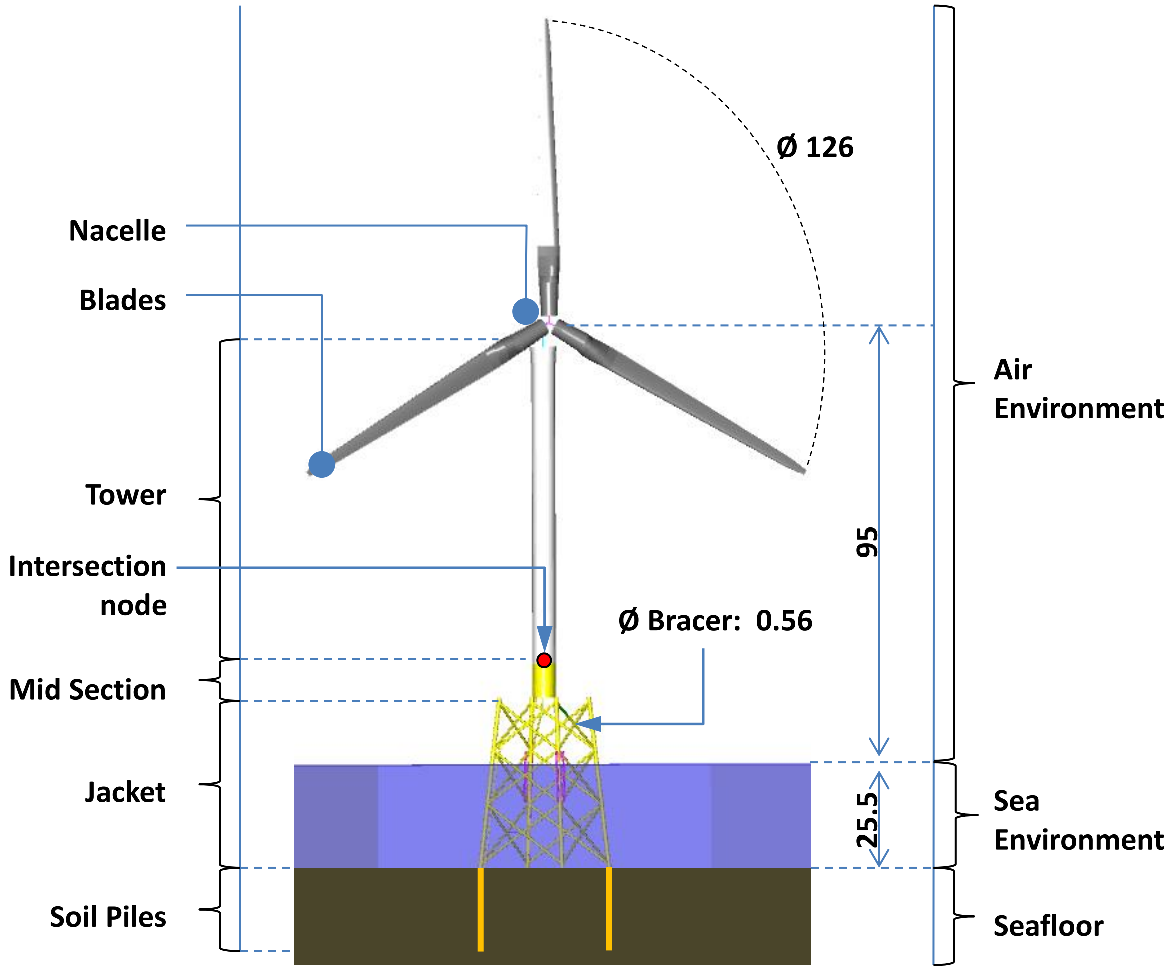


Figure 2 - Basics of the Offshore Wind Turbine of this master project

Figure 2 - Basics of the Offshore Wind Turbine of this master project illustrates the FEDEM model of the complete offshore wind turbine model. This analysis model was used when doing the full coupled analysis. This analysis model is capable of loading the jacket structure with environmental loads from the sea environment, and loading the turbine blades with turbulent wind from the air environment.

2.2 The Wind Turbine

Figure 3 - Ormonde Wind Turbine – ref (Ormonde Wind Turbine - Website)

The purpose is to harvest power from the wind.

Offshore wind turbines might be floating or fixed. The turbine in question is designed for shallow waters and fixed to the ocean floor via the midsection to the jacket.

The blades generate the rotating movement which provides a torque to the generator that in turn produces electrical power. The generator is placed inside the nacelle, which is the enclosed compartment on top of the tower. The nacelle is the transition between the blades and the tower. The generator, the control system and the brakes are inside the nacelle.

2.2.1 Midsection and tower

Figure 4 - Ormonde Midsection – ref (Ormonde Midsection - Website)

The midsection is a transition piece allowing the turbine tower to be attached to the jacket structure. The structure of the midsection and tower is included but not considered in this master project.

2.2.2 Jacket

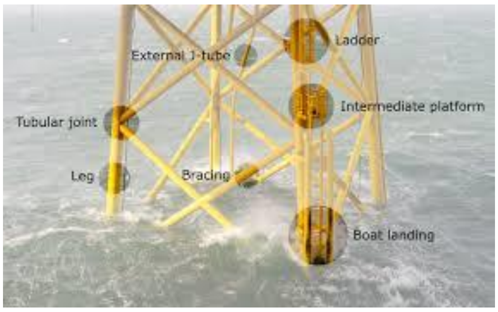


Figure 5 - Ormonde Jacket – ref (Ormonde Jacket - Website)

An offshore wind turbine can be attached to the seabed in several ways. Monopiles and jacket like structures are common. The wind turbine in question is carried by a jacket structure made by steel tubes fixed to the seabed utilizing soil piles. Jacket structures can be advantageous due to their stiffness/mass ratio.

2.2.3 Soil Piles

To be securely held in place, the jacket is mounted into the seabed using soil piles. The properties of the clay, sand and soil at the seafloor, in combination with the structural properties of the piles, strongly influence the response of the offshore structure when subjected to offshore loads.

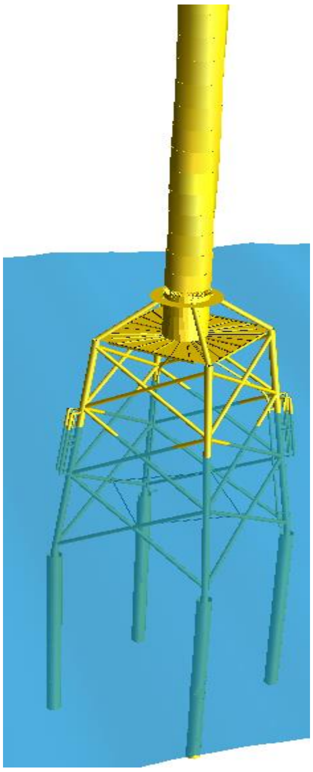


Figure 6 - Soil piles - USFOS model

2.3 Analyses software and methods

The structure of the offshore windturbine is not overly complex. However, the dynamic loading due to the sea and air environment, the dynamics resulting from the rotating wind turbine blades and the wind turbine control system is very complex. To be able to evaluate the design of an offshore wind turbine, one need to have tools that aids doing the job;

- FEA code
- Representative static loads
- Representative variable loads (impact, installation loads etc.)
- Representative environmental loads (for the right region of the world)
- Standards (regulations, knowhow and best practice)

2.3.1 ANSYS ASAS L/NL

ANSYS ASAS is the analysis tool used by TDA AS. This is a renowned marine and offshore code. It is commonly used in the design of offshore wind turbines in combination with other software, allowing coupled analysis to predict results when the structure is loaded by wind and waves. The pre- and postprocessing is text based, but postprocessing can be performed using ANSYS postprocessing tools. A big advantage in ASAS is that it allows for automatic matrix extraction for stiffness-, mass-, and damping matrixes.

For more information, refer to (ASAS Software - Website)

2.3.2 FEDEM Windpower

FEDEM Windpower is a quite new software for linear dynamic analysis, with the capability of including dynamic wind and wave loads during the analysis time interval. Pre- and postprocessing is done using the GUI which gives access to most functionalities. Results can be exported automatically if other postprocessing is desired. The code is quite fast.

For more information, refer to (FEDEM Windpower - Website)

2.3.3 USFOS

USFOS (Ultimate Strength For Offshore Structures) is a renowned software for nonlinear dynamics and collapse analyses. It has multiple special features for capturing effects normally experienced in jacket-type structures. Dynamic wave and wind loads can be utilized in USFOS, but only dynamic waves are used in USFOS during this project. Pre-processing is carried out using textfiles (or excel), while postprocessing can be done by extracting results files or using the postprocessor Xact.

For more information, refer to (Main Page: USFOS)

2.3.4 Fully Coupled analysis method

There are different ways of doing FEA analyses, and analysis codes may be different considering the kind of analysis to be done. Some analysis codes excel in linear dynamics, while others are specialized for analysis involving nonlinearities in a type of structures. Likewise, other analysis deals with fluid flow or fluid structure interaction.

In the fully coupled analysis, as carried out in this project, “all” load types (structural dynamics, wind- and wave dynamics) are considered within the same code.

The fully coupled analysis tool used in this project is FEDEM.

2.3.5 Sequentially coupled analysis method (Simplified explanation)

Sequentially coupled analyses are analysis loops, where (in our case) the total load on the structure is obtained after running a sequence of three analyses. Here two analysis codes, *A* and *B*, are used. *Code A* handling the hydrodynamic loads (waves, current, marine growth, buoyancy) and *code B* handling the aero-elastic loads (turbulent wind). *Code A* was used to obtain wave loads that could be used to load the simplified structure including wind turbine in *Code B*. After running *Code B* the global structure translations were found, and *Code A* could be rerun with global transformations and wave loads. The rerun analysis provides a picture of the local phenomenon in the foundation structure.

Running sequentially coupled analyses in this project was done using USFOS (*code A*) and FEDEM (*code B*). The ordinary sequentially coupled analysis method is described in detail in chapter 4.1.

2.4 Offshore loads

Load conditions typical to occur offshore:

- Static loads (structural mass, and weight of equipment)
- Ship impact
- Earthquake
- Wave and current loads (Hydrodynamic loads)
- Buoyancy and marine growth
- Wind loads

2.4.1 Offshore loads specific to offshore wind turbines

- Dynamic loads due to the rotation of the blades
- Dynamic loads due to gyro forces
- Dynamic loads due to the offshore wind turbine control system (adjustments of pitch and yaw)

2.4.2 Handling wind in offshore wind turbine analyses

Turbulent wind can be generated by internal or external codes. These wind fields describe the fluctuating wind on the area occupied by the rotor blades over the analysis time interval. The windfield is in turn interpreted as load-time vectors on blade elements thus loading the offshore wind turbine blades.

2.4.3 Handling waves and current in offshore wind turbine analyses

Waves and currents are transformed into structural loads by the analysis software. The most appropriate method to do this is generally dependent on the ratio between structure diameter and wave parameters, along with water depth. In this project where the submerged structure is a slender one, Morison’s equation can be applied. Morison’s predict the horizontal and vertical forces acting on the structure in terms of inertial and drag force. The coefficients of drag and added mass (inertia) play a crucial role in this equation.

2.4.4 Damping

Damping is all about transferring energy. Damping in a jacket structure will reduce the vibrational amplitudes and cause decay if excitation stops. Vibrations can be seen as cyclic loads, and can be the dominating loads as for a jacket designed to carry a wind turbine. Such a jacket will typically experience dynamic loads from both wind and waves in addition to the ever repeating loads caused by the rotating turbine blades. The real damping, and the values of damping applied in an analysis model, needs to fit as well as possible to real-life conditions. This is in order to not over- or underestimate the fatigue loading, which will result in a design too heavy and expensive or too fragile giving a too short lifespan.

In the analysis models used in this project two types of damping was applied, namely Rayleigh damping and hydrodynamic damping.

Rayleigh damping:

Applied to structural elements and divided in a part proportional to the stiffness matrix and a part relating to the mass matrix. This is a simple way to define internal damping in a structure, but not well suited for tuning multiple modes.

Hydrodynamic damping:

Applies to objects subjected to fluid flow. It was only considered for the submerged parts of the structure. It consists of an approximated drag coefficient (Cd) which defines the damping force opposite to the direction of the relative fluid structure velocity.

2.5 Vibration

2.5.1 Whiskey Vibrations

The expression *Whiskey Vibration* has a non-technical origin. It is used for local out of plane vibrations of the bracers. The name *Whiskey Vibration* comes from the price (a case of whiskey) to those who can prove whether the vibrations observed in sequentially coupled analysis are real or not.

2.5.2 The causes of vibrations in analyses

When considering analysis results, there might be vibrations. The observed vibrations might be caused by:

1) Excitation; real vibrations – i.e. result of variable/cyclic loading
2) Numerical noise; vibrations caused by the analysis algorithm
3) Analysis method; amplification of amplitudes because of method used

The phenomenon of 2) was found accidentally in the beginning of the project. It was found using USFOS when relative velocities between structure and wave particles were switched on. A submerged beam element was clamped in one end and subjected to wave loads that caused escalating vibrations to the element (ref appendix “Numerical noise”, chapter 11.25).

The effect of 3) is not fully understood, but assumed to relate to the method of sequentially coupled analysis including the method to generate the **K**, **C** and **M** matrices.

2.6 Results

2.6.1 Comparing results

When comparing results from the two methods of fully- and sequentially coupled analysis, simplifications have been made. The simplifications concerns a simplified method for calculating stresses, and a shorter analysis time interval compared to what is in the standards (DNV).

To be independent of "code interpretation" of results, the beam end forces and moments have been extracted from points of interest in both analysis models. These forces and moments have been used to calculate stresses.

The calculated stress-time history has been processed using rainflow counting. In this way it is possible to establish the number of full and half cycles experienced by the structure at the points of interest.

Accumulated damage is calculated for all points of interest using the Miner-Palmgren rule. This is the basis for comparison between the two analysis methods.

2.6.2 Results from this project

The result from this project consists of data from two analysis methods trying to recreate the same loading conditions on an offshore wind turbine jacket foundation. The two methods in question are fully- and sequentially coupled analysis. The matter of interest is to see how well the two methods compare when predicting fatigue life. It is believed that the sequentially coupled analysis method will overpredict the local vibration amplitudes, resulting in a short fatigue life compared to the fully coupled analysis method.

2.7 Standards used in this project

Some DNV standards are used in this project. These standards describe the assessment of fatigue damage on offshore structures, Environmental loads and finally design of offshore wind turbine structures. The documents are listed below:

- *DNV-RP-C203 - Fatigue Design of Offshore Steel Structures*
- *DNV-RP-C205 - Environmental Conditions And Environmental Loads*
- *DNV-OS-J101 - Design of Offshore Wind Turbine Structures*

3 Method

The work on this master project can in retrospect be divided into three main parts. The different parts of the project were not necessarily done chronologically, but rather in a stepwise manner as knowledge grew and new challenges arose and needed attention.

Obtaining knowledge

- Obtain knowledge in the field of offshore, windturbine and hydrodynamics.
- Planning of what to be studied and how to approach the problem in question.
- Decision made to investigate sequential- versus fully coupled analysis method for a complete offshore windturbine.
- Investigation of existing literature on the field of offshore wind turbines and fully- versus sequentially coupled analysis.
- Methods for doing fully- versus sequentially coupled analysis on an offshore wind turbine.
- The analysis code to be used in the project had to be decided.
- Getting to know the analysis codes. Testing and comparing results.
- Creating simplified tests for part of the jacket structure.
- Considering which analyses to be done.
- Definition of parameters to be used when comparing the methods of fully- versus sequentially coupled analysis.
- Retrieval of all input and parameters to be used in the analysis.

Development of tools

- Generation of FEA models including input-files / model-files / event-files.
- Creating scripts for running and extracting results from analysis.
- Creating a method allowing sequentially coupled analysis using USFOS and FEDEM. Demanding the extraction of stiffness-, mass- and damping-matrix for the jacket structure. Done by using USFOS.
- Creating the fully coupled analysis model in FEDEM.
- Creating the FEA models in FEDEM and USFOS needed for running a sequentially coupled analysis.
- Creating scripts for handling input/output during sequentially coupled analysis.
- Creating scripts for results extraction from fully- and sequentially coupled analysis, and excel sheets to calculate beam end stresses.
- Creating Matlab scripts for Rainflow Counting, damage calculation and FFT (signal processing).

Execution of jobs & reporting

- Debugging FEA models and scripts
- Performing Modal analysis in USFOS to find Eigen modes of the jacket and tower structure.
- Performing all dynamic analysis, both fully- and sequentially coupled analysis on complete offshore wind turbine.
- Performing sensitivity study on sequentially coupled analysis by scaling of $[M]$ and $[C]$ matrixes respectively.
- Generating the results from all analysis
- Making the report

Working with this project, several obstacles were encountered. These have been problems with handling of large amounts of data, hardware problems and problems with the analysis codes. Some discrepancies between code and code manuals have also been observed.

The project progress, and some of the obstacles met, are illustrated in *Figure 7 – Progress history*.

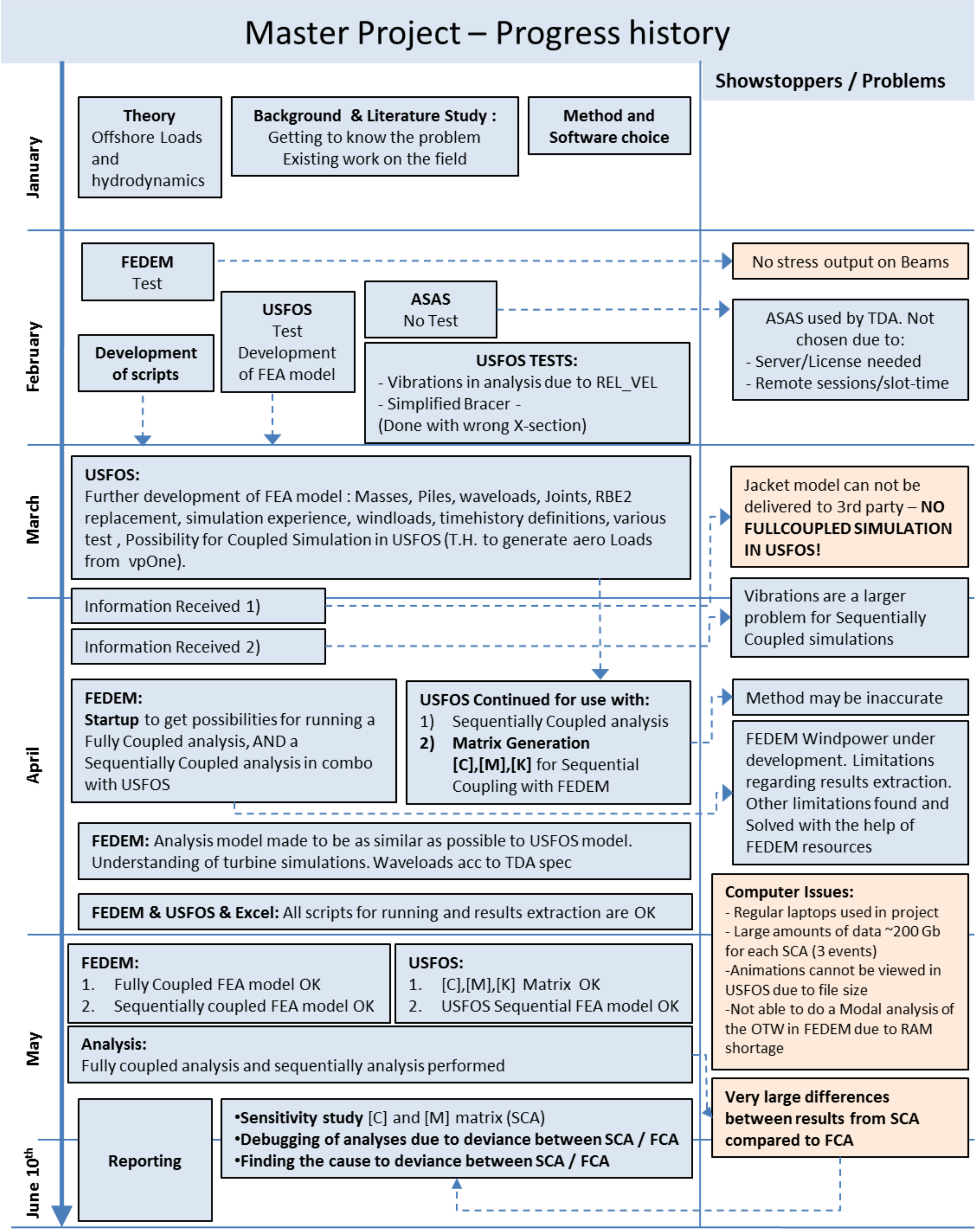


Figure 7 – Progress history

4 Literature study

The literature study is basically divided into four topics.

The first section, chapter 4.1, the method behind the sequentially coupled analysis is discussed. The method is divided into a sequence of different analyses. It is thus explained how to set up every analysis, what the input is, and what output to be extracted. This study is used as basis in the following understanding and recreation of the method in the project.

The second section, chapter 4.2 and 4.3, is a summary of the various offshore loads and a description of the type of loads taken into consideration during the analyses.

The third section, chapter 4.4 to 4.10, some papers concerns different aspects dealing with local vibrations. The fully – and sequentially coupled method is mentioned. These provide a little impression of what to take into account during the analyses. They also highlight which problems and differences that can be expected from the methods.

The last section, chapter 4.11, concerns the signal processing for the results of the analyses. A basic understanding of signal processing is important to be able to interpret the outcome of the analyses.

4.1 Sequentially coupled analysis method (SCM)

The sequentially coupled method used is illustrated as *Figure 8 - Sequentially coupled analysis method*. The theory used in this chapter is from reference (ASAS (Non-Linear) User Manual - Version 14.0, 2011).

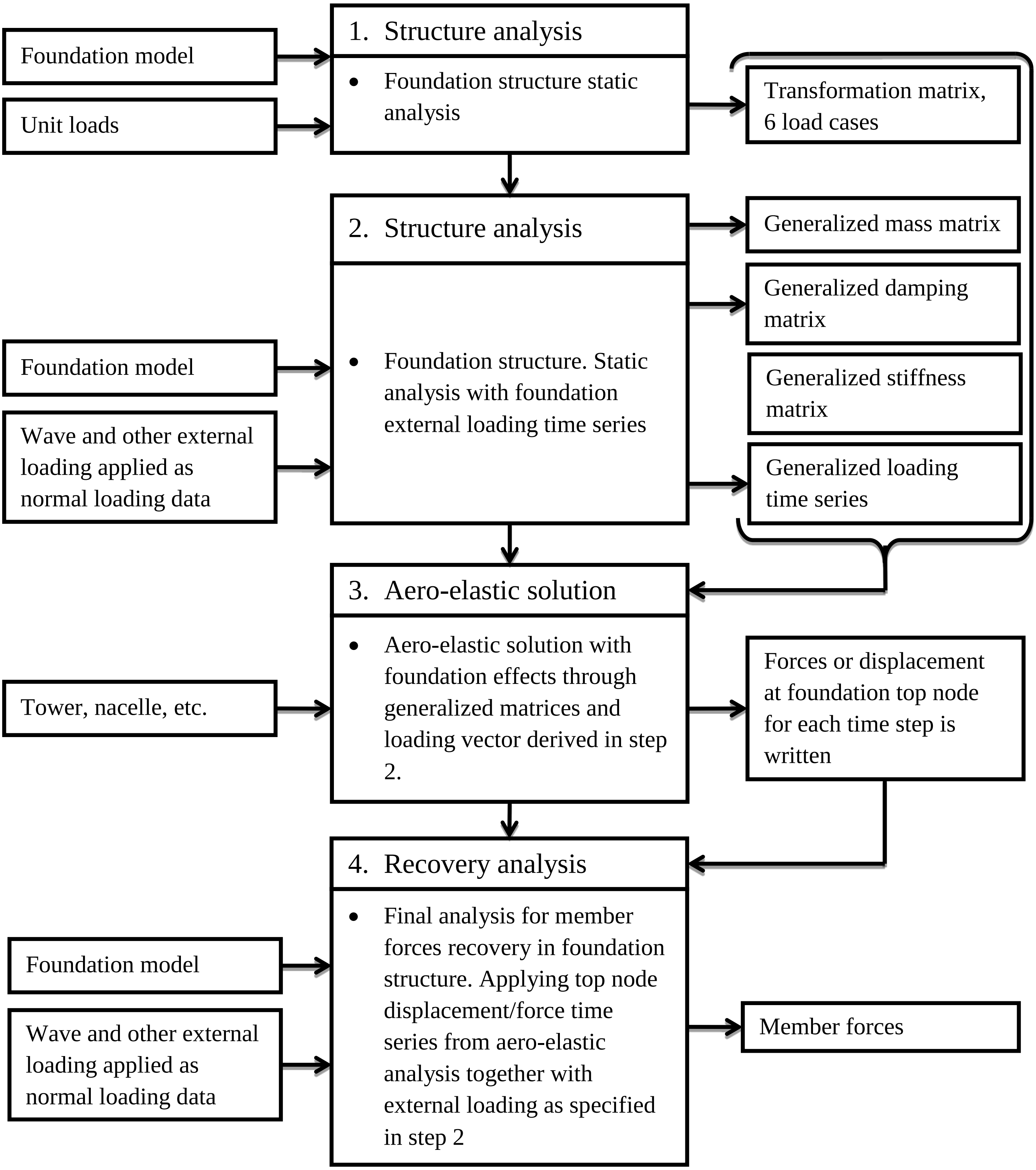


Figure 8 - Sequentially coupled analysis method

During design of an offshore wind turbine, in general, there are several companies responsible for their part of the design. The value of a company often lies in their experience and in-house development. For this reason it would not be desirable to share knowledge directly with the other parties in the project. Instead the companies send simplified results to each other as input for analyses. In general, the sequentially coupled analysis method is created to simplify the cooperation between different companies working on a project.

The method is divided into four steps. The first and second step is for extracting foundation characteristics for the tower supplier. In the third step the aero-elastic solution is executed for the tower, while the fourth step is the stress recovery analysis for the foundation.

4.1.1 SCM – 1. Structure analysis

The first step is to find the deflection shapes of the foundation structure. The structure interface/top node is subjected to unit deflections and rotation during 6 different cases. The Displacement of the interface node is tracked, and forms the transformation matrix.

4.1.2 SCM – 2. Structure analysis

Step 2 is dynamic analysis with the foundation model. The structure is loaded with all external load cases, such as wave and wind etc.

During the dynamic analysis the generalized mass, damping and stiffness matrix is extracted. The loading time series for the interface node is tracked to find the response from the external loading on the foundation structure. The loading time series is found by constraining the interface node and thereby recording the reaction force in the node.

4.1.3 SCM – 3. Aero-elastic analysis

This step is the aero-elastic analysis of the sequence. The analysis includes the full nacelle and tower assembly. The foundation characteristics are included in the analysis by applying the mass, damping and stiffness matrices for the interface node from step 2. The influence from the external load cases on the foundation structure is represented by applying the loading time series in the interface node from step 2. During the analysis the displacement or forces from the interface node is tracked.

4.1.4 SCM – 4. Recovery analysis

In this step a final dynamic foundation analysis is executed. The structure is loaded with all external load cases, such as wave and wind etc., equally as for step 2. The influence from the tower is included by applying the displacement or forces in the interface node found in step 3. The forces and dynamics may then be extracted from the analysis.

4.1.5 Remarks for the method

There are some issues to be aware of regarding the method.

When calculating the generalized mass, damping and stiffness matrices linearity in the structure is assumed. The matrices are extracted during the initial conditions in increment 1, such that the structure is un-deformed.

When calculating the force time history the dynamic displacement of the structure is unknown. The displacement in the interface node is therefore zero, likewise with velocity and acceleration. The force time history extracted has thus limited validity.

4.2 Offshore Loads

Relevant information on offshore loading and hydrodynamics utilized in the master project will be presented in this chapter. A literature study on the subject was done and involved the reading of (Faltinsen, 1998), along with (DNV-RP-C205 - ENVIRONMENTAL CONDITIONS AND ENVIRONMENTAL LOADS) and (DNV-OS-J101 – DESIGN OF OFFSHORE WIND TURBINE STRUCTURES). The theory manuals supplied by both FEDEM Windpower and USFOS also give good descriptions of the theory embedded in the analysis software. The essential regarding the theory used are described here, but refers to the above mentioned literature for details.

Extracted essentials regarding hydrodynamics and offshore loads applying the jacket structure in question are described in the following:

4.2.1 Typical loads on offshore structures for wind turbine (DNV-OS-J101, Section 4):

Permanent Loads:

- Structural mass
- Equipment etc.

Variable functional loads:

- Actuation loads resulting from the turbine control (from wind loads), braking
- Ship impact
- Crane operational loads

Environmental loads:

- Wind loads
- Hydrodynamic loads induced by current and waves, drag and inertia forces
- Earthquake loads
- Tidal effects
- Marine growth
- Snow and ice loads

Windturbine loads:

- Aerodynamic blade loads
- Aerodynamic drag forces on tower and nacelle
- Gravity loads on rotor blades (indirect load)
- Centripetal forces and coriolis forces due to rotation (indirect load)
- Gyroscopic force due to yawing (indirect load)
- Braking forces (indirect load)

As can be seen, the loading of an offshore wind turbine and its structure can be quite complex.

During this project only some of the mentioned loads will be considered:

Permanent Loads:

- Structural mass
- Marine growth

Variable functional loads:

- Actuation loads resulting from the turbine control (from wind loads)

Environmental loads:

- Wind loads
- Hydrodynamic loads induced by current and waves, drag and inertia forces
- Marine growth

Wind turbine loads:

- Aerodynamic blade loads
- Gravity loads on rotor blades (indirect load)

4.2.2 Describing the loads accounted for in the analysis:

Permanent Loads:

The structural mass of the offshore wind turbine is divided as follower on the structure:

- Nacelle and blades: 446 tons
- Tower: 240 tons
- Jacket including piles: 726 tons
- Total mass in model: 1412 tons

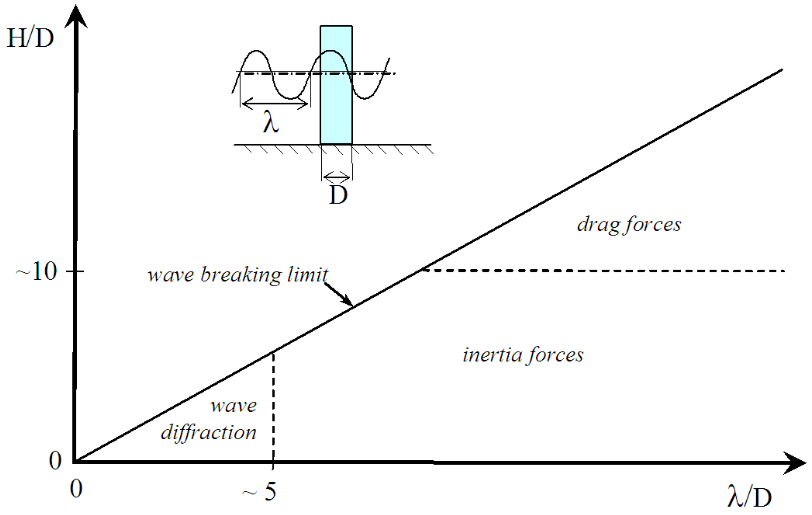
Variable functional loads:

- Actuation loads resulting from the turbine control system is handled automatically via the interaction between the control system and FEDEM for both analysis methods, and is transferred to the interaction between tower and nacelle.

Environmental loads:

- Wind loads (time history turbulence wind field) are generated by TurbSim embedded in FEDEM, and loaded during analysis. Wind loads are generated on the basis of a mean wind speed, blade design, turbulence type, hub height, analysis time interval and time step. The list of wind speeds to be analysed was received from TDA, and ranged from 12 m/s to 14m/s and 16m/s. The wind turbine setup is by default setup to comply with *Definition of a 5-MW Reference Wind Turbine for Offshore System Development* ref (Jonkman, Butterfield, Musial, & Scott, Definition of a 5-MW Reference Wind Turbine for Offshore System Development, 2009) This applies to the setup of the control system, blade design and the nacelle/generator setup.

- Hydrodynamic loads induced by current and waves, drag and inertia forces are acting on the submerged part of the jacket structure. The hydro dynamic loads received from TDA consisted of irregular wave spectrums defining wave loads correlating to the wind speeds mentioned above. The loads are calculated on the assumption of a slender structure, which calls for the use of Morison's equation. It is discussed in DNV-OS-J-101, section 4 that when $\frac{\lambda}{D} \leq 5$, the structure becomes of a considerable size compared to the waves, and will in fact disturb the wave kinematics. Then wave diffraction and wave radiation must be considered, using potential flow theory. In our case, the ratio between $\frac{\lambda}{D}$ can be determined based on the wave periods T_p and the largest submerged diameter of the jacket structure which is D=0.86m (marine growth not included).



	Hs	Tp
Event 1	2.14	6.49
Event 2	2.48	7.03
Event 3	2.82	7.55

Figure 9 - Relative importance of inertia, drag and diffraction wave forces

$$\lambda = \frac{g*T^2}{2\pi} = 1.56\ T\verb|^|2 \qquad \text{, yielding } \lambda \approx 66 \text{ for } T_p = 6.49 \qquad \text{eq. (4.2.2.1)}$$

$$\frac{\lambda}{D} = \frac{66}{0.86} \approx 77 \qquad \text{eq. (4.2.2.2)}$$

This is well inside the hydrodynamic regime that is governed by inertia- and drag forces, and thus slender theory can be applied for the analysis of the jacket structure.

Morison's equation allows the calculation of the horizontal force on a vertical element;

$$dF = dF_M + dF_D \qquad \text{eq. (4.2.2.3)}$$

$$dF = C_M \rho \pi \frac{D^2}{4} \ddot{x} dz + C_D \rho \frac{D}{2} |\dot{x}| \dot{x} dz \qquad \text{eq. (4.2.2.4)}$$

Were C_M and C_D represents the inertia- and drag coefficients respectively. For analyses performed in both USFOS and FEDEM, the values of C_M and C_D supplied by TDA are used; $C_M = 2.0$ and $C_D = 1.3$ on the whole water depth.

During the analyses, the current velocity is zero.

4.3 Vortex induced vibrations

Vortex shedding is the instability of the wake behind objects subjected to fluid flow. Since this project concerns vibrations of submerged bracers, it is of interest to see if such vibrations may occur due to vortex shedding. The analysis software used in this project does not predict vortex shedding, and it is of interest to see if this omission is of importance.

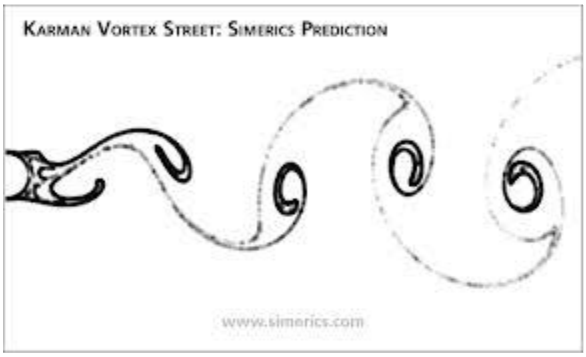


Figure 10 - Illustration of vortex shedding

Vortex shedding exerts oscillatory forces on the object in question. These forces appear in both lift and drag direction. The magnitude of the forces might be of interest, but first of all one has to consider the vortex shedding frequency. The vibration frequencies of interest regarding the bracers lie in the range 2-5 Hz.

It is possible to find the vortex shedding frequency using following equation:

$$St = \frac{f_v D}{U_\infty} \qquad \text{eq. (4.3.1)}$$

Here St is the Strouhal number, which can be set equal to 0.2. This value is considered to be reasonable for subcritical flow, ref (Faltinsen, 1998) page 186. The oscillatory frequency is noted as f_v, while D is the member diameter and U_∞ is the velocity far upstream.

Information on the velocity is needed. A plot of the wave kinematics after an analysis in USFOS for Event 1with corresponding velocities is illustrated in *Figure 11 - Wave velocity profile for bracer junction*. Remember that no current is present during this analysis.

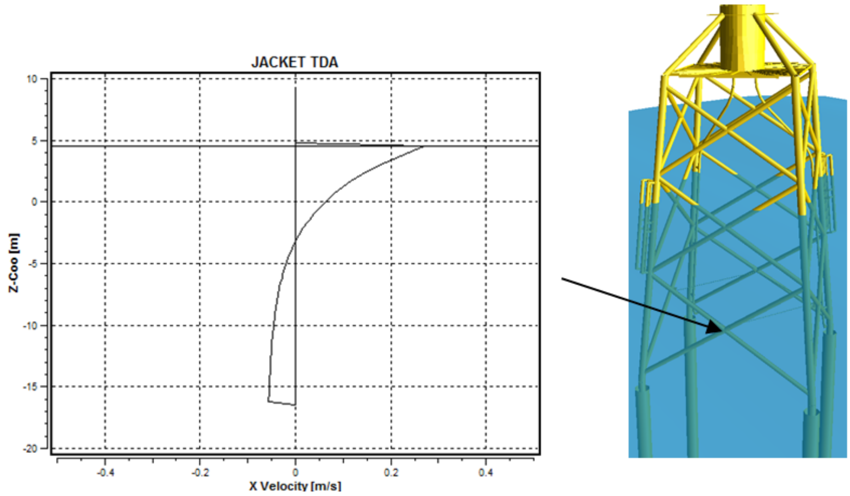


Figure 11 - Wave velocity profile for bracer junction

One can observe that the velocities are quite low. Using maximum velocity equal to ~0.3 m/s yields an oscillatory frequency resulting from vortex shedding equal to:

$$f_v = \frac{U_\infty St}{D} = \frac{0.3*0.2}{0.86} = 0,07\ Hz \qquad \text{eq. (4.3.2)}$$

The predicted frequency of the vibration in question is far below the frequency range we are interested in. The above formula is considering constant velocity while the wave kinematics velocities on the graph are transient and vary over time according to the wave spectrum.

The conclusion from this is that the effect from vortex shedding could be neglected as cause of high frequency vibrations under the present circumstances. However, with current and larger waves, the effect may be present.

- Marine growth is accounted for in all analyses. The marine growth has a uniform thickness over the whole depth range equal to 0.1m which complies with *DNV-OS-J101, section 4*. The thickness value is received from TDA. The effect of marine growth is an increasing diameter and an increased roughness of submerged members and members in the splash zone. It has an effect on the calculation of hydrodynamic forces, since the effective diameter on the structure increases along with added mass.

4.4 Validation of Offshore load simulations using measurement data from the DOWNVInD project

Refer to reference: (Seidel & Ostermann, Validation of Offshore load simulations using measurement data from the DOWNVInD project)

This paper compares results from the sequentially coupled method with the fully coupled method. Ideally the analysis results were supposed to be validated through the measurement campaign of DOWNVInD project, but due to unexpected events this ability became limited.

The paper point out that the Guyan reduction (where the substructure is reduced to mass, stiffness and damping matrices accompanied by wave loading force history) is only accurate at static loads. This applies especially for substructure with large masses inside the substructure.

For the fully coupled analysis method the idea is to link and run the two software codes together. In practice this is done by sending information into the code and sending the results back for every time step of the analysis. In this way the codes take each other into account when executing the calculations. The paper also specifies it is important to consider the validity of the controller implementation and behavior. Some settings perform poorly with the implemented method since the controller is one increment behind the simulation.

When executing an FFT analysis on the tower bottom, there were quite consistent results between sequentially- and fully coupled analysis method. The exception was in the second global mode, in the frequency range 1.5-2 Hz, where there were some differences.

When calculating FFT on out of plane member forces there were major differences. There were good fit up to 1.5 Hz, but at 2.4 Hz there is a large response.

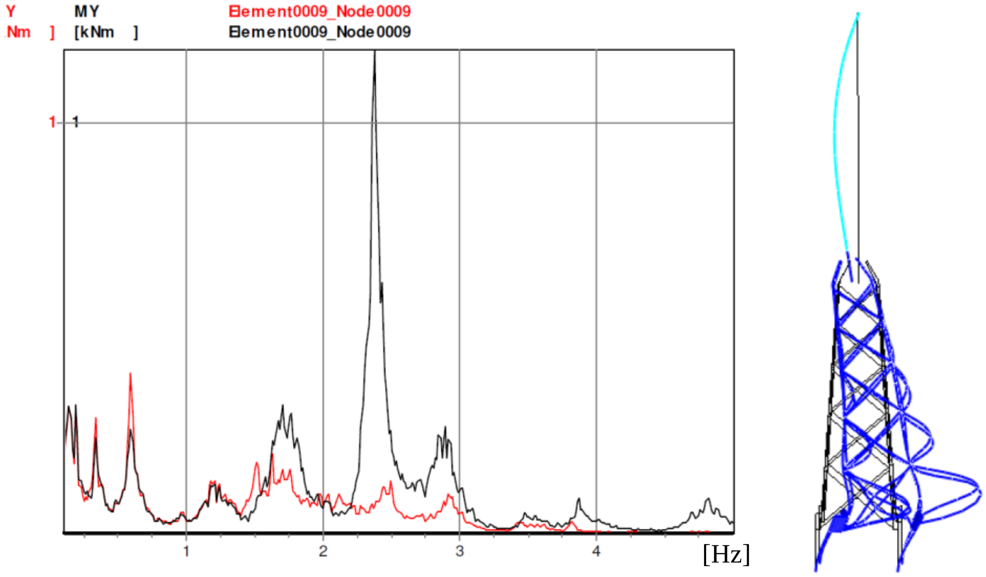


Figure 12 - Comparison of SCA and FCA for local jacket member forces – Reference (Seidel & Ostermann, Validation of Offshore load simulations using measurement data from the DOWNVInD project), page 8

4.5 Integrated analysis of wind and wave loading for complex support structures of Offshore Wind Turbines

Refer to reference: (Seidel, von Mutius, Rix, & Steudel)

The paper is concerning the sequentially coupled analysis method. The paper addresses some challenges in terms of the execution of the method. The allegations especially worth noting are the following:

When executing the stress recovery sequence the input to the interface node may either be added as force controlled or deformation controlled. However, if it is force controlled the stress recovery could only be run as a static analysis. The damping from the turbine (mostly aerodynamic damping) will not be present in the jacket foundation model. This will lead to exaggerated deformations during a dynamic stress recovery analysis. When the analysis is run as deformation controlled, it could be run as both static and dynamic.

If a non-linear foundation is modeled special care has to be considered whether the code accepts nonlinearities or not. This is a code limitation, and not the method in specific.

4.6 The full-height lattice tower

Refer to reference: (Musculus & NTNU-Trondheim)

This paper discusses the full height lattice tower and has a couple of interesting findings regarding jackets used for offshore wind turbines. One is that compared to monopoles, jacket structures are stiff and prevents global vibrations. The paper shows that local vibrations are important and must be considered during the design. It is also shown that for a more severe sea state local vibrations are less important. Results from an optimization study on a lattice tower with varying cross-sectional area are described. The study shows that FLS can be improved by varying cross-sections.

4.7 Offshore Code Comparison Collaboration Continuation (OC4)

Refer to reference: (NREL)

This paper is presenting results from the OC4 - “Offshore Code Comparison Collaboration Continuation” project which did code-to-code comparison of offshore codes. All codes used and mentioned in this project are tested and discussed in this paper (FEDEM, USFOS, ASAS). The paper points out an issue also existing in this master project, namely the overestimated weight due to overlapping members at joints where beam elements connect at the intersection point of the centreline. This also leads to an overestimation of marine growth and hydrodynamic mass. The report shows a small but noticeable discrepancy in hydrodynamic added mass, and mass of marine growth between the analyses codes used in this project. Both USFOS and ASAS predict slightly more mass than FEDEM, and might be a source of discrepancy since the dynamic response of the structure strongly relates to the masses. A comparison of out of plane vibrations for a bracer is shown. The results from FEDEM, USFOS/vpOne and ASAS are remarkably similar. The paper stresses the fact that these analyses are complex, and addresses a need for a further study of the out of plane local vibration phenomenon.

4.8 Comparison of different approaches to load calculation for the OWEC Quattropod® jacket support structure

Refer to reference: (Muskulus, et al.)

This paper describes different approaches for analysing the same jacket structure as we are dealing with in this project. The different approaches are sequentially- and fully coupled analysis. Sequentially coupled is performed by combining ASAS, FEDEM and BLADED. The study summarizes by telling that wave load calculation seems to differ in different codes, and that a fully coupled analysis is generally more favourable for fatigue cases due to significant differences. Sequentially coupled analyses predicts up to 200% increase in fatigue loads for some load cases.

4.9 Wind Turbine for Offshore System Development

> Refer to reference: (Jonkman, Butterfield, Musial, & Scott, Wind Turbine for Offshore System Development)

This report describes a 5-MW baseline conventional wind turbine which is a three-bladed upwind variable speed and variable blade pitch controller developed by NREL (National Renewable Energy laboratory). The aim of this design is to support concept studies when assessing offshore wind technology. The study describes and defines all aspects of an offshore wind turbine and includes standard blade, generator, control system and tower. (Note: The results from this study is implemented as default setup values in FEDEM Windpower, allowing the user to start simulating right away)

4.10 Mechanical Vibrations

> Refer to reference: (AA242B: MECHANICAL VIBRATIONS - Direct Time-Integration Methods) based on reference: (Geradin & Rixen)

These slides describes the:

1. Stability and accuracy of time-integration operators
2. Newmark's family methods
3. Explicit time integration using the Central Difference algorithm.

Related to this project, the slides describe the stability condition $h_{cr} = \frac{2}{\omega_{cr}}$ based on the critical time step h_{cr} and the highest frequency ω_{cr} contained in the model. It shows that an exact solution is obtained when the time step h is equal to the critical time step, while decaying vibrations occur if $h < h_{cr}$. If $h > h_{cr}$, one might observe escalating vibration amplitudes.

4.11 Fourier Transforms, DFTs, and FFTs

Refer to reference: (Cimbala, 2010)

This paper describes the basics of Fast Fourier Transforms (FFT) and how to apply the method using Excel. The method (FFT) is used for analysing a time-response signal in the frequency domain, i.e. to find out which frequencies are contained within the signal in the time domain. This paper also explains common problems encountered when using FFT on signals, like aliasing and leakage. Aliasing is not considered in this thesis, but leakage was an issue. Leakage is known to occur in FFT analysis when the signal processed is not periodic, which is the case here. Leakage causes the spike of the FFT signal to decrease in amplitude, and the energy will be spread nearby the frequency of the actual signal frequency, causing the spike to appear more like a pyramid. The leakage can be problematic, because it makes it harder to interpret the frequency content of the signal, and amplitudes from the FFT will not be comparable to the amplitudes in the real signal.

5 Analyses

5.1 Analysis Software and Analysis setup

5.1.1 Purpose of the analysis study

The purpose of this analysis study is to do a correlation between two analyses methods as it is believed that out of plane bracer vibrations increases if a sequential analysis is preformed compared to a fully coupled analysis. The two methods, fully coupled- and sequentially coupled analysis are often used in the design of offshore wind turbines including the supporting structures. If increased out of plane bracer vibrations occur, the difference will be measured by quantifiable values that can be compared between methods. This measure for comparing can be a fatigue damage parameter or a method for comparing the vibration signal.

In order to do a representative correlation of methods, the input and FEA needs to be equal for both methods. This is not necessarily an easy task to accomplish since different FEA software, although quite similar mathematics, might be quite different in the way input are defined and handled.

5.1.2 Overview of analysis tools used

During the analysis study, which has been the most work intensive part of the master project, mainly two analysis software has been used, namely USFOS and FEDEM Windpower 7.02. During the start-up phase of the project, several FEA codes were considered that could handle dynamics and hydrodynamics. The candidates were ASAS, USFOS, vpOne and FEDEM Windpower. When choosing the analysis tools to be used, the choice was based on:

1) Possibility to install software on the students computers
2) Simulation can be run on the students computers
3) On-site resources (at NTNU or in Trondheim) that could provide guidance

Based on the above criteria, USFOS and FEDEM Windpower were chosen, even though ASAS is being used by TDA. It was considered a bit troublesome to run ASAS since it implied using remote connection to TDA`s servers, and that the resources was located in Oslo, quite far away from Trondheim. An important advantage in choosing USFOS was that the candidates had access to the module vpOne via a resource outside NTNU. This would make it possible to run both sequential- and fully coupled analysis with aerodynamic set up and control system within the same platform. FEDEM was equally chosen as an alternative due to the capabilities of running dynamic analysis of complete windturbines including aerodynamic set up and control system and possibilities of importing the jacket spaceframe.

USFOS (Ultimate Strength For Offshore Structures) is an analysis tool specialized in non-linear static- and dynamic analysis of space frame structures. The software was developed by SINTEF Marintek, and is used by several oil companies and DNV, among others. USFOS has been around for a while, and has many advanced features beneficial for the user. USFOS is a text based simulation tool, were the input files was generated using excel. USFOS is featured with a postprocessing tool, Xact which is a GUI that allows the user to extract and view results from an analysis model.

FEDEM Windpower 7.02 is an analysis tool specialized for dynamic analysis of complete on- and offshore windturbine systems. In this software the user can access and tune all important aspects of the variables needed to run a complete simulation on a windturbine. This includes wave loads on structure, turbulent wind acting on the turbine blades and a control system. The software features a GUI in which the needed information can be defined by the user. FEDEM Windpower 7.02 is a fairly new software built on the FEDEM platform.

In the beginning of the project, FEDEM was intended to be used as comparison to USFOS/vpOne. Due to restrictions later on, the model of the jacket structure could not be shared or handed over to 3rd party. The result of this was that USFOS could not be used in combination with vpOne during the project. A workaround had to be done, resulting in using FEDEM as the source of the fully coupled analysis and a basis for the inclusion of wind loads in the sequentially coupled analysis together with USFOS. The main challenge was how to represent the jacket structure in a simplified way, conserving its complex properties in the analysis step within FEDEM. This was achieved by generating the stiffness matrix (6x6), damping matrix (diagonal) and mass matrix (diagonal). The stiffness matrix was extracted using static loads and responses, the damping matrix using dynamic analyses, while the mass matrix by using gravity fields in USFOS. The procedure is described in chapter 5.6. The resulting matrixes contained data that was included in a FREEJOINT in FEDEM.

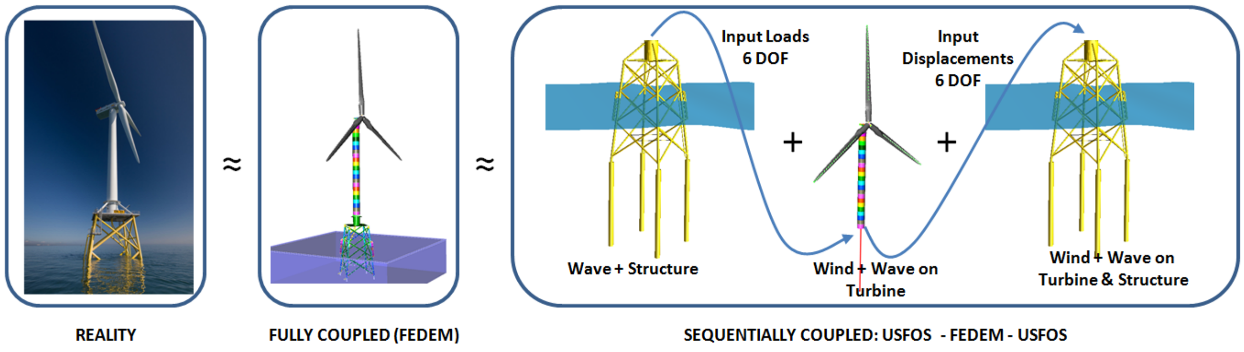


Figure 13 - Analysis representation

5.1.3 Wind Turbine and control system setup

FEDEM Windpower is equipped with a module that takes care of the needed parameters to run a windturbine analysis. During this project all aspects of the windturbine setup regarding blade definitions, nacelle set up (generator and gears) and windturbine control system has been left to default values for a 5MW windturbine. These values are in agreement with a reference study on a 5MW windturbine offshore system. More information on the “Definition of a 5-MW Reference Wind Turbine for Offshore System Development” can be found at reference: (Offshore wind: NREL Website).

5.2 Loads, hydrodynamics and simulation events

5.2.1 Received load data

The load data used in the project has been provided by TDA AS. The load data received was contained within ASAS input files. These load data includes three load events containing wave loads, sea level definitions, marine growth, buoyancy, wind speeds and hydrodynamic parameters. The load data defines three wind speeds (12, 14, 16 m/s) and the corresponding irregular wave definitions.

5.2.2 Waves in USFOS and FEDEM

The load data was first incorporated into USFOS. Not all parameters defining irregular waves were equal between ASAS and USFOS. The definition of irregular waves in FEDEM Windpower was a bit cruder than in USFOS, leaving out some parameters compared to USFOS. The wave generated in FEDEM should ideally have been compared to an USFOS wave, to study the differences. The differences – if any, could maybe lead to different results in USFOS/FEDEM. However, since the energy contained in both wave spectrums should be the same over time it is assumed that the differences have little effect on overall results.

The way USFOS and FEDEM implements wave loads on beam elements might be different. To get a representative wave loading on a beam element structure in FEDEM it is important to have a sufficiently refined element mesh. It is not known how wave discretization is performed in USFOS. The beam element model of the Jacket structure was received from TDA AS and is used "as it is" with the defined mesh density.

The wave parameters are given in *Table 1 - Wave parameters*. Only highlighted values were changed between events.

Table 1 - Wave parameters

USFOS SPEC	Type	Spectype	Hs	Tp	Dir.	Seed	Surflev.	Depth	nFreq	Tmin	Tmax	grid	Gamma
USFOS	spectr	Jonswap	2.14	6.49	0	2	4.5	25.5	120	1	11.5	2	1.913
FEDEM	Jonswap spectrum		2.14	6.49	0,0,0	2	4.5	25.5	120	1	11.5		1.913
FEDEM SPEC	Func. Type		Hs	Tp	Wave Dir	Rnd Seed	MSL	Depth		Tlow	Thigh		

Wave parameters used in the different events are given in *Table 2 - Event wave parameters*.

Table 2 - Event wave parameters

	Hs	Tp	Tmin	Tmax	Gamma
Event 1	2.14	6.49	1	11.5	1.913
Event 2	2.48	7.03	0.5	12.5	1.852
Event 3	2.82	7.55	0.5	13.5	1.785

5.2.3 Hydrodynamics in USFOS and FEDEM

Both in USFOS and FEDEM, the hydrodynamic parameters were defined to be equal. Compared to the definitions in ASAS, the definitions in USFOS and FEDEM were a bit simplified regarding which submerged element groups considered buoyant or filled. In USFOS and FEDEM all submerged elements were considered buoyant since the element groups defined in ASAS had no element lists (unknown elements).

Hydrodynamic drag- and mass parameters was defined along with marine growth influencing the hydraulic diameter and increased weight on structure. In USFOS the parameter REL_VELO (relative velocity) is switched on. This accounts for relative velocity between structure and wave particles (important when calculating drag forces and hydrodynamic damping). In FEDEM relative velocity is on by default when defining hydro dynamic parameters on beam elements imported via the "SPACE FRAME" option.

Table 3 - Hydrodynamic Parameters

	Cd	Ca	Cm	Marine growth (t)	Marine growth (dens)	Water (dens)
USFOS	1.3	-	2	.1	1024	1024
FEDEM	1.3	1.0	2	.1	1024	1024

5.2.4 Structural damping in USFOS and FEDEM

Rayleigh damping is equally defined in USFOS and FEDEM and values are specified for the entire structure:

Table 4 - Rayleigh damping parameters

Mass Proportional	0.011600
Stiffness proportional	0.001995

5.2.5 Wind in USFOS and FEDEM

USFOS has the capability of generating static wind loads (turbulent wind fields created by 3rd party software can be loaded and used). However, the function was deactivated since FEDEM only calculates wind fields on the windturbine blades, and not on the tower and jacket structure.

5.2.6 Turbulent wind load in FEDEM

FEDEM has a dedicated module for generating turbulent wind fields. The stand-alone software TurbSim can be loaded from the GUI to define turbulent wind fields prior to the analysis. TurbSim generates files with the desired length and resolution, containing the turbulent wind fields (wind loads) needed for running a windturbine analysis.

Both fully coupled analysis (time increment at 0.02 s) and sequentially coupled analysis (time increment at 0.01 s) used the exact same turbulent wind field-file in order to create the same conditions. Both methods were using a wind field with a time increment of 0.02 s. In this way it was curtain that both structures were given the exact same turbulent wind loads. As mentioned previously, all values were left to defaults to maintain equal conditions for all analyses.

5.2.7 Load events in USFOS and FEDEM

It can be convenient to be able to run several analyses in sequence with different input parameters. This functionality is embedded in FEDEM, and events can be defined in a separate events-file that contains information on which parameters changes between analysis cases. This was not the case directly with USFOS, but scripts were made to generate and run multiple analyses in sequence.

5.2.8 Postprocessing in USFOS and FEDEM

Postprocessing in USFOS was done by using the postprocessor (Xact) to fetch the required output data from the analysis and save the data to files. This process was automated using scripting to save time. In FEDEM the desired output data was automatically exported after the completion of the simulation.

5.2.9 Comparison of control node displacement data generated by the project and the professionals

During the project, an input displacement history data was received from TDA AS which was used as input to their 3rd sequential step. The data contained load history for the intersection node (attachment point between windturbine tower and Jacket structure, ref *Figure 2 - Basics of the Offshore Wind Turbine of this master project*). The displacements received from TDA AS can be compared with displacement time history extracted from FEDEM after the 2nd step in the sequential analysis done in this project.

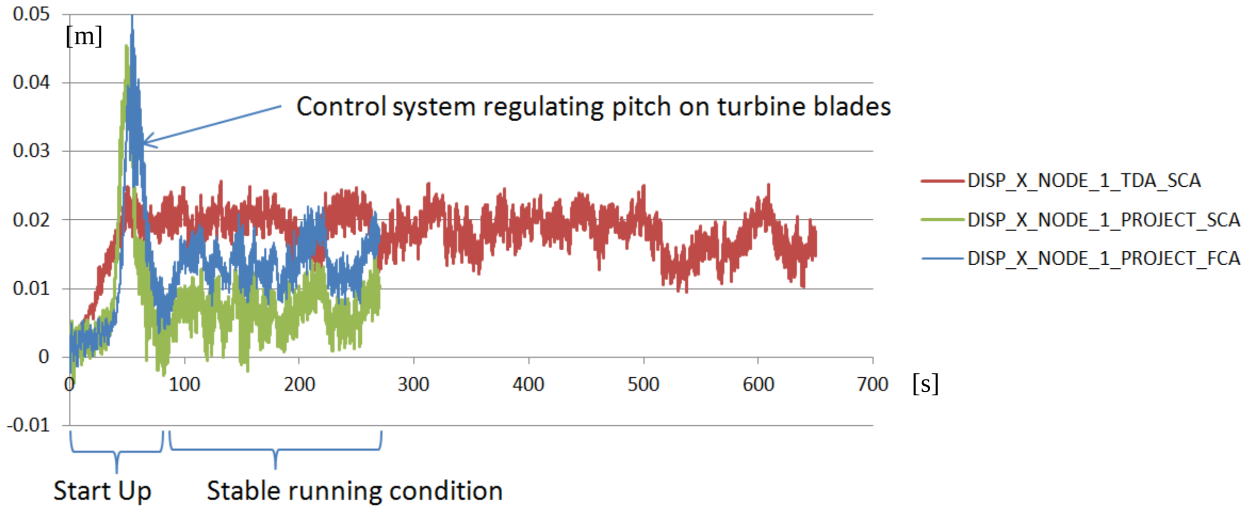


Figure 14 - Comparison - Sequentially coupled analysis - TDA vs Project

As can be seen from the *Figure 14 - Comparison - Sequentially coupled analysis - TDA vs Project*, there are several differences between the curves produced in the project and the ones received from TDA AS.

The start-up phase is very different comparing the results. The data generated by the project are similar, but different compared to the curve received from TDA AS. This is due to the difference in control system setup. The curve supplied by TDA is extracted from an analysis made by a 3rd party dealing with windturbine and control systems.

One can also observe a difference in deflection in X-Direction compared to each other after the stable running condition is reached. It seems that the overall movement in X-direction is approximately four times higher for the sequentially coupled analysis by TDA compared to the analysis in this project. The source of this offset may be due to:

1) Soil Pile stiffness might be higher in project analysis than for the results from TDA
2) Wind load on tower included in the sequentially coupled analysis by TDA – resulting in drag forces and therefore larger displacement of the control node.

Increasing Cd increases windloading
OK

Basis for analysis:

- Model J4
- Control File: CF7 = 2013-03-26
- Control File: CF10 = CF7, aero drag chg to 2.

Changes:

Analysis with Cd = 0.7 —
Analysis with Cd = 2 —

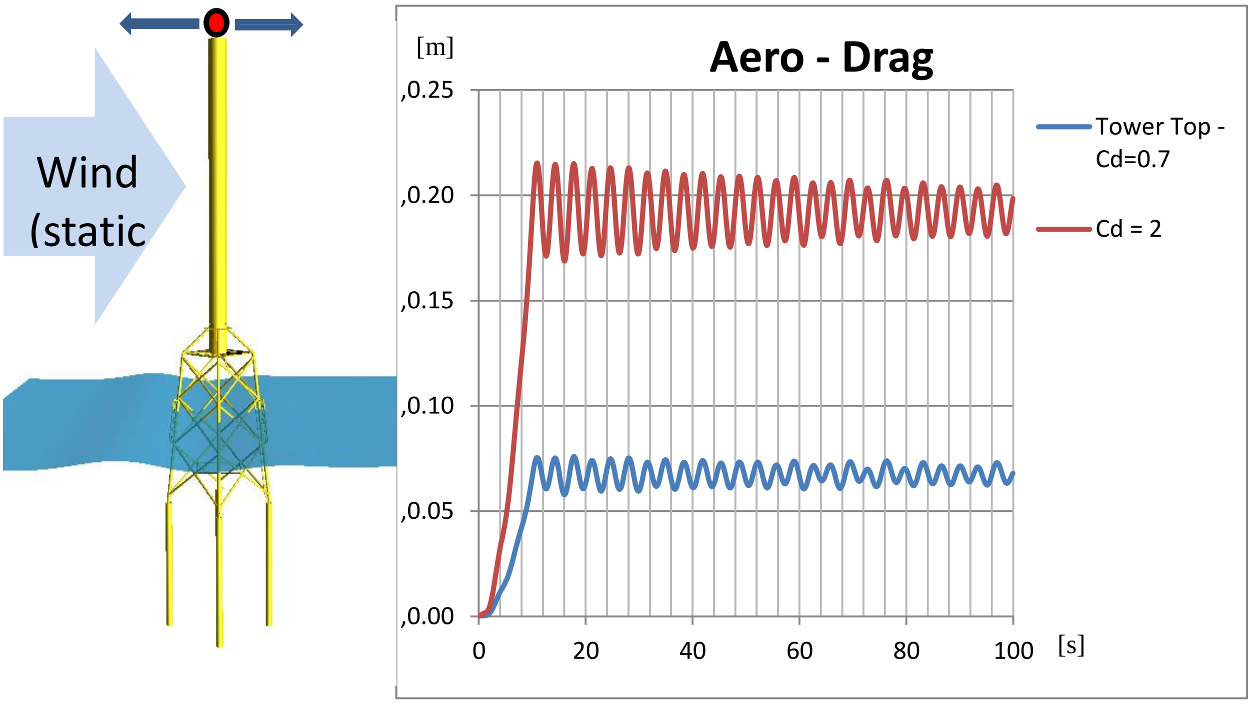


Figure 15 - Change in displacement due to change in drag coefficients – USFOS

The *Figure 15 - Change in displacement due to change in drag coefficients – USFOS* illustrates results of the effect of increasing the aerodynamic drag coefficient (Cd) from 0.7 to 2.0. The static wind speed starts at 0 increasing to 20 m/s, and the resulting displacement is measured at the tower top (red dot indicated on the figure). The case is not representative as for comparing the displacements generated from the sequentially coupled analysis by TDA against the analysis by this project. Regardless, it merely shows the significant effect from the aero drag forces on the tower and structure without considering the turbine blades.

5.2.10 Analysis Time Interval

When beginning the project the intention was to run all analysis for 600 seconds, which is the same length of the time interval as TDA AS is using.

Running for 600 seconds doesn’t seem to be very much, but with time increments at 0.02 for fully coupled analysis and 0.01 for sequentially coupled analysts, the data generated during analysis become very large. This applies especially for the sequentially coupled analyses since a complete simulation consists of three analyses, and could comprise of close to 300 GB. One other aspect was the analysis run time which is proportional to the analysis time interval. In that way it was beneficial to keep the analysis time interval quite small, but large enough to be of a “comparable size” compared to 600 seconds. A preliminary limitation in FEDEM Windpower caused the solver to stop writing beam forces and moments after 13675 increments – all other output variables continues to be logged. 13675 increments of dt=0.02 seconds equals to a total simulation time of 273.5 seconds. Since the fully- and the sequentially coupled analysis methods will be compared with a damage parameter, beam forces and moments are necessary output. There is no reason for continuing simulation without these results, and this will be the limiting factor. The time domain for the performed analysis is set to 270 seconds both fully- and sequentially coupled analysis.

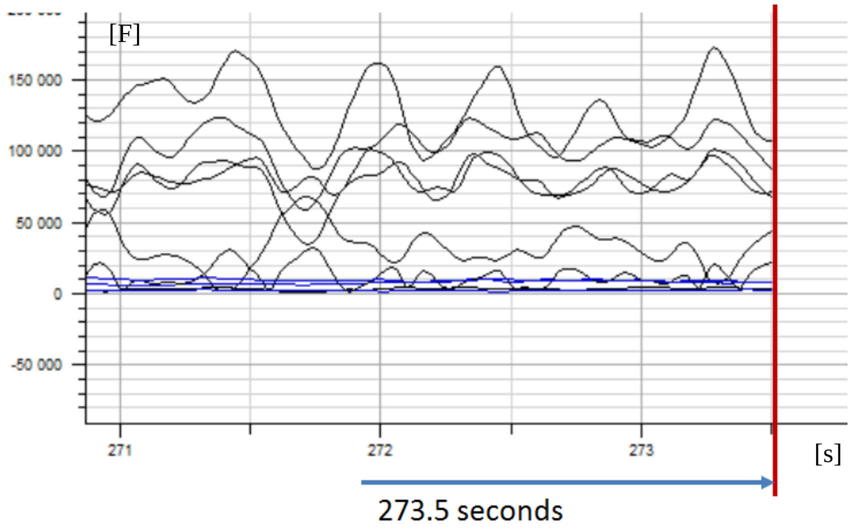


Figure 16 - Force – missing force print out after 273.5 seconds

5.3 Analysis models

The refinement of the analysis model is important parameter reflecting how well the analysis reflects the reality. To be able to compare the analysis at best basis, the models similarities and weaknesses are presented for the FEDEM and USFOS analysis model.

5.3.1 FEDEM fully coupled analysis model

Input geometry:

1) Jacket: ORMO_JACKET_NAS_10.nas.
2) Midsection: ORMO_MIDSECTION_NAS_6.nas
3) Tower: Nodes, elements, cross sectional- and mass properties received in text file.

The Nastran-files contained the definition of nodes, elements, cross sectional properties and material properties. The jacket structure was imported into FEDEM Windpower using the ”Import Spaceframe” option. The Midsection was imported as a regular part whilst the tower was included into the analysis model in the same way as the Jacket structure.

The original jacket structure contained elongated legs inserted into the soil. The legs were shortened to just reach the seabed and the full (6x6) spring properties were added to make the structure stay in place.

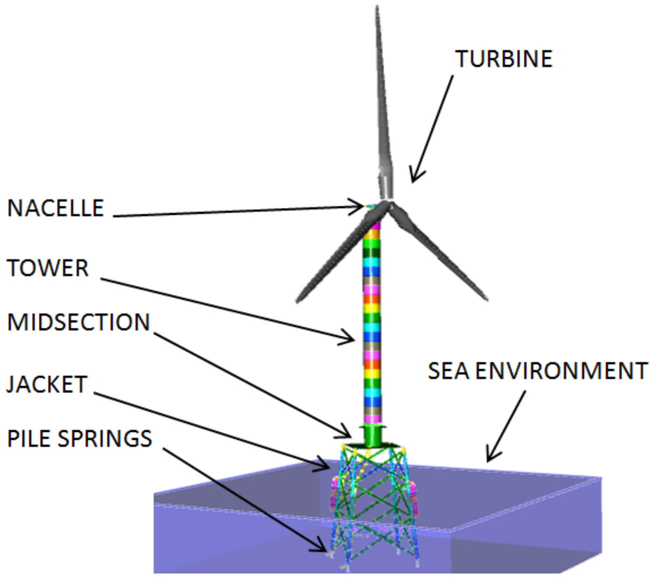
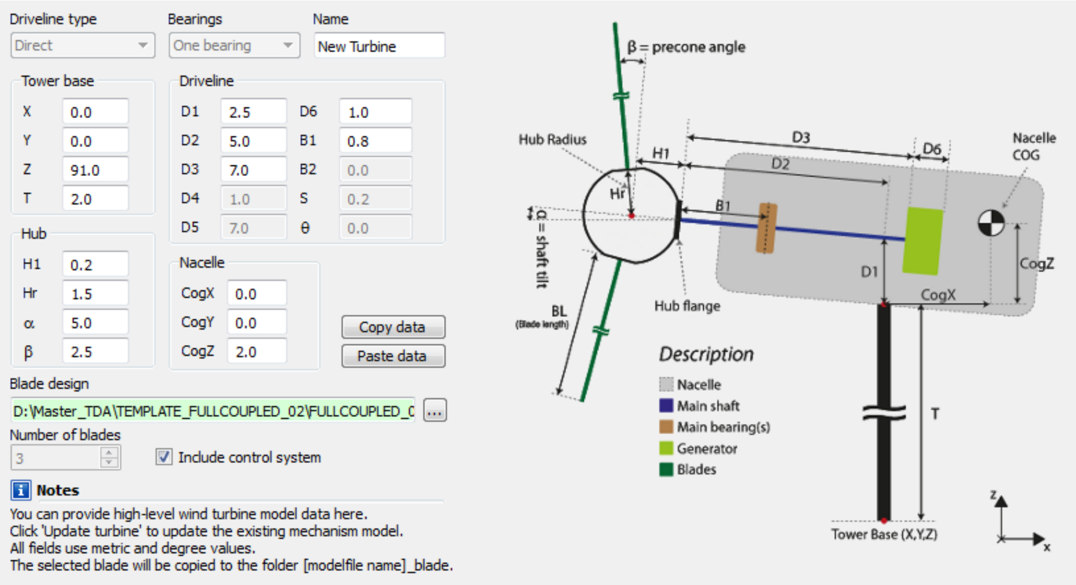


Figure 17 - FEDEM fully coupled analysis model

Nacelle definition including turbine blades and generator:

A standard nacelle was chosen, and attached to the top of the tower. The only parameters changed in the nacelle setup were the starting point of the tower base, and the Z-position of the nacelle. The driveline type was set to direct drive, with one bearing. Blade definitions and control system was left as default parameters for 5MW offshore wind turbine.



Driveline type: Direct
Bearings: One bearing
Name: New Turbine

Tower base

X	0.0
Y	0.0
Z	91.0
T	2.0

Driveline

D1	2.5	D6	1.0
D2	5.0	B1	0.8
D3	7.0	B2	0.0
D4	1.0	S	0.2
D5	7.0	θ	0.0

Hub

H1	0.2
Hr	1.5
α	5.0
β	2.5

Nacelle

CogX	0.0
CogY	0.0
CogZ	2.0

Blade design: D:\Master_TDA\TEMPLATE_FULLCOUPLED_02\FULLCOUPLED_C
Number of blades: 3
☑ Include control system

Notes

You can provide high-level wind turbine model data here.
Click 'Update turbine' to update the existing mechanism model.
All fields use metric and degree values.
The selected blade will be copied to the folder [modelfile name]_blade.

Figure 18 - FEDEM Windpower Turbine definition

Connecting the different parts:

In order to be able to perform an analysis, the parts contained within the analysis model had to be joined together. The Jacket was connected to the Tower using *Rigid Joints* (all DOF`s fixed), and the nacelle was connected to the tower in the same way.

To secure the jacket to the seabed, springs were used. FEDEM Windpower has the ability to import soil springs, but this feature was not tried. Instead the piles used in the USFOS analysis model was copied by extracting the full (6x6) spring properties, and the corresponding mass and damping properties from the USFOS piles. The extracted data was put into a *Free Joint* which in turn was attached to the end of each jacket leg. Spring properties and damping properties was added to the *Free Joint* definitions, while the mass properties were added to the node onto which the *Free Joint* was attached. The full spring was defined using the command *#GlobalSpring #Kij* for specifying off diagonal stiffness properties. The mass property was defined as a single value for all translational degrees of freedom, and three different values for rotational mass. The Damping coefficients were specified one per DOF.

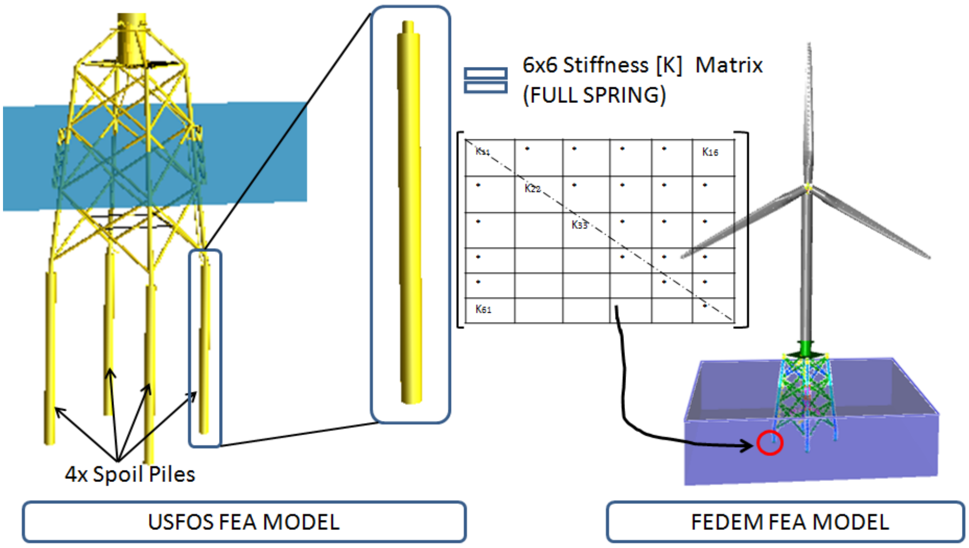


Figure 19 - FEDEM 6x6 spring setup

Node masses:

A list of node masses was received from TDA, and they were all added to the FEDEM model.

Wire Pre-tension:

In order to stabilize the lower bracer junction, the designers of the jacket structure have chosen to connect all four lower junctions with a pretension wire. The pretension needs to be reproduced in FEDEM. This is achieved in FEDEM by adding an axial spring with the specified pretension force at 225 kN.

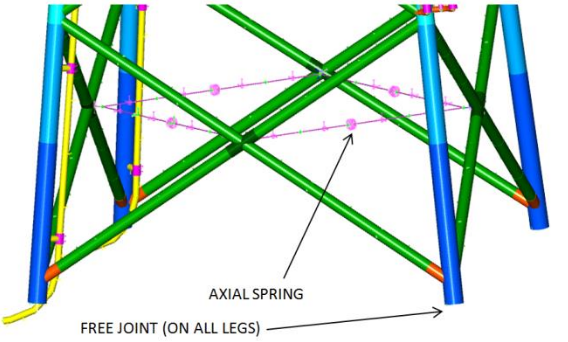


Figure 20 - FEDEM wire and pile definition

Analysis procedure:

The analysis in FEDEM is set up as a dynamic analysis including an initial equilibrium at the beginning. In the initial equilibrium step, all static loads (mass, eigenweight, pretension forces etc.) are introduced onto the structure, and equilibrium is achieved. This is the starting condition for the dynamic analysis. If an initial equilibrium is not performed prior to starting the dynamic analysis, the chance of sudden onset of vibrations and convergence problems is present.

Results Output:

The results generated from this analysis are used for:

1) Study of out of plane vibrations in the centre of upper-, middle- and lower bracer junction. Displacement of nodes is extracted from the analysis results (T).
2) Study of fatigue damage parameter (Palmgren-Miner) on joints between bracers and jacket legs. Beam end axial forces and moments extracted from analysis results (F).
3) Study of displacement in tower attachment node.

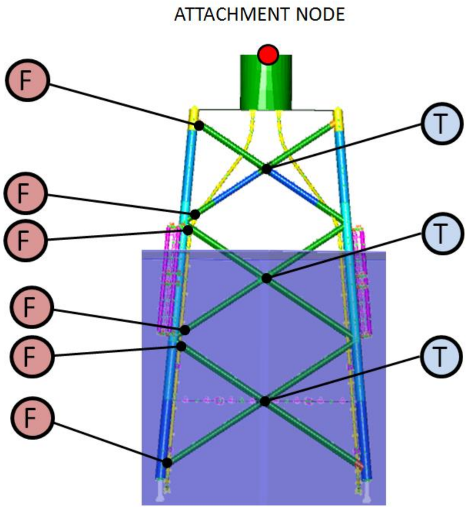


Figure 21 - FEDEM result extraction

5.3.2 USFOS analysis model, used in sequentially coupled analysis

Input geometry:

1) Jacket: ORMO_JACKET_NAS_10.nas.
2) Midsection: ORMO_MIDSECTION_NAS_6.nas
3) Soil Piles: Nodes, elements, cross sectional-, mass- and spring properties received in text file.

Model conversion

The structural model in USFOS consists of three separate models manually merged into one.

The jacket and the midsection model had to be converted from Nastran to USFOS text file input format before being able to run analysis in USFOS. The Soil piles were received as nodal and elemental information together with spring properties in a text file.

The work of conversion was time consuming, and several issues had to be overcome in order to have a working FEA model. One issue worth mentioning is that USFOS has a criterion for the ratio between element length and diameter. Such short elements existed several places in the element model and the challenge was to rearrange the nodes without changing the geometry.

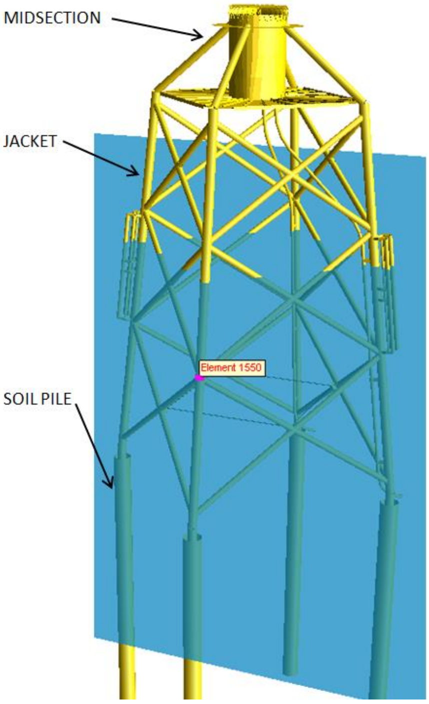
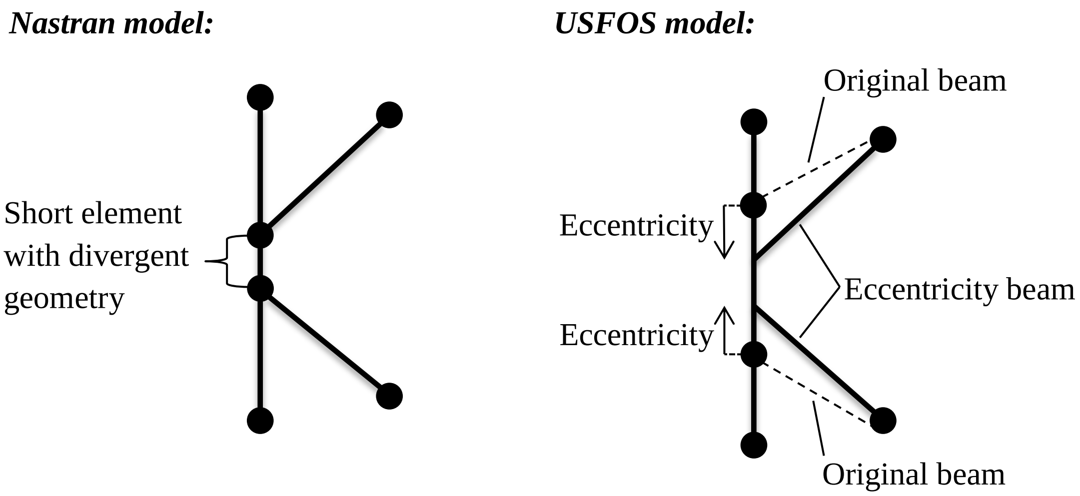


Figure 22 - USFOS model

K-Joints

The elements of the legs in the K-Joints (highlighted on *Figure 22 - USFOS model*) were a challenge regarding too short element. The bracer legs don’t share the same node in the joint. There is a small gap of 170 mm. This short element also has a larger wall thickness than the surrounding elements. To be able to keep the geometry and element properties in a satisfactory manner, eccentricities for the beams was applied.

In practical terms, the method involves to move the nodes connecting the bracer and the legs apart from each other until the length-diameter ratio for the element is fulfilled. In order to maintain the original geometry, the bracer element has to be specified as an eccentricity in the end of the element closest to the leg. The method is illustrated visually as *Figure 23 - K-Joint eccentricity in USFOS* and as USFOS input in *Table 5 - Element end eccentricities*.


Nastran model:

USFOS model:

Figure 23 - K-Joint eccentricity in USFOS

Table 5 - Element end eccentricities

ECCENT	ECC ID	ex	ey	ez
	1149	-5.29E-7	-1.59E-2	1.30E-1
	1151	5.29E-7	1.59E-2	-1.30E-1

Adding parts (midsection and piles)

Merging parts manually requires the modeller to create common nodes at interface areas between parts that originally were separate. This means replacing nodes from an element with the interface node of the adjoining element.

Transferring loads (RBE2) from tower to foundation structure

The jacket foundation interface node is in principle rigidly connected to the midsection quad shell elements. In the NAS model file this is solved by a RBE2 element connecting the interface node to all nodes to the shell element along the tower wall. An RBE2 element (Rigid Body Element, Form 2) is an element introducing equal displacement between nodes. In this way the relative displacement between the nodes of the element is zero. USFOS do not have any integrated solution for adding RBE2 elements to the model. This rigid connection had to be addressed in another way.

For more detailed information for RBE2 element, see “Rigid Elements and Multipoint Constraints” page 167 reference (MSC Nastran Reference Manual).

Some different methods were tried in order to imitate the properties of an RBE2 element. In USFOS it is possible to enforce an eccentricity at a node for an element. Generally the code will treat the eccentricity as a rigid coupling. The challenge for this method is the restriction of use embedded in the code. The element length ratio to the eccentricity length was too high, and the code refused to run.

Another method is to replace the RBE2 element with an infinitely stiff spring element. In USFOS this is in practice done by defining a beam element representing the spring with spring properties. The spring works perfectly with longitudinal, lateral and vertical movement. The problem was that the spring transferred rotation directly from one end to the other without any coupling to xyz movement. In practice this means that rotation in the interface node was not transferred to the surrounding nodes by longitudinal, lateral and vertical movement, see *Figure 24 – Spring element as RBE2 imitation*.

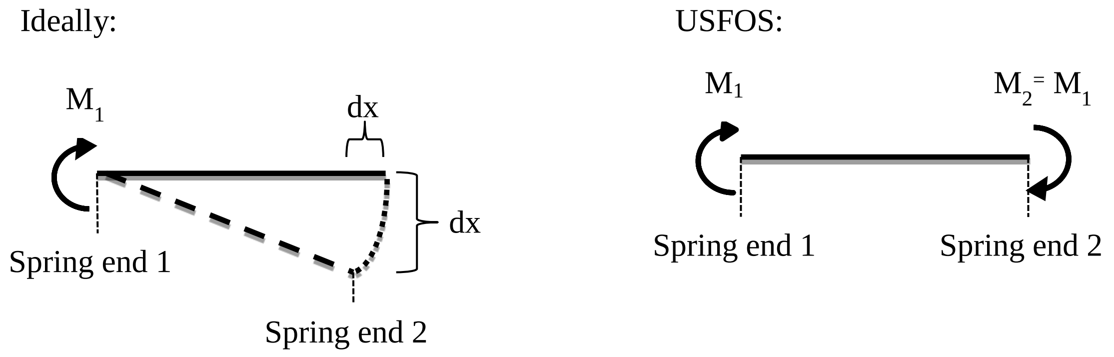


Figure 24 – Spring element as RBE2 imitation

USFOS has a utility defining a master-slave relationship. The method is called “BLINDP2” and includes the possibility to define a master element and a slave node, se *Table 6 - Master element/Slave node - USFOS*. The drawback is by this method is the master element naturally has to be a beam element. It is possible to implement this solution, but only by including several other utilities. This will include unnecessary many uncertainties to the model.

Table 6 - Master element/Slave node - USFOS

BLINDP2	Slave Node	Master Elem	Ix	iy	iz	irx	iry	irz
	4001	4101	1	1	1	1	1	1

The method that got implemented was to imitate the RBE2 elements with ordinary beam elements. The geometry of the pipe was set to the maximal tolerated length-diameter ratio; see *Table 7 - Stiff beam - Geometry*.

Table 7 - Stiff beam - Geometry

PIPE	Geom ID	Do	Thick	Shear_y	Shear_z
	35	1.25	0.6		

As for the geometry, the material properties were set to maximal values; see *Table 8 - Stiff beam - Material model*. This method is not absolutely correct mathematically, but should give approximately the same properties as using a RBE2 element.

Table 8 - Stiff beam - Material model

MISOIEP	Matno	E-Mod	Poiss	Yield	Density	Therm. Exp.	c1	a1
	5	1E+99	0.3	1E+99	0	0		

Node masses

Nodal masses were defined in the same way as for the FEDEM model.

Wire Pre-Tension

As with the FEDEM model, wire pretension had to be added. The method used to generate a pretension force in USFOS was slightly different than in FEDEM. Instead of using an axial spring, a very low temperature (-600 degrees) was imposed on the wires in question using the command BELTEMP (*Table 9 - Wire pre-tension - Beltemp*). This results in shrinking of the wires and thereby giving the desired pretension force (225 kN). The out of plane movement of the wire attachment point to the bracer junction is approximately 50mm.

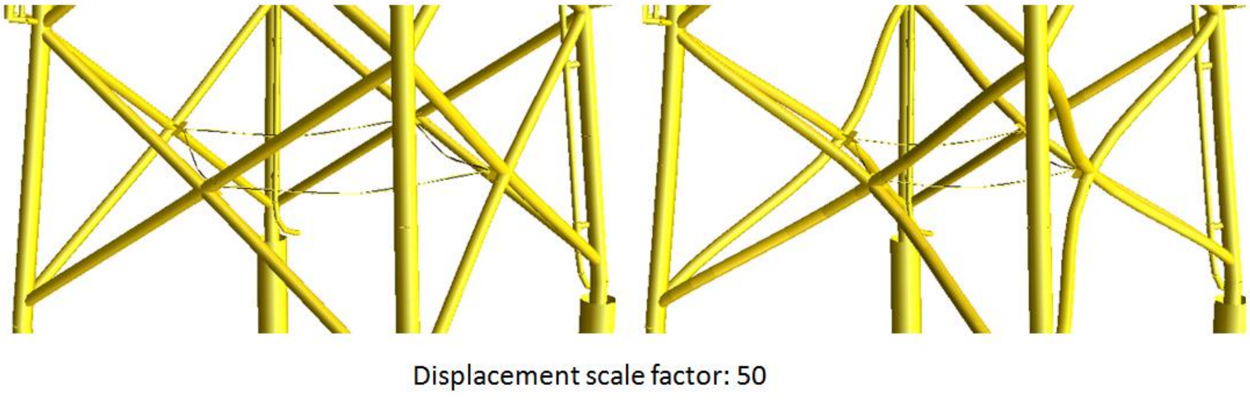


Figure 25 - Wire pre-tension

Figure 26 - Wire pre-tension - Beltemp

Table 9 - Wire pre-tension - Beltemp

BELTEMP	L_Case	ElnoX	T0	Ty	Tz
	1	615	-600	1	1

Soil piles

Originally the soil piles were defined by two set of nodes. The first set was for defining the element sections. The second was to define the linear soil springs, and was fixed in all directions. The springs were added from the fixed nodes to the pile element nodes by a zero-length spring. In USFOS the spring end is attached to the pile, while the other is automatically attached to ground. The spring properties are defined by "material" properties.

Table 10 - Soil Springs - USFOS

SPRINGS2GR	Elem ID	Np1	Material	L_coor	Ecc1
	10001	6	35		

5.3.3 FEDEM Sequential analysis model

The FEDEM analysis model used for sequentially coupled analysis is equal to the fully coupled analysis model from the intersection node and upwards. The tower, nacelle, blade and turbine setup is completely the same for both analysis methods. This is also true for the turbulent wind file.

While the tower and the windturbine are equal to the fully coupled analysis model, the lower part of the structure is governed by the properties extracted from the jacket by the **K**, **C** and **M** matrices. This is done by adding the extracted values from the matrices to a *Free Joint* in FEDEM. The *Free Joint* allows the user to specify the desired property in the desired degree of freedom. The full 6x6 stiffness matrix (**K**) and the diagonal damping matrix (**C**) are defined here. The diagonal mass matrix (**M**) is defined as one mass value and three inertial mass values at the interface node where the *Free Joint* is attached to the tower. The lower part of the *Free Joint* is attached to ground and all DOF are constrained.

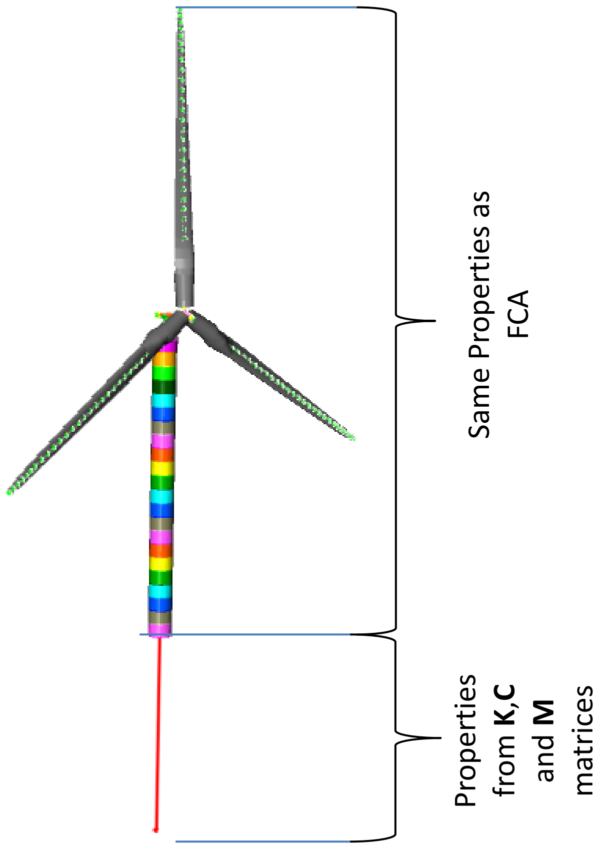


Figure 27 - FEDEM sequentially coupled analysis model

5.4 Running USFOS analyses

Running of USFOS is based on being run through CMD (Command Prompt, MS command-line interpreter). In this way there is no automatics related to analysis series. When running USFOS right out of the box it is necessary to enter the input data manually for each run. To make running analysis efficient, a set of analysis batch scripts were written. All scripts are attached in chapter 11.26.

5.4.1 Cleaning batch system

The "_00_00_CleanBat" batch file is attached as chapter 11.26.1. It deletes all batch files in the folder, with all analysis text help files (input files: *UAR.txt, ref chapter 5.4.3).

5.4.2 Software running

There are three software programs in the USFOS family used in this project.

USFOS:

- Run batch file: "_01_01_RunUsfos", chapter 11.26.2.
- USFOS is the structure analysis software

Dynres

- Run batch file: "_01_01_RunDynres", chapter 11.26.3.
- Dynres is the software for extracting dynamic data during the analysis. It is limited to displacement.

Postfos

- Run batch file: "_01_01_RunPostfos", chapter 11.26.4.
- Postfos is the software for extracting results from "*.raf" analysis files after analyses are run. It is strictly limited, and may not be connected to time history.

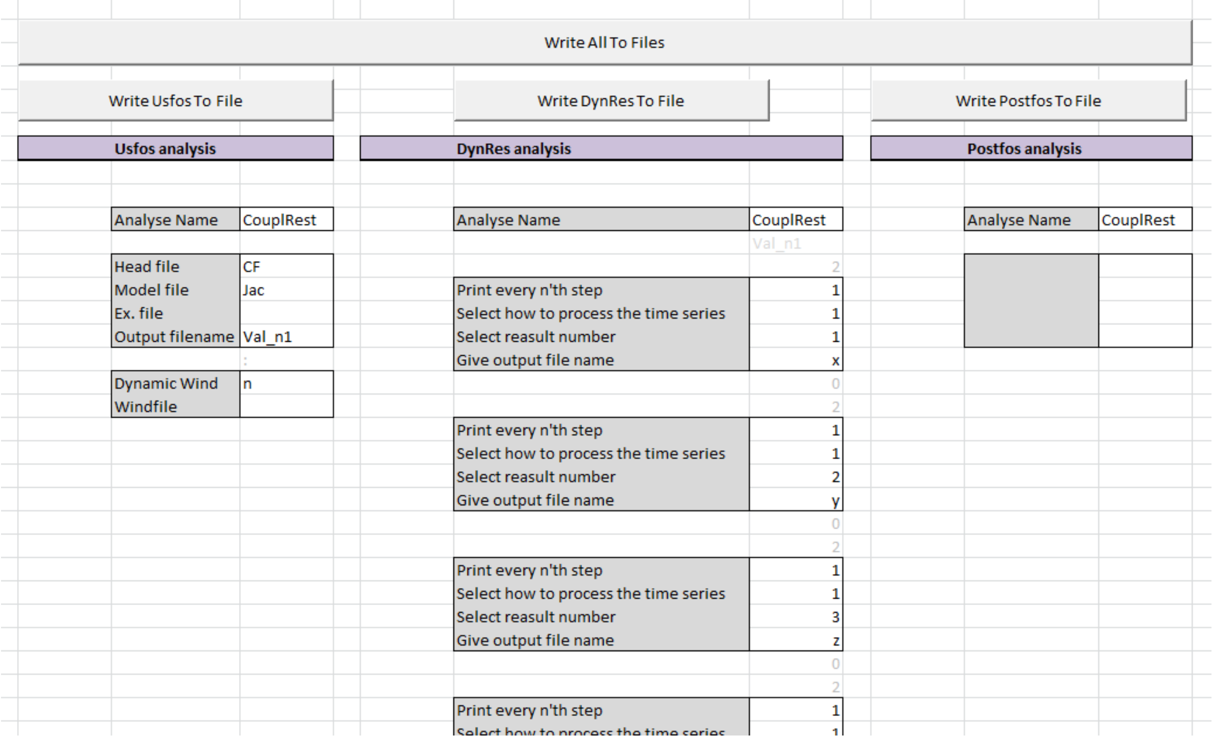
Xact

- Run batch file: "_01_01_RunXact", chapter 11.26.5.
- Xact is postprocessor software with GUI. It may extract all data after the analysis is ran, but is therefore restricted to data written to the "*.raf" file.

5.4.3 Analysis run

The software is run in a command window. When the command window is set to correct work path, the analysis software is executed. The specific analysis input parameters are then written and the analysis executed. To have a clean "run" system, an excel workbook that writes analysis specific "run files" were made. The input setup sheet is illustrated as *Figure 28 - Excel sheet for generation of analysis run files*.

The sheet is made of three parts. The first block is the input data for the USFOS analysis, the second block is for the Dynres result extraction, and the third block is for the Postfos result extraction. The Dynres and Postfos block is unlimited in length. The blocks may be used one by one, or all three together. When the macros "Write *** To File" is executed, the analysis files is written to workfolder.



Usfos analysis

Analyse Name	CouplRest
Head file	CF
Model file	Jac
Ex. file	
Output filename	Val_n1
Dynamic Wind	n
Windfile	

DynRes analysis

Analyse Name	CouplRest
	Val_n1
	2
Print every n'th step	1
Select how to process the time series	1
Select reasult number	1
Give output file name	x
	0
	2
Print every n'th step	1
Select how to process the time series	1
Select reasult number	2
Give output file name	y
	0
	2
Print every n'th step	1
Select how to process the time series	1
Select reasult number	3
Give output file name	z
	0
	2
Print every n'th step	1
Select how to process the time series	1

Postfos analysis

Analyse Name	CouplRest

Figure 28 - Excel sheet for generation of analysis run files

The analysis package consists of two files per block, and is written by the excel sheet in *Figure 28 - Excel sheet for generation of analysis run files*. The *Figure 29 - Analysis run batch package* illustrates the USFOS analysis file system. The script "*UA.bat" is the batch is the file executed to run the analysis. When the workfolder is set, and the software is run, it retrieves the file "*UAR.txt". This file contains the analysis specific input data. In this way the analyses may be executed without manual input in the command window.

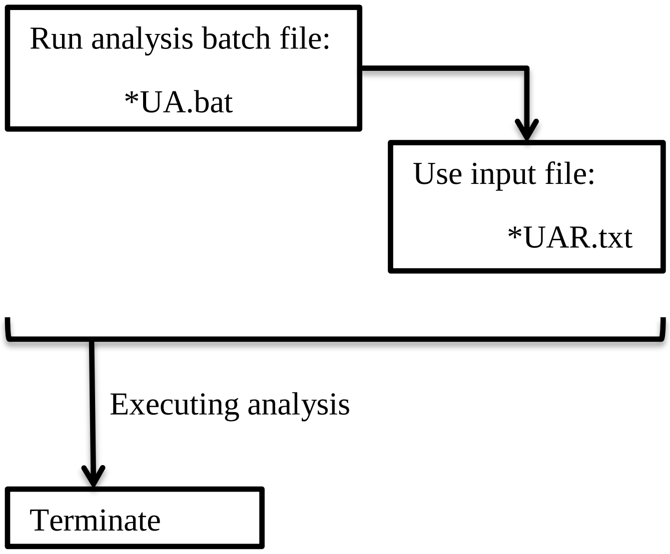


Figure 29 - Analysis run batch package

This system enables the analysis to be run in sequence automatically. When all analysis setups are written the batch file "_02_01_RunUsfosAnalyses.bat" may be run to execute all analyses in sequence. It is illustrated as *Figure 30 - Running of USFOS analysis sequence*. This script is attached in chapter 11.26.7.

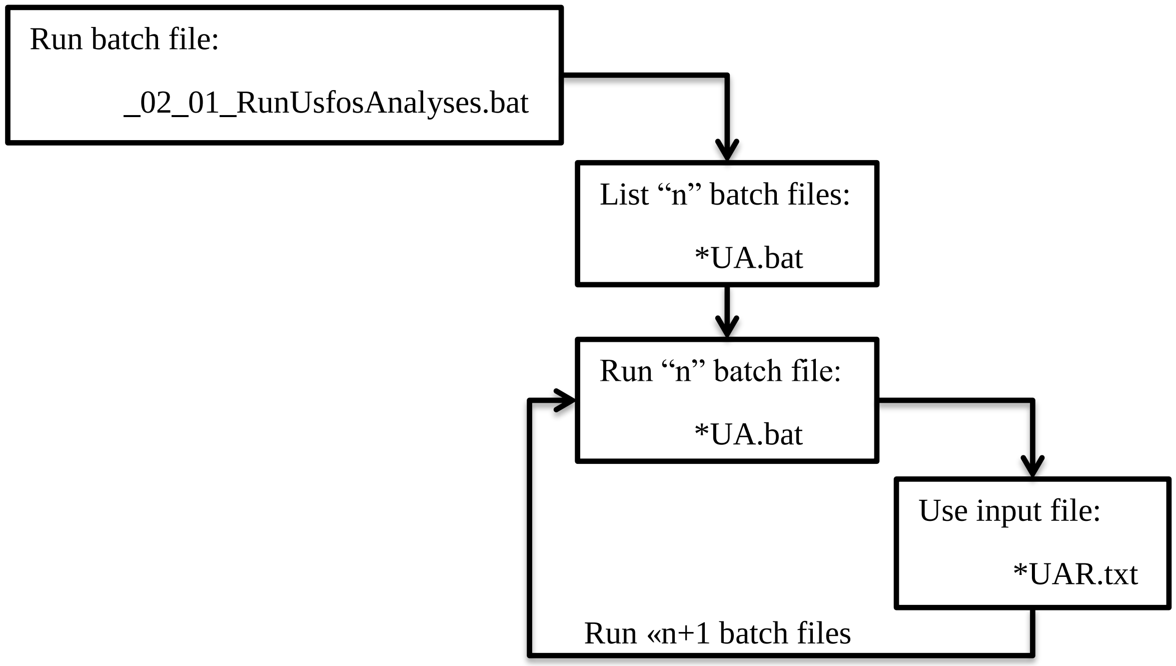


Figure 30 - Running of USFOS analysis sequence

The “Run” scripts consist of four batch files:

“_02_00_RunAllAnalyses.bat”:

- This batch file runs all analysis scripts (01-03) in sequence
- The script is attached in chapter 11.26.6.

“_02_01_RunUsfosAnalyses.bat”:

- This batch file runs the USFOS analyses in the workfolder in sequence
- The script is attached in chapter 11.26.7.

“_02_02_RunDynresAnalyses.bat”:

- This batch file runs the Dynres analyses in the workfolder in sequence
- The script is attached in chapter 11.26.8.

“_02_03_RunPostfosAnalyses.bat”:

- This batch file runs the Postfos analyses in the workfolder in sequence
- The script is attached in chapter 11.26.9.

5.4.4 Cleaning of analysis results

Due to large analysis result files, the files have to be deleted after the specific results of interest are extracted. A set of scripts are written for this purpose:

“_03_00_CleanAllResults.bat”:

- This batch file runs all clean scripts (01-03) in sequence
- The script is attached in chapter 11.26.10.

“_03_01_CleanUsfosResults.bat”:

- This batch file cleans all the USFOS analyses results in the workfolder in sequence
- The script is attached in chapter 11.26.11.

“_03_02_CleanDynresResults.bat”:

- This batch file cleans all the Dynres analyses results in the workfolder in sequence
- The script is attached in chapter 11.26.12.

“_03_03_CleanPostfosResults.bat”:

- This batch file cleans all the Postfos analyses results in the workfolder in sequence
- The script is attached in chapter 11.26.13.

5.5 Sequentially coupled method USFOS/FEDEM

The sequentially coupled method used to integrate USFOS and FEDEM is illustrated by *Figure 31 - Sequentially coupled analysis method*.

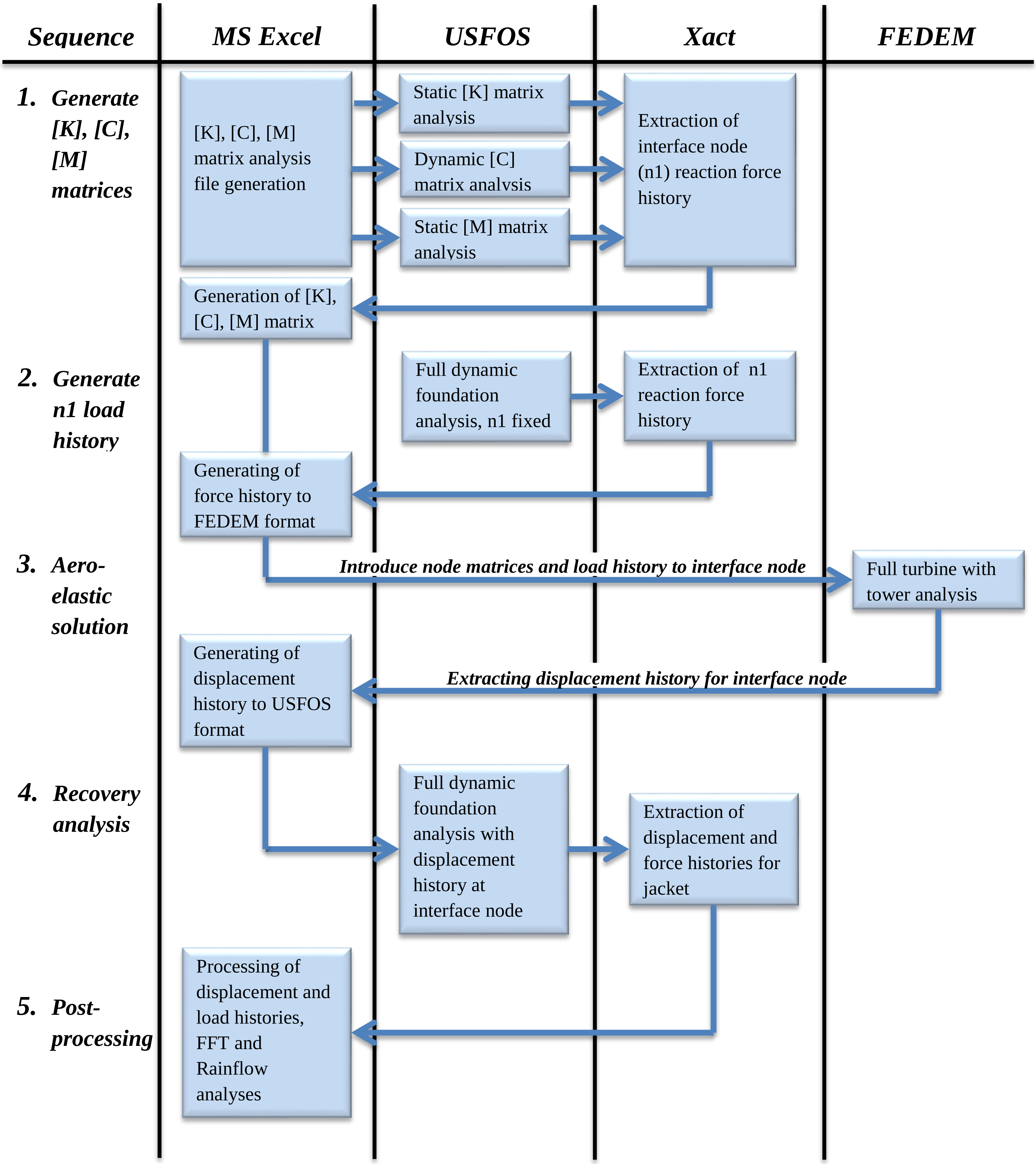


Figure 31 - Sequentially coupled analysis method

Neither USFOS nor FEDEM was designed to run the sequentially coupled analysis method. To be able to compare the fully coupled analysis with sequentially coupled analysis it was necessary to manually put together a sequentially coupled analysis method that links together the selected programs. The original sequentially coupled method is described in detail in the literature study chapter 4.1.

Sequence 1 – Generate [K], [C], [M] matrices

The major challenge to overcome was that neither USFOS nor FEDEM had the ability to calculate the mass, damping and stiffness matrices from the interface node. Ideally the matrices should be extracted automatically from first increment of the dynamic analysis meant to find the interface node reaction force.

To be able to calculate the matrices from the jacket foundation in USFOS, a series of static and dynamic analyses was run. A matrix extraction tool that combines the use of MS Excel, USFOS and Xact was written. The tool is described in detail in chapter 5.6.4. The schematic of the matrices generation is illustrated as sequence 1 in *Figure 31 - Sequentially coupled analysis method*. The calculations and evaluations of the matrices are covered in chapter 5.6.

Sequence 2 – Generate n1 load history

The generation of the load force history for the interface node is described as sequence 2. This sequence method is similar to the ordinary sequentially coupled analysis (chapter 4.1). The node is fixed in the interface node while the jacket foundation undergoes a full-time load history. The reaction force history in the interface node is recorded. MS Excel imports the data and converts it from USFOS format to FEDEM format.

Sequence 3 – Aero elastic solution

FEDEM Windpower was used for the aero elastic solution. The full load history was applied the interface node together with the mass, damping and stiffness matrices. FEDEM only had the possibility to integrate the full symmetric 6x6 stiffness matrix. For the damping matrix it was only possible to apply the 6x6 diagonal matrices, while for the mass matrix only four of the diagonals could be defined, one mass component and three rotational inertias. When the analysis was run the displacement history of the interface node was recorded. The time history was imported by MS Excel and converted from FEDEM format to USFOS format.

Sequence 4 – Recovery analysis

The jacket foundation stress recovery analysis was run in USFOS. The displacement time history from the aero elastic solution was applied the interface node. Displacement and force histories were then extracted by Xact from a number of control points for the jacket.

Sequence 5 – Postprocessing

The result was imported by MS Excel for comparison. To be able to run FFT and Rainflow the results was exported to Matlab for processing.

5.6 Extraction of K-C-M Matrices

Step 1 in the sequentially coupled analysis is to extract the stifness [K], damping [C] and mass [M] matrices.

5.6.1 Stiffness matrix [K]

To find the stiffness matrix [K] the Hooke's law is derived as equation 5.6.1.1.

$kx = F$ eq. (5.6.1.1)

In matrix form the constant factor "k" is the 6x6 matrix [K], the distance "x" is the 6x1 matrix [r], and the force "F" is the 6x1 reaction force matrix [R], equation 5.6.1.2.

$[K][r] = [R]$ eq. (5.6.1.2)

The annotations used for the matrix is represented as equation 5.6.1.3

$$\begin{bmatrix} K_{11} & K_{12} & K_{13} & K_{14} & K_{15} & K_{16} \\ K_{21} & K_{22} & K_{23} & K_{24} & K_{25} & K_{26} \\ K_{31} & K_{32} & K_{33} & K_{34} & K_{35} & K_{36} \\ K_{41} & K_{42} & K_{43} & K_{44} & K_{45} & K_{46} \\ K_{51} & K_{52} & K_{53} & K_{54} & K_{55} & K_{56} \\ K_{61} & K_{62} & K_{63} & K_{64} & K_{65} & K_{66} \end{bmatrix} \begin{bmatrix} r_1 \\ r_2 \\ r_3 \\ r_4 \\ r_5 \\ r_6 \end{bmatrix} = \begin{bmatrix} R_1 \\ R_2 \\ R_3 \\ R_4 \\ R_5 \\ R_6 \end{bmatrix} \quad \text{eq. (5.6.1.3)}$$

The matrices are abbreviated as equation 5.6.1.4

$$\begin{bmatrix} K_{11} & \cdots & K_{16} \\ \vdots & \ddots & \vdots \\ K_{61} & \cdots & K_{66} \end{bmatrix} \begin{bmatrix} r_1 \\ \vdots \\ r_6 \end{bmatrix} = \begin{bmatrix} R_1 \\ \vdots \\ R_6 \end{bmatrix} \quad \text{eq. (5.6.1.4)}$$

To extract the stiffness matrix, the analysis software "USFOS" is used. The analysis sequence is represented as *Figure 32 - Analysis for [K] extraction.*

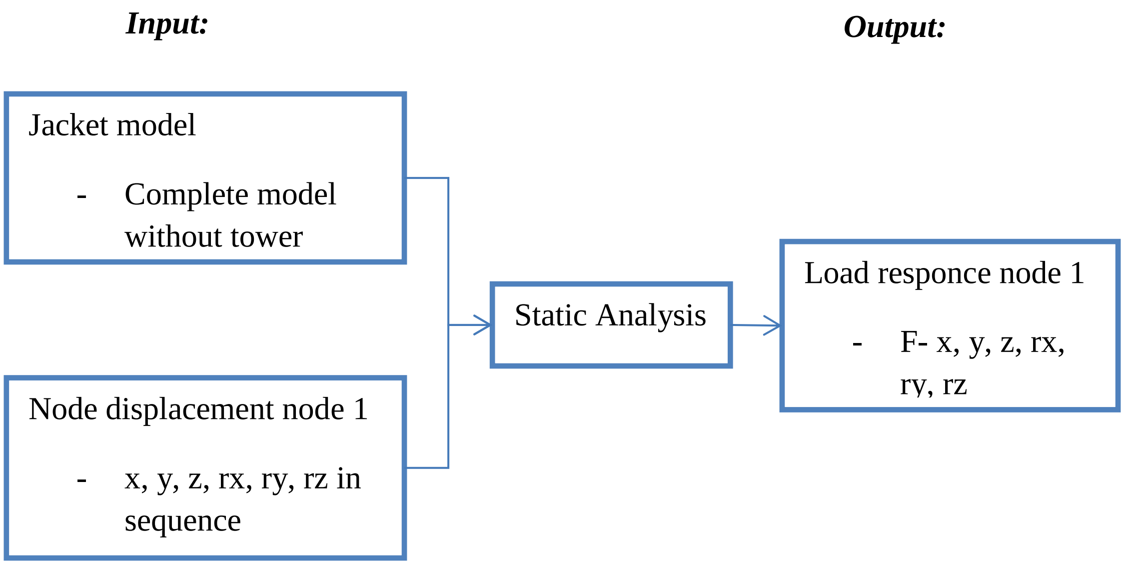


Figure 32 - Analysis for [K] extraction

A static analysis is written for this purpose. The model of the jacket structure includes the piles with linear springs, the jacket structure, and the midsection. In total this sequence consists of 6 analyses, illustrated in *Table 11 - Analysis sequence for matrix [K]*:

Table 11 - Analysis sequence for matrix [K]

Analysis 1		**Analysis 2**		**Analysis 3**		**Analysis 4**		**Analysis 5**		**Analysis 6**	
Inp	***Outp***	***Inp***	***Outp***	***Inp***	***Outp***	***Inp***	***Outp***	***Inp***	***Outp***	***Inp***	***Outp***
dx	Fx	0	Fx	0	Fx	0	Fx	0	Fx	0	Fx
0	Fy	dy	Fy	0	Fy	0	Fy	0	Fy	0	Fy
0	Fz	0	Fz	dz	Fz	0	Fz	0	Fz	0	Fz
0	Mx	0	Mx	0	Mx	drx	Mx	0	Mx	0	Mx
0	My	0	My	0	My	0	My	dry	My	0	My
0	Mz	0	Mz	0	Mz	0	Mz	0	Mz	drz	Mz

Displacement for node 1 in 6 degrees of freedom are applied, and reaction forces are printed for the matrix.

The analysis consists of two static steps. In the first step the wire pretension is activated. This introduces forces in the interface node that has to be subtracted from the equilibrium calculations. To be able to identify these forces, the pretension is conducted under a separate step. The second static step is the displacement of the interface node.

Table 12 - Analysis displacement for matrix [K]

Analysis	**1**		**2**		**3**		**4**		**5**		**6**	
Step	***1***	***2***	***1***	***2***	***1***	***2***	***1***	***2***	***1***	***2***	***1***	***2***
r1 [m]	0	0.001	0	0	0	0	0	0	0	0	0	0
r2 [m]	0	0	0	0.001	0	0	0	0	0	0	0	0
r3 [m]	0	0	0	0	0	0.001	0	0	0	0	0	0
r4 [m]	0	0	0	0	0	0	0	0.001	0	0	0	0
r5 [m]	0	0	0	0	0	0	0	0	0	0.001	0	0
r6 [m]	0	0	0	0	0	0	0	0	0	0	0	0.001

This displacement history gives the following reaction forces in step 1 and 2:

Table 13 - Analysis reaction force for matrix [K]

Analysis	**1**		**2**		**3**		**4**		**5**		**6**	
Step	***1***	***2***	***1***	***2***	***1***	***2***	***1***	***2***	***1***	***2***	***1***	***2***
R1 [N]	-727	59 747	-727	-727	-727	-1 300	-727	-363	-728	-396 291	-727	-751
R2 [N]	-771	-772	-771	59 632	-771	-1 356	-771	396 230	-771	-430	-772	-743
R3 [N]	-142 782	-143 341	-142 782	-143 353	-142 786	739 439	-143 111	-137 134	-143 111	-145 107	-143 001	-143 198
R4 [N]	-2 417	-2 053	-2 417	394 586	-2 417	1 570	-2 386	35 620 711	-2 419	-2 357	-2 418	-2 010
R5 [N]	2 449	-393 118	2 449	2 790	2 449	-1 534	2 451	2 487	2 517	35 656 088	2 450	2 768
R6 [N]	-18 660	-18 684	-18 660	-18 632	-18 660	-20 188	-18 661	-18 254	-18 661	-18 336	-18 655	8 504 263

Calculations for analysis 1 are derived as following:

$$\left[R^{(s2-s1)}\right] = \left[R^{(s2)}\right] - \left[R^{(s1)}\right] = \begin{bmatrix} R_1{}^{(s2)} \\ \vdots \\ R_6{}^{(s2)} \end{bmatrix} - \begin{bmatrix} R_1{}^{(s1)} \\ \vdots \\ R_6{}^{(s1)} \end{bmatrix} \qquad \text{eq. (5.6.1.4)}$$

$$\left[R^{(s2-s1)}\right] = \begin{bmatrix} 59747 \\ -772 \\ -143341 \\ -2053 \\ -393118 \\ -18684 \end{bmatrix} - \begin{bmatrix} -727 \\ -771 \\ -142782 \\ -2417 \\ 2449 \\ -18660 \end{bmatrix} = \begin{bmatrix} 60474 \\ 0 \\ -560 \\ 364 \\ -395567 \\ -24 \end{bmatrix} \qquad \text{eq. (5.6.1.5)}$$

The results is derived at r=0.001 m. To scale it linearly for r=1, the matrix $[R^{(s2-s1)}]$ is divided by the displacement 0.001:

$$[R] = \begin{bmatrix} 60474 \\ 0 \\ -560 \\ 364 \\ -395567 \\ -24 \end{bmatrix} /0.001 = \begin{bmatrix} 60474128 \\ -152 \\ -559670 \\ 363779 \\ -395567433 \\ -23941 \end{bmatrix} \quad \text{eq. (5.6.1.6)}$$

When r=1 we got following relationship:

$$\begin{bmatrix} K_{11} \\ K_{21} \\ K_{31} \\ K_{41} \\ K_{51} \\ K_{61} \end{bmatrix} \begin{bmatrix} 1 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} = \begin{bmatrix} R_1 \\ R_2 \\ R_3 \\ R_4 \\ R_5 \\ R_6 \end{bmatrix} \quad \text{eq. (5.6.1.7)}$$

Where:

$$\begin{bmatrix} K_{11} \\ K_{21} \\ K_{31} \\ K_{41} \\ K_{51} \\ K_{61} \end{bmatrix} = \begin{bmatrix} R_1 \\ R_2 \\ R_3 \\ R_4 \\ R_5 \\ R_6 \end{bmatrix} = \begin{bmatrix} 60474128 \\ -152 \\ -559670 \\ 363779 \\ -395567433 \\ -23941 \end{bmatrix} \quad \text{eq. (5.6.1.8)}$$

This calculation is repeated for all the analyses, and the complete stiffness matrix for the interface node is:

$$[K] = \begin{bmatrix} K_{11} & K_{12} & K_{13} & K_{14} & K_{15} & K_{16} \\ K_{21} & K_{22} & K_{23} & K_{24} & K_{25} & K_{26} \\ K_{31} & K_{32} & K_{33} & K_{34} & K_{35} & K_{36} \\ K_{41} & K_{42} & K_{43} & K_{44} & K_{45} & K_{46} \\ K_{51} & K_{52} & K_{53} & K_{54} & K_{55} & K_{56} \\ K_{61} & K_{62} & K_{63} & K_{64} & K_{65} & K_{66} \end{bmatrix} \quad \text{eq. (5.6.1.9)}$$

$$= \begin{bmatrix} 60474128 & -152 & -566448 & 363683 & -395565382 & -23872 \\ -152 & 60403804 & -578250 & 397001711 & 341186 & 28088 \\ -566448 & -578250 & 882224996 & 4981832 & -2989319 & -862191 \\ 363683 & 397001711 & 4981832 & 35623097337 & 48891 & 407625 \\ -395565382 & 341186 & -2989319 & 48891 & 35653570937 & 321434 \\ -23872 & 28088 & -862191 & 407625 & 321434 & 8522918451 \end{bmatrix}$$

5.6.2 Damping matrix [C]

The damping matrix [C] is a dynamic analysis factor. To be able to find this parameter it was necessary to use the dynamic equation of motion:

$$[M][\ddot{r}] + [C][\dot{r}] + [K][r] = [R] \qquad \text{eq. (5.6.2.1)}$$

Where

$$[M] = \begin{bmatrix} M_{11} & M_{12} & M_{13} & M_{14} & M_{15} & M_{16} \\ M_{21} & M_{22} & M_{23} & M_{24} & M_{25} & M_{26} \\ M_{31} & M_{32} & M_{33} & M_{34} & M_{35} & M_{36} \\ M_{41} & M_{42} & M_{43} & M_{44} & M_{45} & M_{46} \\ M_{51} & M_{52} & M_{53} & M_{54} & M_{55} & M_{56} \\ M_{61} & M_{62} & M_{63} & M_{64} & M_{65} & M_{66} \end{bmatrix} \qquad \text{eq. (5.6.2.2)}$$

$$[C] = \begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ C_{21} & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ C_{31} & C_{32} & C_{33} & C_{34} & C_{35} & C_{36} \\ C_{41} & C_{42} & C_{43} & C_{44} & C_{45} & C_{46} \\ C_{51} & C_{52} & C_{53} & C_{54} & C_{55} & C_{56} \\ C_{61} & C_{62} & C_{63} & C_{64} & C_{65} & C_{66} \end{bmatrix} \qquad \text{eq. (5.6.2.3)}$$

That gives:

$$\begin{bmatrix} M_{11} & \cdots & M_{16} \\ \vdots & \ddots & \vdots \\ M_{61} & \cdots & M_{66} \end{bmatrix} \begin{bmatrix} \ddot{r}_1 \\ \vdots \\ \ddot{r}_6 \end{bmatrix} + \begin{bmatrix} C_{11} & \cdots & C_{16} \\ \vdots & \ddots & \vdots \\ C_{61} & \cdots & C_{66} \end{bmatrix} \begin{bmatrix} \dot{r}_1 \\ \vdots \\ \dot{r}_6 \end{bmatrix} + \begin{bmatrix} K_{11} & \cdots & K_{16} \\ \vdots & \ddots & \vdots \\ K_{61} & \cdots & K_{66} \end{bmatrix} \begin{bmatrix} r_1 \\ \vdots \\ r_6 \end{bmatrix} = \begin{bmatrix} R_1 \\ \vdots \\ R_6 \end{bmatrix} \qquad \text{eq. (5.6.2.4)}$$

A method for finding the damping matrix [C] is to force a constant velocity [¨r] in the interface node. When the velocity is constant, the acceleration is equal to 0. The mass matrix [M] may then be removed from the equation:

$$\begin{bmatrix} M_{11} & \cdots & M_{16} \\ \vdots & \ddots & \vdots \\ M_{61} & \cdots & M_{66} \end{bmatrix} [0] + \begin{bmatrix} C_{11} & \cdots & C_{16} \\ \vdots & \ddots & \vdots \\ C_{61} & \cdots & C_{66} \end{bmatrix} \begin{bmatrix} \dot{r}_1 \\ \vdots \\ \dot{r}_6 \end{bmatrix} + \begin{bmatrix} K_{11} & \cdots & K_{16} \\ \vdots & \ddots & \vdots \\ K_{61} & \cdots & K_{66} \end{bmatrix} \begin{bmatrix} r_1 \\ \vdots \\ r_6 \end{bmatrix} = \begin{bmatrix} R_1 \\ \vdots \\ R_6 \end{bmatrix} \qquad \text{eq. (5.6.2.5)}$$

$$\begin{bmatrix} C_{11} & \cdots & C_{16} \\ \vdots & \ddots & \vdots \\ C_{61} & \cdots & C_{66} \end{bmatrix} \begin{bmatrix} \dot{r}_1 \\ \vdots \\ \dot{r}_6 \end{bmatrix} + \begin{bmatrix} K_{11} & \cdots & K_{16} \\ \vdots & \ddots & \vdots \\ K_{61} & \cdots & K_{66} \end{bmatrix} \begin{bmatrix} r_1 \\ \vdots \\ r_6 \end{bmatrix} = \begin{bmatrix} R_1 \\ \vdots \\ R_6 \end{bmatrix} \qquad \text{eq. (5.6.2.6)}$$

If the displacement of the interface node is known, the reaction force may be found, and the damping matrix be calculated:

$$\begin{bmatrix} C_{11} & \cdots & C_{16} \\ \vdots & \ddots & \vdots \\ C_{61} & \cdots & C_{66} \end{bmatrix} \begin{bmatrix} \dot{r}_1 \\ \vdots \\ \dot{r}_6 \end{bmatrix} + \begin{bmatrix} K_{11} & \cdots & K_{16} \\ \vdots & \ddots & \vdots \\ K_{61} & \cdots & K_{66} \end{bmatrix} \begin{bmatrix} r_1 \\ \vdots \\ r_6 \end{bmatrix} = \begin{bmatrix} R_1 \\ \vdots \\ R_6 \end{bmatrix} \qquad \text{eq. (5.6.2.7)}$$

$$\begin{bmatrix} C_{11} & \cdots & C_{16} \\ \vdots & \ddots & \vdots \\ C_{61} & \cdots & C_{66} \end{bmatrix} \begin{bmatrix} \dot{r}_1 \\ \vdots \\ \dot{r}_6 \end{bmatrix} = \begin{bmatrix} R_1 \\ \vdots \\ R_6 \end{bmatrix} - \begin{bmatrix} K_{11} & \cdots & K_{16} \\ \vdots & \ddots & \vdots \\ K_{61} & \cdots & K_{66} \end{bmatrix} \begin{bmatrix} r_1 \\ \vdots \\ r_6 \end{bmatrix} \qquad \text{eq. (5.6.2.8)}$$

$$\begin{bmatrix} C_{11} & \cdots & C_{16} \\ \vdots & \ddots & \vdots \\ C_{61} & \cdots & C_{66} \end{bmatrix} [1] = \begin{bmatrix} R_1 \\ \vdots \\ R_6 \end{bmatrix} - \begin{bmatrix} K_{11} & \cdots & K_{16} \\ \vdots & \ddots & \vdots \\ K_{61} & \cdots & K_{66} \end{bmatrix} \begin{bmatrix} r_1 \\ \vdots \\ r_6 \end{bmatrix} \qquad \text{eq. (5.6.2.9)}$$

$$\begin{bmatrix} C_{11} & \cdots & C_{16} \\ \vdots & \ddots & \vdots \\ C_{61} & \cdots & C_{66} \end{bmatrix} = \begin{bmatrix} R_1 \\ \vdots \\ R_6 \end{bmatrix} - \begin{bmatrix} K_{11} & \cdots & K_{16} \\ \vdots & \ddots & \vdots \\ K_{61} & \cdots & K_{66} \end{bmatrix} \begin{bmatrix} r_1 \\ \vdots \\ r_6 \end{bmatrix} \qquad \text{eq. (5.6.2.10)}$$

To extract the damping matrix, the analysis software “USFOS” is used. The analysis sequence is represented as *Figure 33 - Analysis for [C] matrix extraction*.

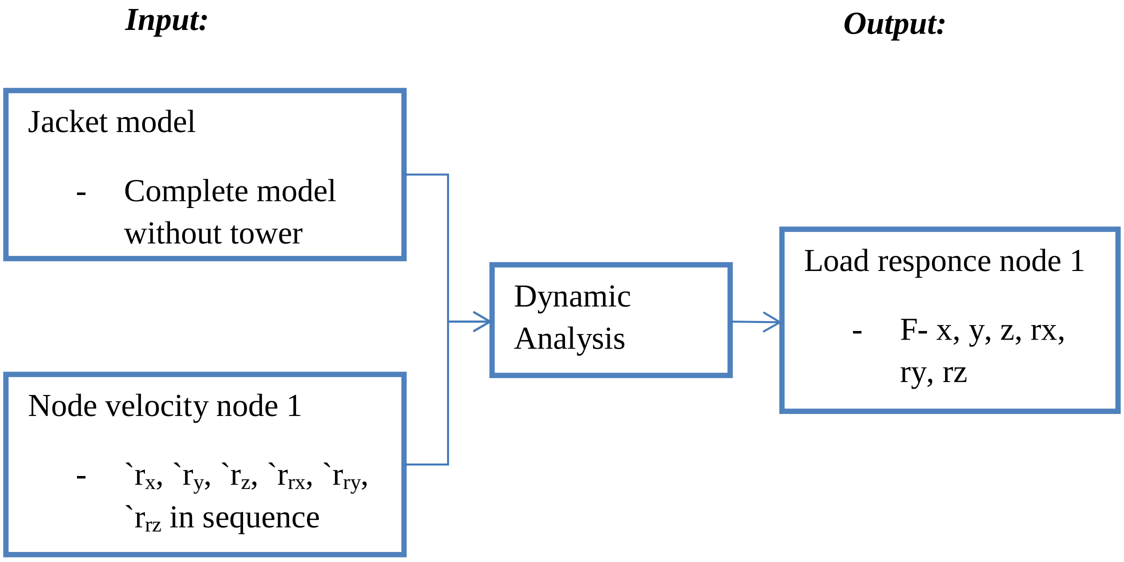


Figure 33 - Analysis for [C] matrix extraction

Forced velocity (`r) (or acceleration) at a node in USFOS is not possible. The preferred method would be to assign a initial velocity (`r) where the analysis software calculates equalibrium at step 0. In this way the dynamic response and the structure stiffness would not interact with the force response, and the results could be read directly.

Unfortenatly this is not the case, and an alternative method has to be used. Instead a lineary forced displacement is applied the interface node. The main challenge for a good result is to avoid execessive flutctuations in the structure while applying the displacement. There are two paramteters affecting structural fluctations; the length and the speed of displacement. The maximum displacement possible without nonlinearity (buckling) for all DOF’s was first found, see *Table 14 - Jacket structure instability limit*.

Table 14 - Jacket structure instability limit

Variable	**Input data [m]**	**Buckling instability [m]**	**Tolerable value [m]**	**Value for analyses [m]**
r_x	1.00	0.520	0.510	0.500
r_y	1.00	0.520	0.510	0.500
r_z	0.20	0.056	0.055	0.050
r_{rx}	0.10	0.006	0.005	0.005
r_{ry}	0.10	0.006	0.005	0.005
r_{rz}	0.10	0.032	0.031	0.030

When forcing a displacement in USFOS, the node gets maximum speed at the first increment. Because of the structure interia, only the node will achieve this velocity. This causes a shock wave propagating through the structure. This effect will act as undesirable noise in the analysis. When applying this method it is important that the noise has the time to decay before the reasults are retrived.

When the described displacement, or velocity, was applied at the interface node at equalibrium, the shock wave in the structure got to large for the noise to cancel out before the instabillity limit was reached. In order to prevent this the instability limit, displacement was forced at the interface node in negative direction at a static step. The displacement (velocity) at the dynamic step was thus from the negative instability limit to the positive instability limit. In this way the length of the displacement got twice as big, and the noise got more time to extinct. As an additional effect of this method the stiffness of the structure act in the positive velocity direction at the beginning of the dynamic step. In this way the structure is attempting to achieve the velocity of the interface node, and the shock wave that occurs gets smaller than at the first method.

Table 15 - Displacement analysis 1-3 for damping matric [C]

Analysis	**1**			**2**			**3**		
Step	***1***	***2***	***3***	***1***	***2***	***3***	***1***	***2***	***3***
r1 [m]	0	-0.5	0.5	0	0	0	0	0	0
r2 [m]	0	0	0	0	-0.5	0.5	0	0	0
r3 [m]	0	0	0	0	0	0	0	-0.05	0.05
r4 [m]	0	0	0	0	0	0	0	0	0
r5 [m]	0	0	0	0	0	0	0	0	0
r6 [m]	0	0	0	0	0	0	0	0	0

Table 16 - Displacement analysis 4-6 for damping matrix [C]

Analysis	**4**			**5**			**6**		
Step	***1***	***2***	***3***	***1***	***2***	***3***	***1***	***2***	***3***
r1 [m]	0	0	0	0	0	0	0	0	0
r2 [m]	0	0	0	0	0	0	0	0	0
r3 [m]	0	0	0	0	0	0	0	0	0
r4 [m]	0	-0.005	0.005	0	0	0	0	0	0
r5 [m]	0	0	0	0	-0.005	0.005	0	0	0
r6 [m]	0	0	0	0	0	0	0	-0.03	0.03

Table 17 - Velocity analysis 1-6 for damping matrix [C]

Analysis	**1**		**2**		**3**		**4**		**5**		**6**	
Step	***1***	***2-3***	***1***	***2-3***	***1***	***2-3***	***1***	***2-3***	***1***	***2-3***	***1***	***2-3***
`r1 [m/s]	0	1	0	0	0	0	0	0	0	0	0	0
`r2 [m/s]	0	0	0	1	0	0	0	0	0	0	0	0
`r3 [m/s]	0	0	0	0	0	0.5	0	0	0	0	0	0
`r4 [m/s]	0	0	0	0	0	0	0	0.5	0	0	0	0
`r5 [m/s]	0	0	0	0	0	0	0	0	0	0.5	0	0
`r6 [m/s]	0	0	0	0	0	0	0	0	0	0	0	0.5

Table 17 - Velocity analysis 1-6 for damping matrix [C] shows the final values of forced velocity for the interface node. It was important to achieve a high enough velocity to gain sufficient damping in the system. Without a significant damping the calculation of the damping factors will be inaccurate and give poor analysis results. On the other hand, a too high velocity will result in that the noise from the shock wave wont die, and the readings will be incorrect. The table values is therefore a result of trial and error.

Figure 34 - Damping matrix reaction force R11 to *Figure 36 - Damping matrix reaction force R51* is some examples of reaction forces for the damping matrix calculations. The graph “***_**_Cij” is the reaction force from the analyses, while the graph “***_**_CijL” is the linear apporxemation of the force history for step 3.

Figure 34 - Damping matrix reaction force R11 is reaction force in main velocity direction. The first ¼ of step 3 (2.00-2.25 s) is dominated by noice from the shock wave. The rest of the step is dominated by the structure fluctations caused by the structure dynamics. A positive aspect is that the fluctations are quite even. When looking at *Figure 35 - Damping matrix reaction force R21*, where the reaction force is transverse to the velocity direction, the fluctations are dominating. In this direction the result would not be reliable. Rotation, *Figure 36 - Damping matrix reaction force R51*, gives at the other hand quite clean readings.

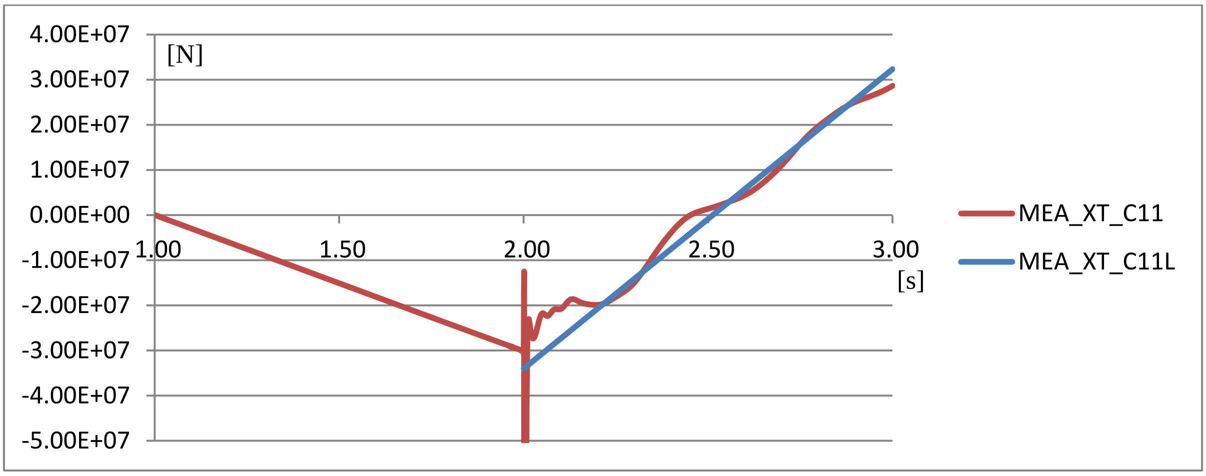


Figure 34 - Damping matrix reaction force R11

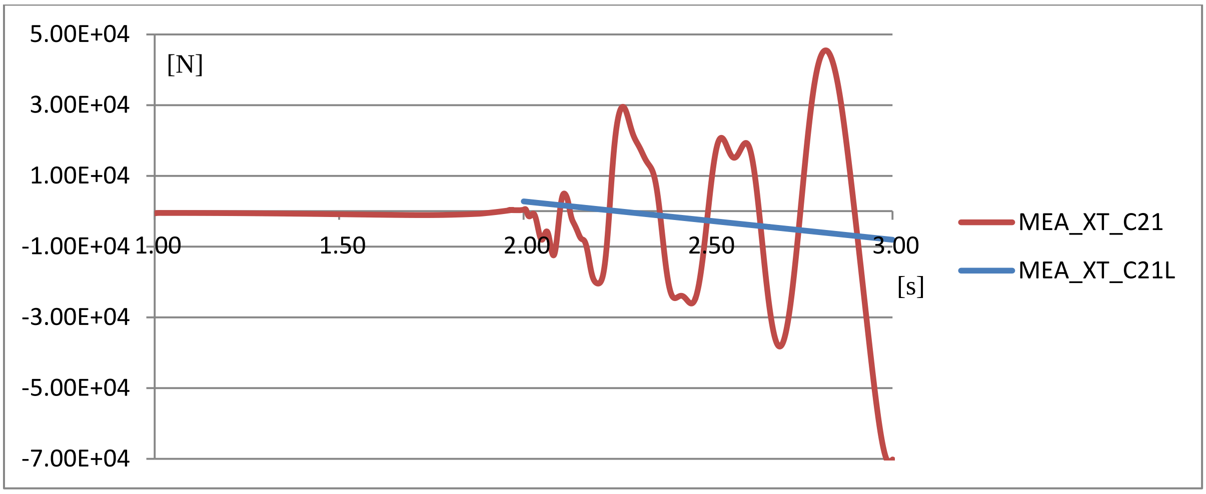


Figure 35 - Damping matrix reaction force R21

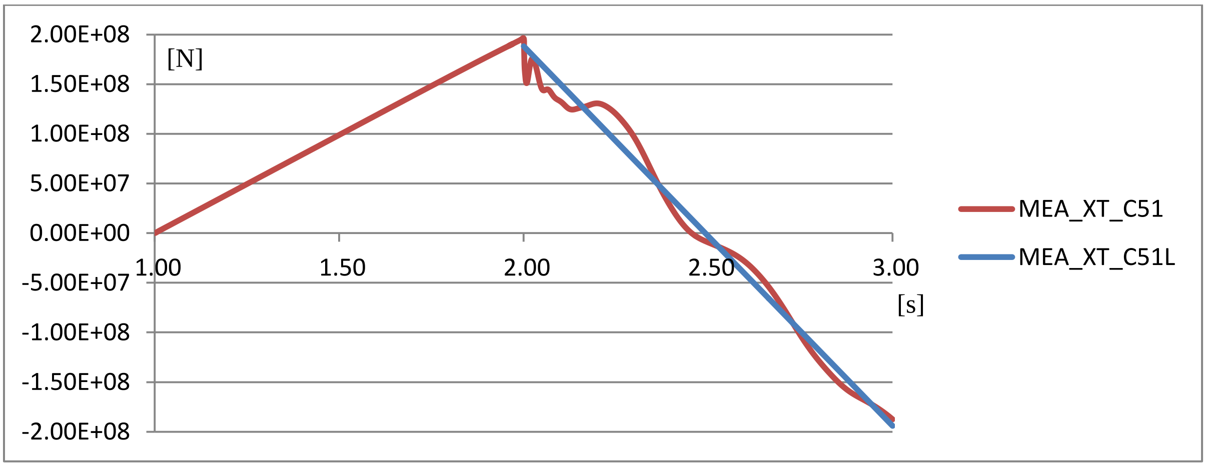


Figure 36 - Damping matrix reaction force R51

The analysis method gives good results when analysing the main directions. Of this reason only the diagonal damping matrix is extracted for the interface node. When the matrix is made for FEDEM for a sequentially coupled analysis series it will not cause any difference. FEDEM has at this point only possibillity to integrate a diagonal damping matrix for a node.

Calculations for analysis 1 are derived as following:

$$\left[R^{(s3-s1)}\right] = \left[R^{(s3)}\right] - \left[R^{(s1)}\right] = \begin{bmatrix} {R_1}^{(s3)} \\ \vdots \\ {R_6}^{(s3)} \end{bmatrix} - \begin{bmatrix} {R_1}^{(s1)} \\ \vdots \\ {R_6}^{(s1)} \end{bmatrix} \qquad \text{eq. (5.6.2.12)}$$

$$\left[R^{(s3-s1)}\right] = \begin{bmatrix} 32315790 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} - \begin{bmatrix} -831 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} = \begin{bmatrix} 32316621 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} \qquad \text{eq. (5.6.2.13)}$$

The dynamic equation of motion:

$$[M][\ddot{r}] + [C][\dot{r}] + [K][r] = [R] \qquad \text{eq. (5.6.2.14)}$$

$$[M][0] + [C][\dot{r}] + [K][r] = [R] \qquad \text{eq. (5.6.2.15)}$$

$$[C][\dot{r}] + [K][r] = [R] \qquad \text{eq. (5.6.2.16)}$$

$$[C]\begin{bmatrix} 1 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} + [K]\begin{bmatrix} 0.5 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} = \begin{bmatrix} 32316621 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} \qquad \text{eq. (5.6.2.17)}$$

$$[C]\begin{bmatrix} 1 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} = \begin{bmatrix} 32316621 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} - [K]\begin{bmatrix} 0.5 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} \qquad \text{eq. (5.6.2.18)}$$

$$[C1] = \begin{bmatrix} 32316621 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} - \begin{bmatrix} 30237064 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} \qquad \text{eq. (5.6.2.19)}$$

$$[C1] = \begin{bmatrix} 2079557 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix} \qquad \text{eq. (5.6.2.20)}$$

When the calculation is repeated for all analyses the complete damping matrix is as following:

$$[C] = \begin{bmatrix} 2079557 & 0 & 0 & 0 & 0 & 0 \\ 0 & 2041690 & 0 & 0 & 0 & 0 \\ 0 & 0 & 525296 & 0 & 0 & 0 \\ 0 & 0 & 0 & 168585573 & 0 & 0 \\ 0 & 0 & 0 & 0 & 168258766 & 0 \\ 0 & 0 & 0 & 0 & 0 & 201781469 \end{bmatrix} \text{eq. (5.6.2.21)}$$

In FEDEM Windpower the damping matrix is imported as damping constants. The damping constants are defined as the ratio between the damping matrix [C] and the stiffness matrix [K]:

$$kC = C_{ij}/K_{ij} \qquad \text{eq. (5.6.2.22)}$$

$$[kC] = \begin{bmatrix} 0{,}0344 \\ 0{,}0338 \\ 0{,}0006 \\ 0{,}0047 \\ 0{,}0047 \\ 0{,}0237 \end{bmatrix} \qquad \text{eq. (5.6.2.23)}$$

5.6.3 Mass matrix [M]

To find the mass matrix, three methods were tried.

5.6.3.1 Forced controlled dynamic of motion

The mass matrix [M] is a quantity affecting the structural dynamic. To be able to find this parameter through the structure response of an external force it was necessary to use the dynamic equation of motion:

$$[M][\ddot{r}] + [C][\dot{r}] + [K][r] = [R] \qquad \text{eq. (5.6.3.1)}$$

To extract the mass matrix, the analysis software "USFOS" is used. The analysis sequence is represented as *Figure 37 - Analysis for mass matrix [M] extraction*.

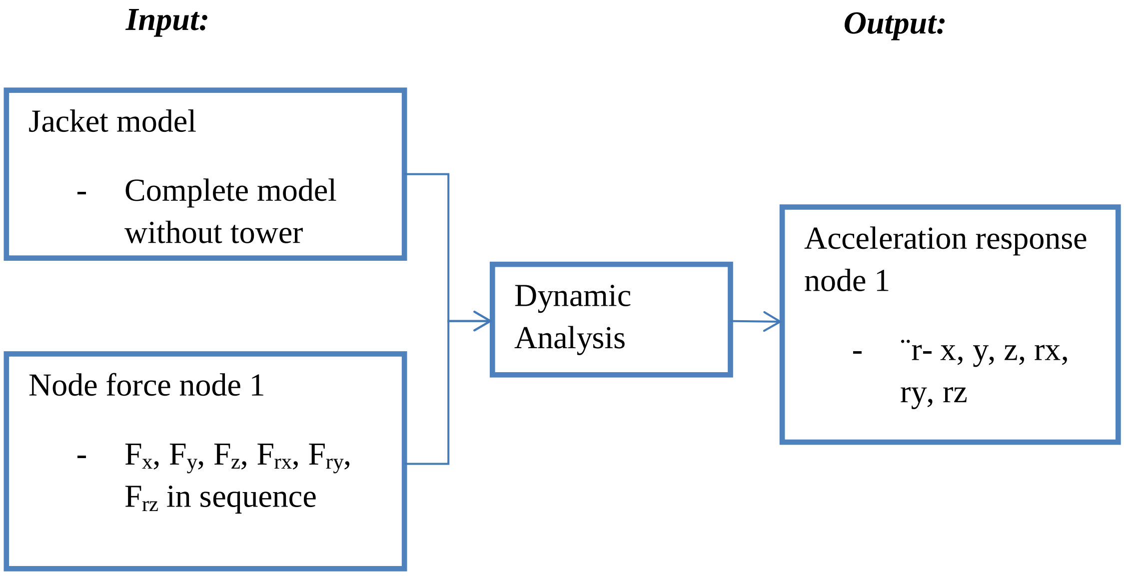


Figure 37 - Analysis for mass matrix [M] extraction

A nodeload was applied the interface node at increment 1 of the dynamic analysis step. The acceleration and velocity was recorded for all DOF's of the node. This nodeload procedure was repeated for all directions.

All the analysis experiments are represented in *Table 18 - Analyses conducted for force activated structure acceleration for mass matrix [M]*. Analysis FR01 to FR03 is an analysis series where the nodal force sensitivity and the increment sensitivity are tested. FR00 is an analysis test to check its response to high loads.

Table 18 - Analyses conducted for force activated structure acceleration for mass matrix [M]

Analysis Name:	**Nodeforce** *[N]*	***0.1 inc*** [s]	***0.05 inc*** [s]	***0.01 inc*** [s]	***0.005 inc*** [s]	***0.001 inc*** [s]
RF00	*1 000 000*			-13.5 [m/s^2]		
RF01	*1 000*	.0022 [m/s^2]	.0044 [m/s^2]	.0135 [m/s^2]	.0203 [m/s^2]	.0225 [m/s^2]
RF02	*850*	.0019 [m/s^2]	.0038 [m/s^2]	.0115 [m/s^2]	.0173 [m/s^2]	.0382 [m/s^2]
RF03	*500*	.0011 [m/s^2]	.0022 [m/s^2]	.0068 [m/s^2]	.0102 [m/s^2]	.0225 [m/s^2]

The acceleration history for the different tests is illustrated with *Figure 38 – FR01 - Acceleration due to node force at 1000 N* to *Figure 41 - FR00 - Acceleration due to node force at 1 000 000 N*. The analysis results are typical for all load cases. The acceleration is highly dependent on the increment size. There is too much noise in the signal to be able to extract reasonable results. Ideally it is desired to get a pure acceleration after the shock wave has extinct. One way to improve the results is to filter and levelling the time-displacement curve, and calculates the acceleration curve from this. In this way it is possible to achieve a plain curve, but there will still be challenging to extract consistent results.

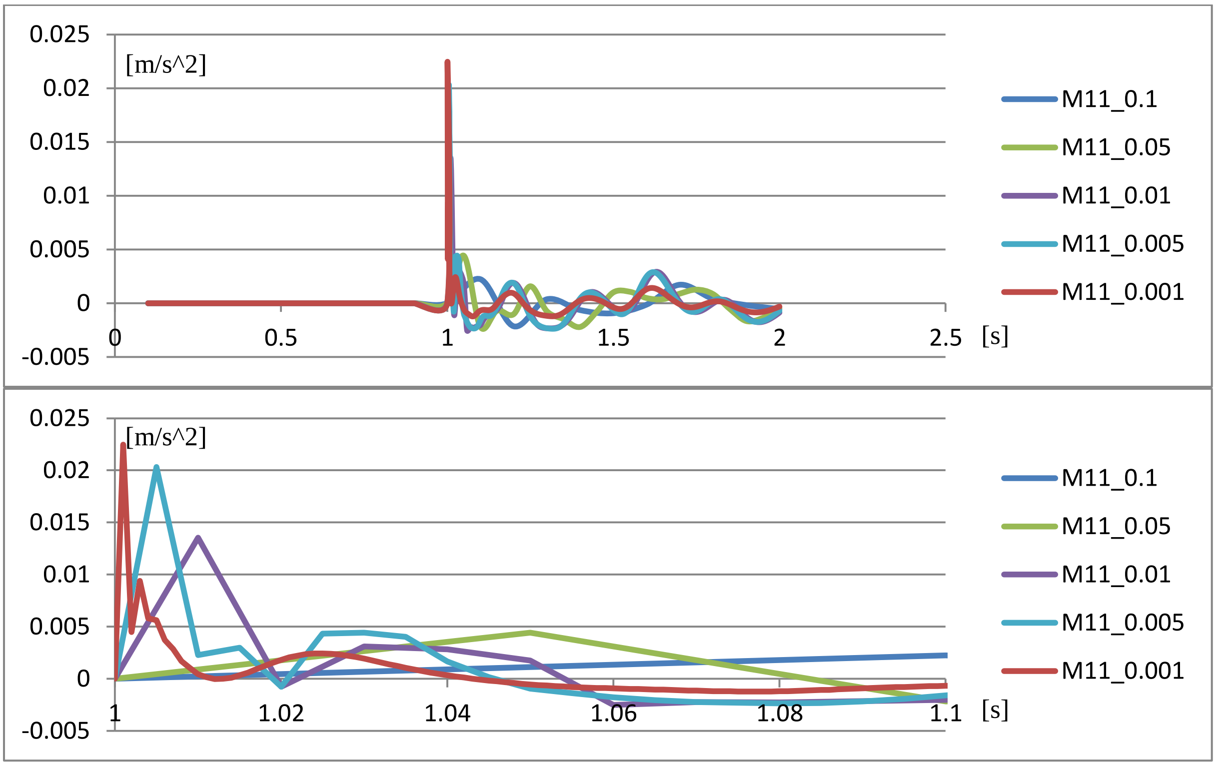


Figure 38 – FR01 - Acceleration due to node force at 1000 N

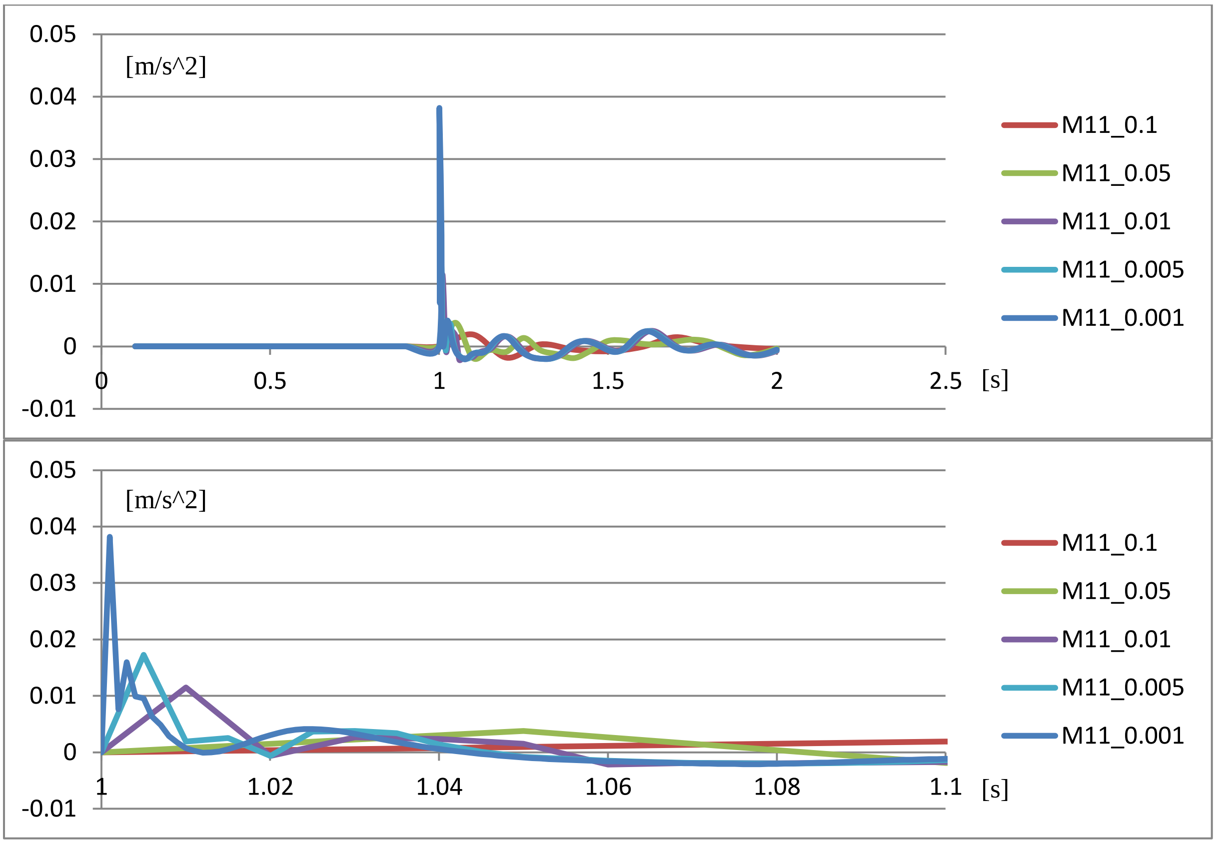


Figure 39 - FR02 - Acceleration due to node force at 850 N

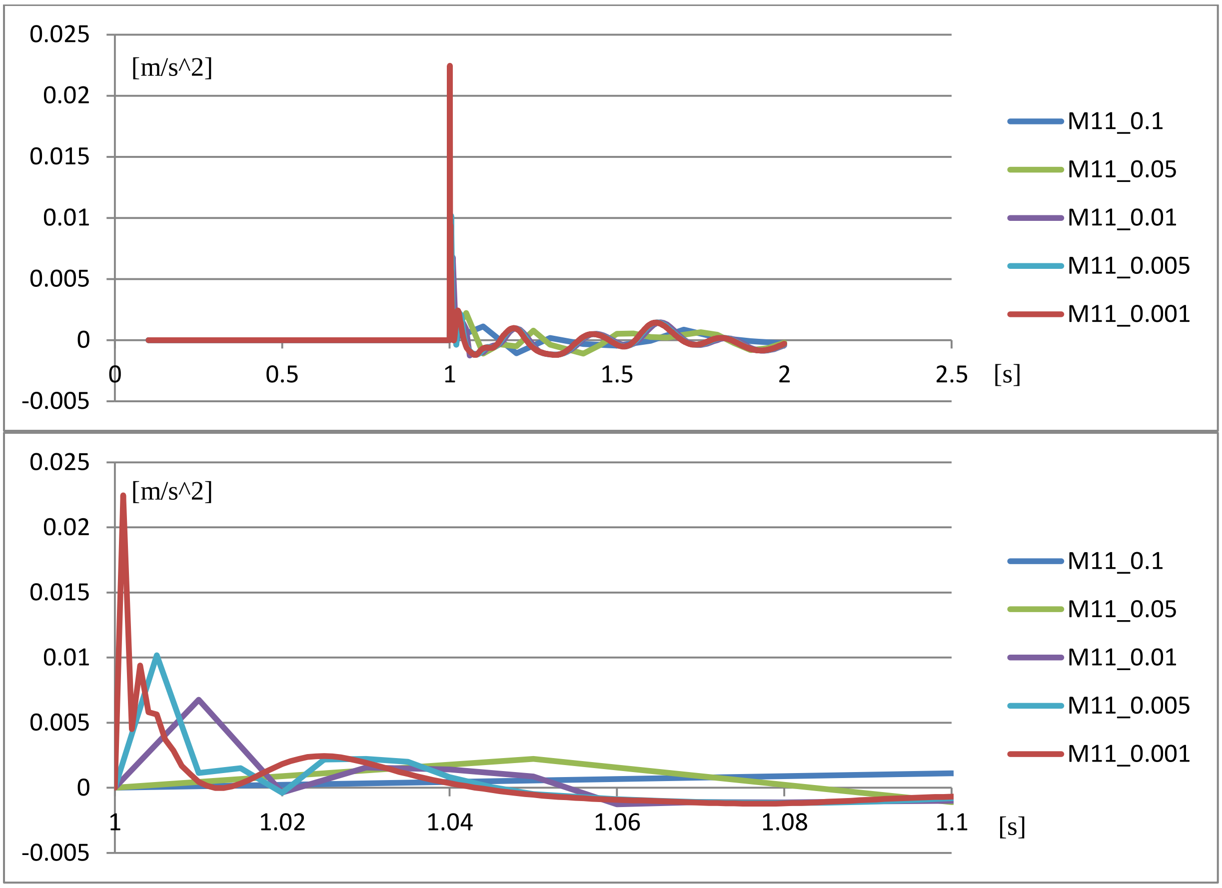


Figure 40 - FR03 - Acceleration due to node force at 500 N

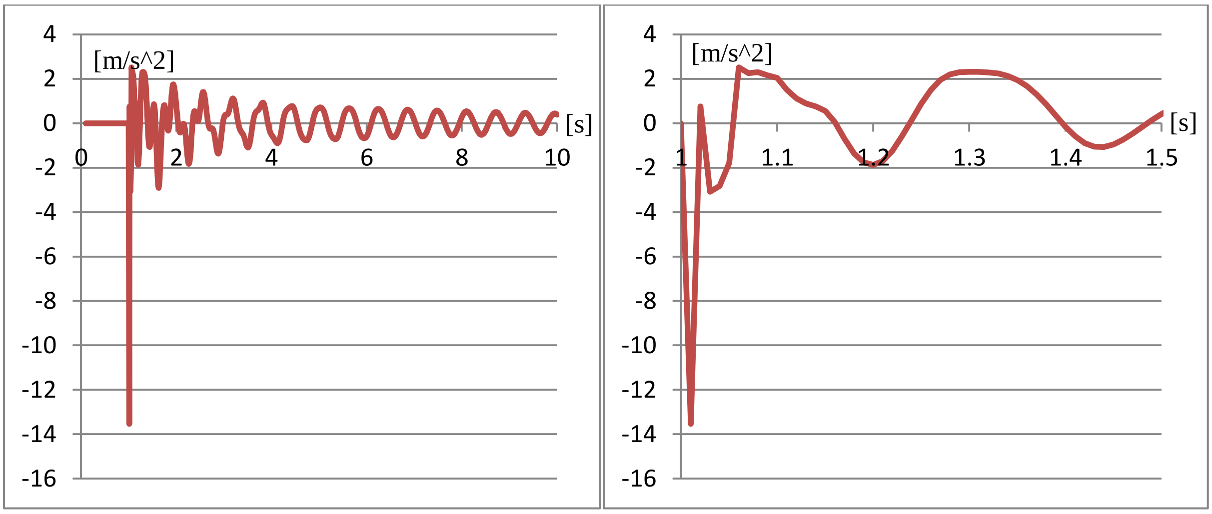


Figure 41 - FR00 - Acceleration due to node force at 1 000 000 N

5.6.3.2 Displacement controlled dynamic of motion

There is also possible to track the node reaction force by performing a controlled acceleration in the interface node. In basic this should be a simple operation. USFOS do have a node acceleration controlled feature, called “NODEACC”. This feature gives inconsistent results, and therefore do not work. To be able to apply the acceleration to the node it was necessary to calculate a displacement time series giving the specific acceleration.

To be able to find the mass matrix through the structure response of a displacement controlled acceleration it was necessary to use the dynamic equation of motion:

$$[M][\ddot{r}] + [C][\dot{r}] + [K][r] = [R] \qquad \text{eq. (5.6.3.2)}$$

To extract the mass matrix, the analysis software “USFOS” is used. The analysis sequence is represented as *Figure 42 - Analysis for mass matrix [M] extraction*.

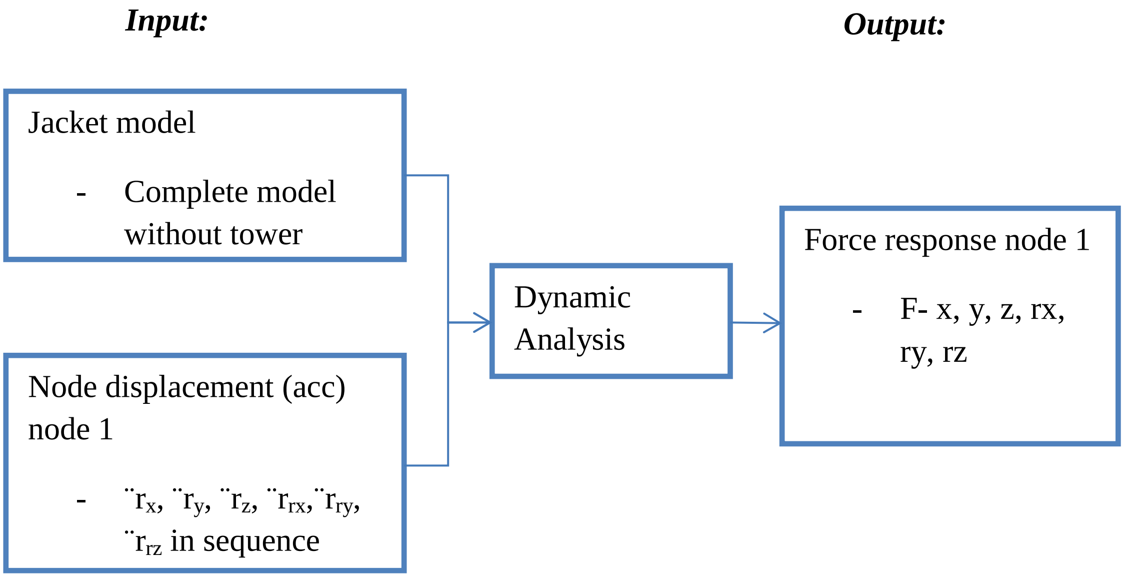


Figure 42 - Analysis for mass matrix [M] extraction

The reaction force time series for the interface node under a acceleration at 1 m/s^2 in x-direction is given by *Figure 43 - Reaction force at 1m/s^2, 0.001 s increment*. Basically the results is quite good, when having in mind the metod is a dynamic approach. When the mass matrix is calculated on the basis of the dynamic equation of motion using the stiffness and damping matrix calculated, the mass in x-direction is ~270 000 kg. This turns out, during the next extraction method, to not be too far from the reality. The challenge with this method is that these results is the best found from a displacement size of 0.5 m for x and y-direction. For z-direction, and x, y, z, rotation, the buckling limit is much lower. The problem is that either the acceleration is too low, or a shock wave occur. The conclution is that this method is only possible to be used in limited cases, and if it is the results will be inaccurate and to low.

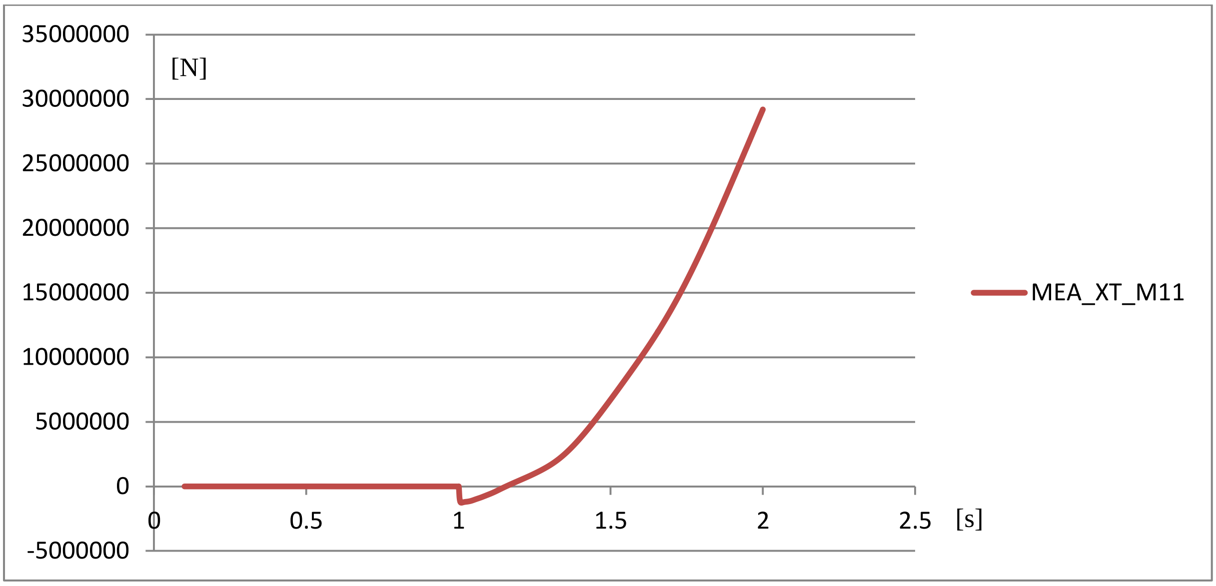


Figure 43 - Reaction force at 1m/s^2, 0.001 s increment

5.6.3.3 Reaction force from gravity field

The methods for extracting the mass matrix [M] with use of the dynamic equation of motion gave poor results. A method to avoid using being exposed to dynamic effects of the structure was to use the gravity field feature in USFOS.

To extract the mass matrix, the analysis software "USFOS" is used. The analysis sequence is represented as *Figure 44 - Analysis for mass matrix [M] extraction*.

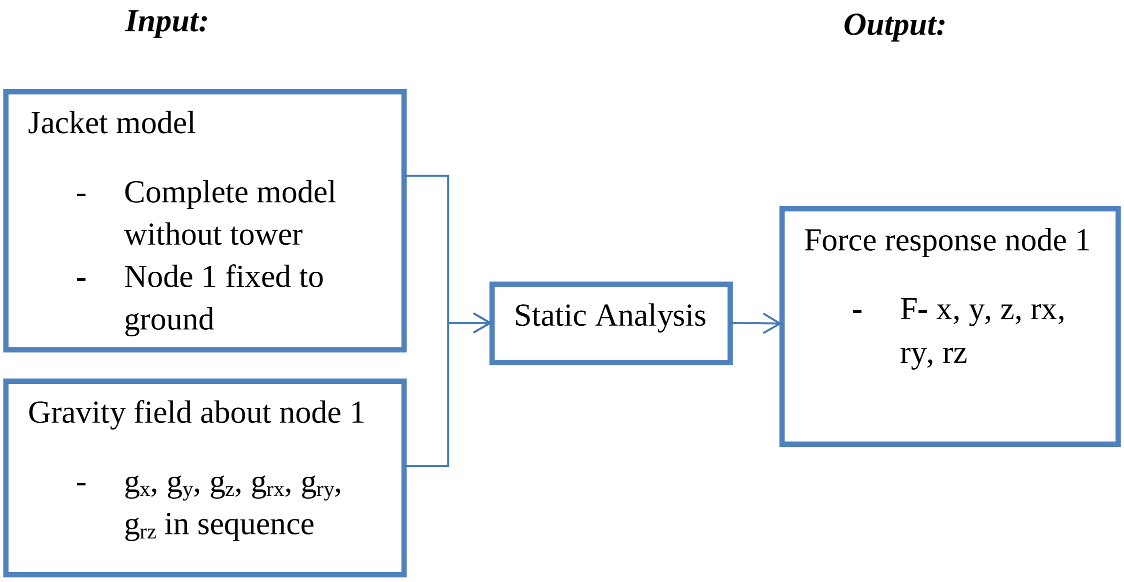


Figure 44 - Analysis for mass matrix [M] extraction

A static analysis was run with the jacket structure without tower. The interface node (node 1) was fixed in all degrees of freedom. To extract all mass values, 6 analyses, one in each directon, was executed. Analysis 1-3 uses the ordinary gravity command, see *Table 19 - USFOS GRAVITY command*. It means the gravity forces act out of the plane. In this way the reaction forces in node 1 represent the interaction of the structure mass with the accfield for node 1.

For analysis 4-6 the purpose is to extract the moment of interia of node 1. In this case the ordinary gravity command is quite limited. For this case the command ACCFIELD was used, se *Table 20- USFOS ACCFIELD command*. aRx, aRy and aRz represent the rotational acc field. Xc, Yc and Zc represent the coordinate in space where the acc field revolve around. This coordinate was set to node 1, and acc field was set to 1.

Table 19 - USFOS GRAVITY command

Analysis	GRAVITY	Load Case	Ax	Ay	Az
1		1	1	0	0
2		1	0	1	0
3		1	0	0	1

Table 20- USFOS ACCFIELD command

Analysis	ACCFIELD	Load Case	Type	Opt	aRx	aRy	aRz	Xc	Yc	Zc
4		1	Rot	Stat	1	0	0	0	0	25.44
5		1	Rot	Stat	0	1	0	0	0	25.44
6		1	Rot	Stat	0	0	1	0	0	25.44

Example of reaction force for node 1 at analysis 1 direction 1, see *Figure 45 - Reaction force for [M11]*.

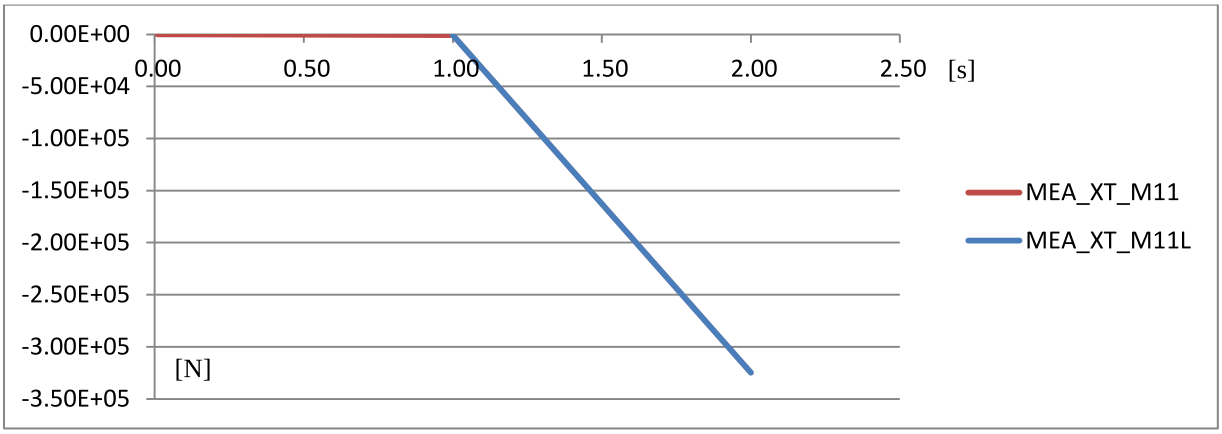


Figure 45 - Reaction force for [M11]

Calculations for analysis 1 are derived as following:

$$\left[R^{(s2-s1)}\right] = abs\left(\left[R^{(s2)}\right]-\left[R^{(s1)}\right]\right) = abs\left(\begin{bmatrix}{R_1}^{(s2)}\\ \vdots \\ {R_6}^{(s2)}\end{bmatrix} - \begin{bmatrix}{R_1}^{(s1)}\\ \vdots \\ {R_6}^{(s1)}\end{bmatrix}\right) \qquad \text{eq. (5.6.3.3.1)}$$

$$\left[R^{(s2-s1)}\right] = abs\left(\begin{bmatrix}-324770\\ -785\\ -141796\\ -2644\\ 1570353\\ 6326\end{bmatrix} - \begin{bmatrix}-727\\ -771\\ -142790\\ -2418\\ 2450\\ -18660\end{bmatrix}\right) = \begin{bmatrix}324044\\ 13\\ 994\\ 226\\ 1567903\\ 24986\end{bmatrix} \qquad \text{eq. (5.6.3.3.2)}$$

Newtons 2. law:

$$F = m * a$$ eq. (5.6.3.3.3)

$$F = m * a$$ eq. (5.6.3.3.4)

$$m = F/a$$ eq. (5.6.3.3.5)

$$[m1] = \begin{bmatrix} 324044 \\ 13 \\ 994 \\ 226 \\ 1567903 \\ 24986 \end{bmatrix} /1 = \begin{bmatrix} 324044 \\ 13 \\ 994 \\ 226 \\ 1567903 \\ 24986 \end{bmatrix}$$ eq. (5.6.3.3.6)

When the calculation is repeated for all analyses the complete mass matrix is as following:

$$[M] = \begin{bmatrix} 324044 & 48 & 210 & 947 & 6376907 & 33626 \\ 13 & 323941 & 214 & 6385861 & 226 & 33657 \\ 994 & 1007 & 200909 & 88453 & 42884 & 13869 \\ 226 & 1573430 & 5661 & 42718725 & 13638 & 221093 \\ 1567903 & 601 & 5655 & 23827 & 42760632 & 224107 \\ 24986 & 24841 & 77 & 679603 & 684854 & 23888580 \end{bmatrix}$$ eq. (5.6.3.3.7)

FEDEM has only the possibility to use the diagonal mass matrix:

$$[M] = \begin{bmatrix} 324044 \\ 323941 \\ 200909 \\ 42718725 \\ 42760632 \\ 23888580 \end{bmatrix}$$ eq. (5.6.3.3.8)

5.6.4 Matrices analysis extraction tool

To determine the matrices there was a need of a complex analysis filesystem. The files had to be changed an indefinite number of times during tweaking of the input and output data. To create and change these files manually would be time-consuming task, with high probability of errors that are dificult to find. To be able to find the matrices in a effective way a matrix genreation script system was created.

5.6.4.1 Matrix tool overview

The system script is written to perform all sequences in the calculation automatically. The first step is to insert the input data for the calculation, such as analysis increment time, deflection, velocity, etc for all matrices. All data may be specified for each sub-analysis alone. There are three steps in the analysis, illustrated as *Figure 46 - Matrix tool overview*.

In the first step the system script generates the entire file system needed to execute the matrix tool. The generation is performed by a number of "loop" scripts which sets the unique variables for each file.

When all files are generated the system script initiate step two. In this step all matrix analyses are executed in sequence. When the analyses are finished the software "Xact" is opened, and the result extraction scripts are run.

When the result files are extracted, step three is initialized. In this step all result files are imported to excel files for preprocessing. Each matrix has its own excel file that calculates the matrix for the interface node.

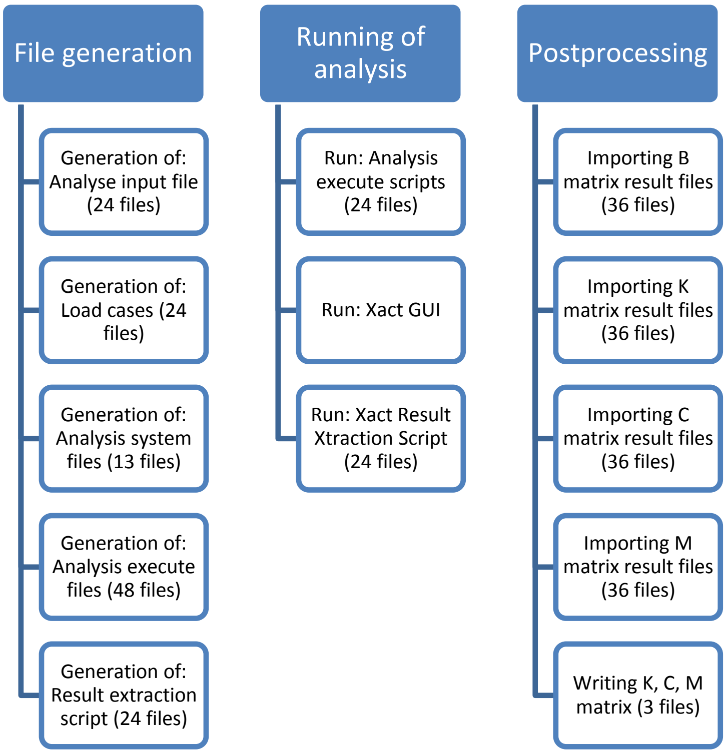


Figure 46 - Matrix tool overview

5.6.4.2 **Matrix tool file system**

The file system is listed from *Table 21 - Matrix system tool macro* to *Table 32 - Matrix analysis [M] results processing*. All the macros listed are attached in chapter 11.30.

The file “01_01_SRM.xlsx” at step 1.01 contains the system run macro. This is the head macro that calls all sub macros in the file system in correct sequence.

Table 21 - Matrix system tool macro

Step:	***Main File***	***Macro name***	***Output file***
0.00	00_00_AM Analysis Map xlsx		
1.01	01_01_SRM System Run Macro xlsx	SRM System Run Macro	

The file “01_02_MAIF.xlsx” at step 1.02 contains the control files for the analyses of [B], [K], [C], and [M] matrix. The macros “CF_#” (eg. CF_B) insets the input values specific for the analysis and writes the file to “CF_##.fem” (eg. CF_B1 to 6).

Table 22 - Matrix analysis input file macro

1.02	01_02_MAIF Matrix Analysis Input File xlsx	CF_B	CF_B1 Control File B1 fem
			### ## #
			CF_B6 Control File B6 fem
		CF_K	CF_K1 Control File K1 fem
			### ## #
			CF_K6 Control File K6 fem
		CF_C	CF_C1 Control File C1 fem
			### ## #
			CF_C6 Control File C6 fem
		CF_M	CF_M1 Control File M1 fem
			### ## #
			CF_M6 Control File M6 fem

The file “01_03_ML” at step 1.03 contains the load files for the analyses of [B], [K], [C], and [M] matrix. The macros “ML_FG_#” (eg. ML_FG_C) insets the input values specific for the analysis and writes the file to “ML_FG_##.fem” (eg. ML_FG_C1 to 6).

Table 23 - Matrix analysis load cases macro

1.03	01_03_ML	Matrix Load	xlsx	ML_FG_K	Matrix Load File Generator K	ML_K1	Matrix Load K1	fem
						###	##	#
						ML_K6	Matrix Load K6	fem
				ML_FG_C	Matrix Load File Generator C	ML_C1	Matrix Load C1	fem
						###	##	#
						ML_C6	Matrix Load C6	fem
				ML_FG_M	Matrix Load File Generator M	ML_M1	Matix Load M1	fem
						###	##	#
						ML_M6	Matix Load M6	fem

The file “01_04_RABG” at step 1.04 contains all the batch scripts to run the analyses. This file system is described in chapter 5.4 and the codes are attached as chapter 11.26. The macro simply writes the files to batch scripts in the folder.

Table 24 - Matrix analysis system files macro

1.04	01_04_RABG	Run Analyses Bat Generator	xlsx	RABG	Run Analyses Bat Generator	_00_00_CleanBat	bat
						_01_01_RunUSFOS	bat
						_01_02_RunDynres	bat
						_01_03_RunPostfos	bat
						_01_04_RunXact	bat
						_02_00_RunAllAnalyses	bat
						_02_01_RunUSFOSAnalysis	bat
						_02_02_RunDynResAnalysis	bat
						_02_03_RunPostfosAnalysis	bat
						_03_00_CleanAllResults	bat
						_03_01_CleanUSFOSResults	bat
						_03_02_CleanDynresResults	bat
						_03_03_CleanPostfosResults	bat

The file “01_05_MEABG” at step 1.05 is a analysis file generator for USFOS, DynRes and Postfos analyses. The workbook main function is described in chapter 5.4.3. This version is specialized for this analysis series. The scripts “MEAGBU_#” (eg. MEAGBU_K) automatically inserts the correct analysis specifications and print it to file. It is automatically looped through all analyses.

Table 25 - Matrix analysis execute files macro

1.05	01_05_ MEABG	Matrix Extract Analysis Bat Generator	xlsx	MEABGU_K	Matrix Extract Analysis Bat Generator USFOS K	MA_K1_UA	Matrix Analysis K1 USFOS Analysis	bat
						MA_K1_UAR	Matrix Analysis K1 USFOS Analysis Redir	txt
						###	##	#
						###	##	#
						MA_K6_UA	Matrix Analysis K6 USFOS Analysis	bat
						MA_K6_UAR	Matrix Analysis K6 USFOS Analysis Redir	txt
				MEABGU_C	Matrix Extract Analysis Bat Generator USFOS C	MA_C1_UA	Matrix Analysis C1 USFOS Analysis	bat
						MA_C1_UAR	Matrix Analysis C1 USFOS Analysis Redir	txt
						###	##	#
						###	##	#
						MA_C6_UA	Matrix Analysis C6 USFOS Analysis	bat
						MA_C6_UAR	Matrix Analysis C6 USFOS Analysis Redir	txt
				MEABGU_M	Matrix Extract Analysis Bat Generator USFOS M	MA_M1_UA	Matrix Analysis M1 USFOS Analysis	bat
						MA_M1_UAR	Matrix Analysis M1 USFOS Analysis Redir	txt
						###	##	#
						###	##	#
						MA_M6_UA	Matrix Analysis M6 USFOS Analysis	bat
						MA_M6_UAR	Matrix Analysis M6 USFOS Analysis Redir	txt

The file “02_01_MEA_XS.xlsx” at step 2.01 contains the macros to create the Xact result extraction scripts. There are three macros run in sequence, one for each matrix [K], [C], and [M]. Each matrix get 6 scripts, each writing 6 result time histories.

Table 26 - Matrix analysis result extraction script macro

2.01	02_01_MEA_XS	Matrix Extraction Analysis Xact Script	xlsx	MEA_XS_K	Matrix Extraction Analysis Xact Script K	MEA_XS_K1	Matrix Extraction Analysis Xact Script K1	usf
						###	##	#
						MEA_XS_K6	Matrix Extraction Analysis Xact Script K6	usf
				MEA_XS_C	Matrix Extraction Analysis Xact Script C	MEA_XS_C1	Matrix Extraction Analysis Xact Script C1	usf
						###	##	#
						MEA_XS_C6	Matrix Extraction Analysis Xact Script C6	usf
				MEA_XS_M	Matrix Extraction Analysis Xact Script M	MEA_XS_M1	Matrix Extraction Analysis Xact Script M1	usf
						###	##	#
						MEA_XS_M6	Matrix Extraction Analysis Xact Script M6	usf

The file “02_02_XME.xlsx” at step 2.02 contains the macro to run the Xact scripts to extract the stiffness matrix result files from the analyses.

Table 27 - Matrix analysis [K] result extraction macro

2.02	02_02_XME	Xact Matrix Extraction	xlsx	XME_K	Xact Matrix Extraction K	MEA_XT_K1	Matrix Extraction Analysis Xact Table K11	.
						###	##	#
						MEA_XT_K6	Matrix Extraction Analysis Xact Table K66	.

The file “02_03_XME.xlsx” at step 2.03 contains the macro to run the Xact scripts to extract the damping matrix result files from the analyses.

Table 28 - Matrix analysis [C] result extraction macro

2.03	02_02_XME	Xact Matrix Extraction	xlsx	XME_C	Xact Matrix Extraction C	MEA_XT_C1	Matrix Extraction Analysis Xact Table C11	.
						###	##	#
						MEA_XT_C6	Matrix Extraction Analysis Xact Table C66	.

The file “02_04_XME.xlsx” at step 2.04 contains the macro to run the Xact scripts to extract the mass matrix result files from the analyses.

Table 29 - Matrix analysis [M] result extraction macro

2.04	02_02_XME	Xact Matrix Extraction	xlsx	XME_M	Xact Matrix Extraction M	MEA_XT_M1	Matrix Extraction Analysis Xact Table M11	.
						###	##	#
						MEA_XT_M6	Matrix Extraction Analysis Xact Table M66	.

The file “03_01_MEA_XTP_K” at step 3.01 contains the macros to post processing the results from the analyses. The macro “MEA_XTP_K” imports the analysis result time series. The macro “MEA_XTP_KL” generates the linear curve for a selected range of the time series. The macro “MEA_TG_KL” generates both the graph for the original timeseries together with the linear approach. The macro “MEA_ME_K” writes the complete calculated matrix to “K_Matrix.txt” in the analysis folder.

Table 30 - Matrix analysis [K] results processing

3.01	03_01_MEA_XTP_K	Matrix Extration Analysis Xact Table Processing K	xlsx	MEA_XTP_K	Matrix Extration Analysis Xact Table Processing K		
				MEA_XTP_KL	Matrix Extration Analysis Xact Table Processing K Linear		
				MEA_TG_K	Matrix Extration Analysis Table Graph K		
				MEA_TG_KL	Matrix Extration Analysis Table Graph K Linear		
				MEA_ME_K	Matrix Extration Analysis Matrix Extraction K	K_Matrix	.txt

The file “03_02_MEA_XTP_C” at step 3.02 contains the macros to post processing the results from the analyses. The macro “MEA_XTP_C” imports the analysis result time series. The macro “MEA_XTP_CL” generates the linear curve for a selected range of the time series. The macro “MEA_TG_CL” generates both the graph for the original timeseries together with the linear approach. The macro “MEA_ME_C” writes the complete calculated matrix to “C_Matrix.txt” in the analysis folder.

Table 31 - Matrix analysis [C] results processing

3.02	03_02_MEA_XTP_C	Matrix Extration Analysis Xact Table Processing C	xlsx	MEA_XTP_C	Matrix Extration Analysis Xact Table Processing C		
				MEA_XTP_CL	Matrix Extration Analysis Xact Table Processing C Linear		
				MEA_TG_C	Matrix Extration Analysis Table Graph C		
				MEA_TG_CL	Matrix Extration Analysis Table Graph C Linear		
				MEA_ME_C	Matrix Extration Analysis Matrix Extraction C	C_Matrix	.txt

The file “03_03_MEA_XTP_M” at step 3.03 contains the macros to post processing the results from the analyses. The macro “MEA_XTP_M” imports the analysis result time series. The macro “MEA_XTP_ML” generates the linear curve for a selected range of the time series. The macro “MEA_TG_ML” generates both the graph for the original timeseries together with the linear approach. The macro “MEA_ME_M” writes the complete calculated matrix to “M_Matrix.txt” in the analysis folder.

Table 32 - Matrix analysis [M] results processing

3.03	03_03_MEA_XTP_M	Matrix Extration Analysis Xact Table Processing M	xlsx	MEA_XTP_M	Matrix Extration Analysis Xact Table Processing M		
				MEA_XTP_ML	Matrix Extration Analysis Xact Table Processing M Linear		
				MEA_TG_M	Matrix Extration Analysis Table Graph M		
				MEA_TG_ML	Matrix Extration Analysis Table Graph M Linear		
				MEA_ME_M	Matrix Extration Analysis Matrix Extraction M	M_Matrix	.txt

5.7 Creating comparable results between analysis methods

In order to be able to evaluate and compare the analysis methods (fully- and sequentially coupled analysis), a set of common parameters that could be used for both methods needed to be defined. It was decided to compare the two methods by comparing vibration amplitudes at bracer junctions, and a stress based damage parameter describing the fatigue damage accumulated during the analysis time span.

It was important to use simple parameters that could be derived from both FEDEM and USFOS in the same way. This was desired since we wanted to extract pure signals that were not interpreted by the code in any way.

The signal processing for both parameters were carried out for the analysis time series ranging from 100 to 270 seconds. This was done to avoid the irregular behaviour during the start-up phase due to the onset of wind and the resulting adjustments made by the wind turbine control system.

To evaluate differences in bracer vibrations between the two analysis methods, node displacements of certain nodes were extracted for the investigated bracer. The displacements were treated to extract the relative position of the bracer junction compared to its “ideal” position relative to the structure. The resulting vibration signal could then be processed using FFT and a comparison between methods could be made.

To evaluate accumulated damage predicted by the two different analysis methods, beam end forces and moments were extracted at the joint between bracer and leg. The resulting beam stresses were calculated from the axial forces and bending moments using excel, utilizing the beam diameter and cross section area. The stress signal was interpreted using rainflow counting before applying the Palmgren-Miner rule to calculate the resulting damage.

5.7.1 Extracting bracer vibrations

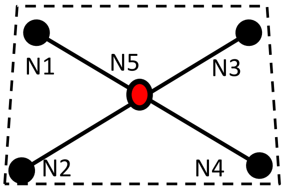


Figure 47 - Idealized bracer junction

From the fully- and sequentially coupled analysis in FEDEM and USFOS, the bracer vibrations had to be extracted.

The positions of each node (N1-N5) from all three bracer junctions were logged during both analysis methods for the whole load history. Since the whole jacket structure is moving during the load sequence, the movement of the bracer junction had to be extracted relative to the movement of the whole jacket. This was achieved by calculating the average *x-y* position of node N1-N4, which should be the ”ideal” location of N5 during the load history.

$$Ideal\ location\ of\ N5:\ N5_i = \frac{N1+N2+N3+N4}{2} \quad \text{eq. (5.7.1.1)}$$

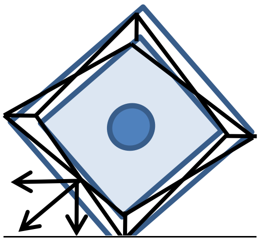


Figure 48 - Jacket positioned 45 degrees to X direction

When subtracting the average position $N5_i$ from the logged position of N5, the approximate out of plane movement $N5_v$ is known.

$$N5_v = N5 - N5_i \qquad \text{eq. (5.7.1.2)}$$

So far, the movements are described by x- and y- coordinates. To get the ”pure” out of plane movement in N5, a transformation matrix was made to read out the junction vibration amplitudes as values alternating between positive and negative values. The transformed signal was then used during further processing of the vibration signal $N5_V$.

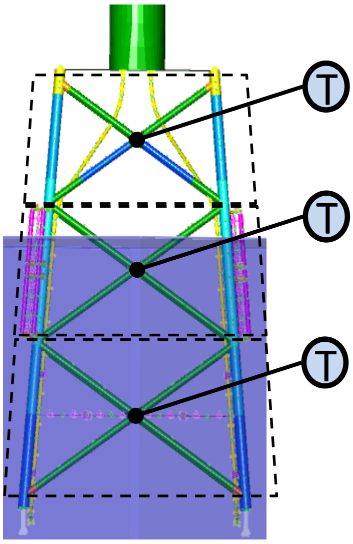


Figure 49 - Vibration results of interest

Vibrations of interest:

- Frequency: 2-5 Hz
- Amplitude: 0.56-5.60mm (0.1-1% of bracer diameter)

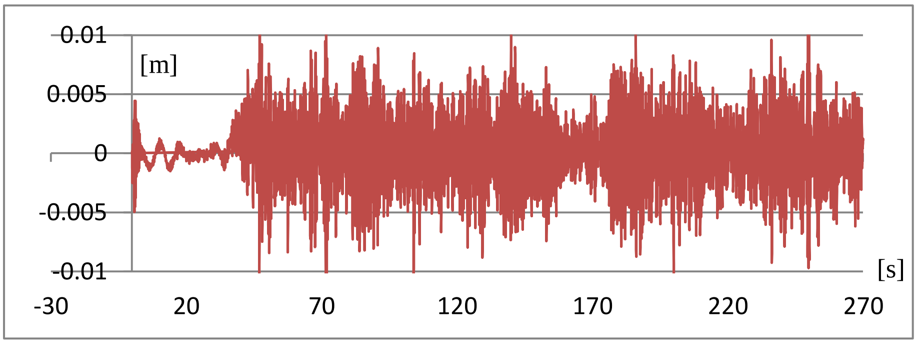


Figure 50 - Lower Junction - Out of plane displacement from test analysis

5.7.2 Bracer vibrations and signal processing

The signal (out of plane bracer vibrations) achieved from the simulation events had to be processed to identify frequency and magnitude in order to evaluate if they were of relevant vibrations.

A frequency spectrum analysis was performed using FFT (Fast Fourier Transform). Excel was first tried, since several scripts made during the project already was adapted to Excel. However, Matlab was used instead since Excel has a limitation regarding the maximum number of data points that can be used during the FFT. The FFT has the advantage of showing a signal in the frequency domain, as the original signal is shown in the time domain.

The next chapters will describe the technique used.

FFT

When using the FFT capability in Excel, the signal has to contain a certain number of datapoints. The number of datapoints has to be a power of 2, and cannot contain more than 4096 datapoints.

The number of datapoints in the signal obtained during analysis varied between the analysis methods, since the resolution differed.

A signal from the fully coupled analysis, with a time increment of 0.02 in a simulation event of 270 seconds, will give 13500 datapoints. The sampling frequency is:

0.02^{-1} = 50 Hz. The sampling frequency is twice the Nyqvist frequency.

The Nyqvist frequency describes the highest frequency that can be represented using an FFT. The Nyqvist frequency is half the sampling frequency. Using the whole frequency range (50Hz) will show a fictional mirror image of the frequency spectrum at the far end of the frequency scale. This shows the importance of having a sufficient sampling frequency in order to describe the frequency of interest.

Another effect of using FFT is aliasing. This is a phenomenon that will make vibration energy at high frequencies show up at lower ranges. This is not considered in this case.

Leakage is also a known issue regarding the use of FFT, and was a particular problem during the interpretation of the vibration signal, see Figure 51 - FFT Sample Mag.

For more details about FFT leakage, see reference (Cimbala, 2010).

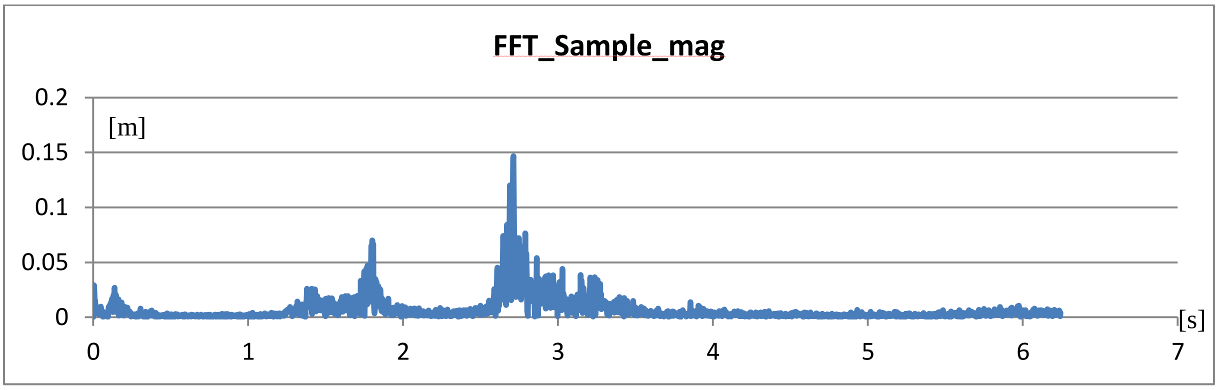


Figure 51 - FFT Sample Mag

This leakage is thought to be due to tree main reasons, mentioned below:

1) The signal (13500 datapoints) had to be truncated with a factor of 4.

   This was done using the sampling function in excel. The signal was re-sampled to the number which was equal to a power of 2 containing 4096 datapoints or less. The signal was re-sampled to 3375 datapoints, which is not a power of two. The technique then used is called ”padding”. The time axis continued beyond 3375 datapoints until 4096 datapoints where reached, whilst the amplitude was equal to zero. For padded sampled signal, see *Figure 52 - Sampled Signal*.

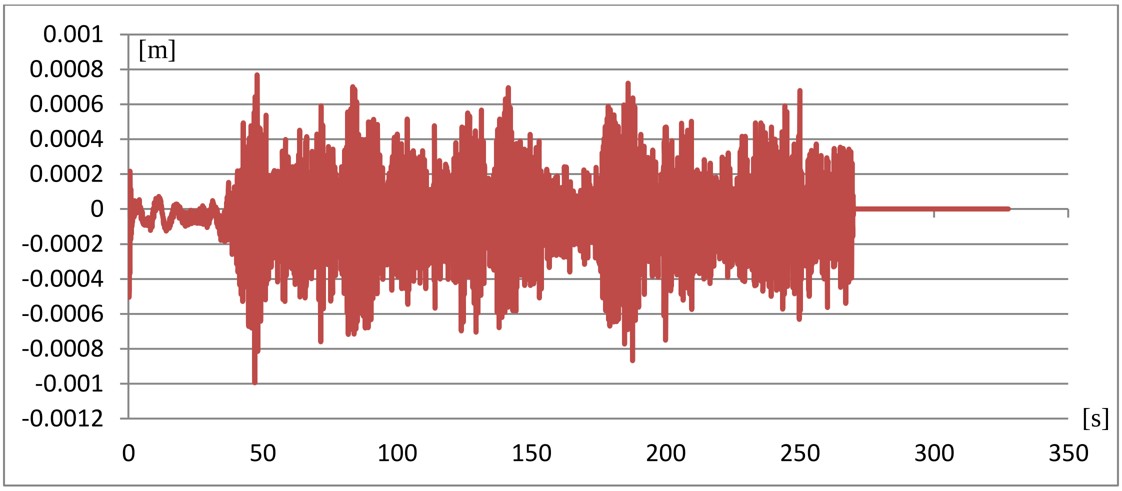


Figure 52 - Sampled Signal

The re-sampling caused informational deterioration of the signal. This is shown in the *Figure 53 - Signal deterioration*.

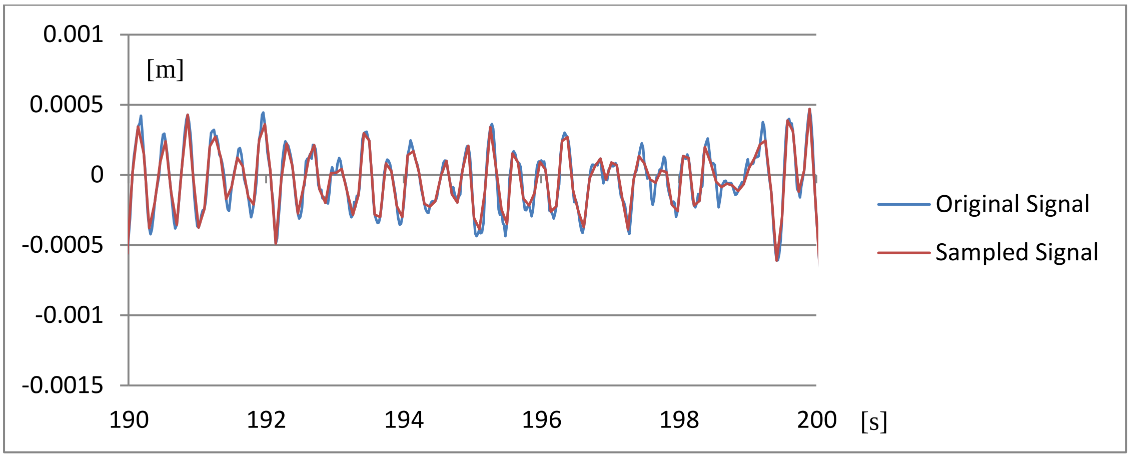


Figure 53 - Signal deterioration

2) The signal is not periodic (repeating itself periodically)
3) The signal is not always symmetric to the x-axis (time axis)
4) The signal has variable amplitude.

FFT – Matlab tool

Since MS Excel gave questionable results, a code for Matlab was written to process the signals. The Matlab code is described in the appendix, chapter 11.27.

The first step was to process the raw displacement histories for N1-N5 in Excel, as explained in chapter 5.7.1. The displacement history for N5' was exported as "*.IFFT" file. The Matlab code calculates the FFT for all files with extension ".IFFT" and extracting the results as "*.OFFT" files. The file is then imported back to Excel for result processing.

Since the displacement histories don't include information about the time history, it is important to specify the increment size (Ts) in the Matlab code. The first step for FFT calculation is to specify the signal parameters.

"Fs" is the frequency range, and is described with function 5.7.2.1:

$$fs = \frac{1}{Ts} \qquad \text{eq. (5.7.2.1)}$$

"N" is the number of increments in the signal, where Tmax is the signal time range:

$$N = \frac{Tmax}{Ts} \qquad \text{eq. (5.7.2.2)}$$

The frequency vector [f] is calculated as:

$$[f] = [-\frac{fs}{2} : \frac{fs}{N-1} : \frac{fs}{2}] \qquad \text{eq. (5.7.2.3)}$$

The code then calculates the FFT for the input signal. The output is written as imaginary numbers. The FFT is converted by calculating the "abs" for the signal. The result is as *Figure 54 - "abs" values of FFT calculation*. Mark that in this case is the matrix "f" calculated as:

$$[f] = [0 : \frac{fs}{N-1} : fs] \qquad \text{eq. (5.7.2.4)}$$

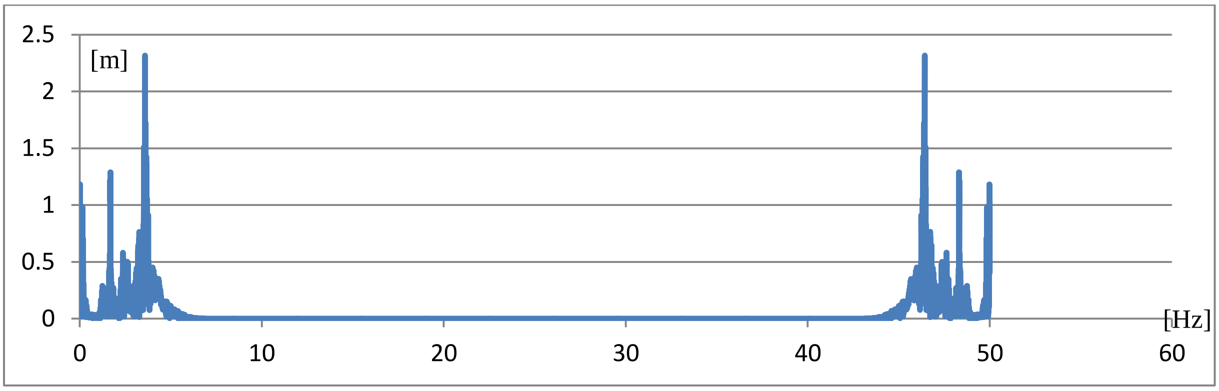


Figure 54 - "abs" values of FFT calculation

One of the consequences of the mathematics behind the FFT calculation is that the signal is “mirrored” about half the Nyqvist frequency. For visualization the graph is shifted about 0. The result is as *Figure 55 - "abs" value of FFT calculation shifted about 0*. Mark that in this case the matrix “f” is calculated as function 5.7.2.3.

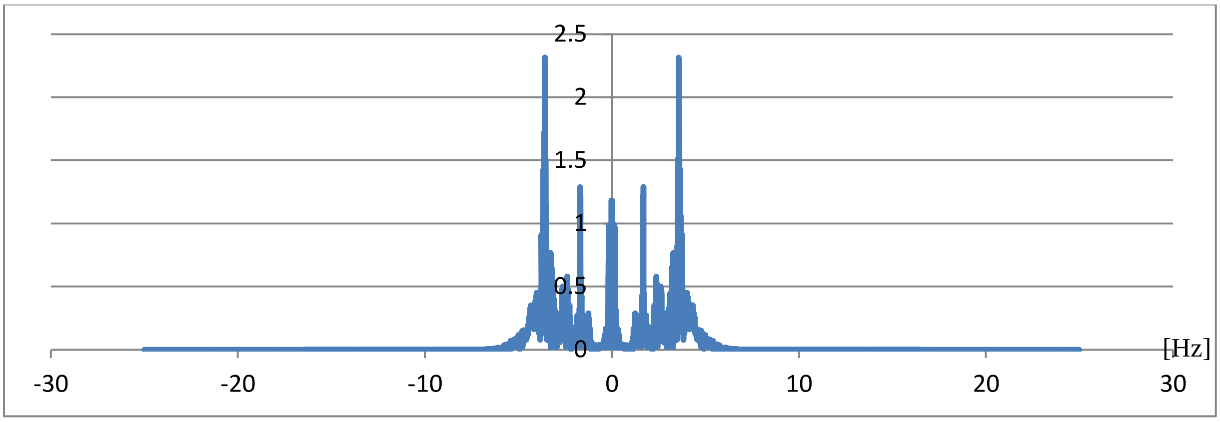


Figure 55 - "abs" value of FFT calculation shifted about 0

The consequence of the leakage as described in chapter 5.7.2 is the magnitude error and the broadening effect. If FFT calculation *Figure 55 - "abs" value of FFT calculation shifted about 0* is compared with the input signal *Figure 50 - Lower Junction - Out of plane displacement from test analysis*, the magnitude is not comparable. To deal with this the FFT signal is first scaled to 1 by dividing its values by its own maximal value:

$$[FFT(1)] = \frac{[FFT]}{\max[FFT]} \qquad \text{eq. (5.7.2.5)}$$

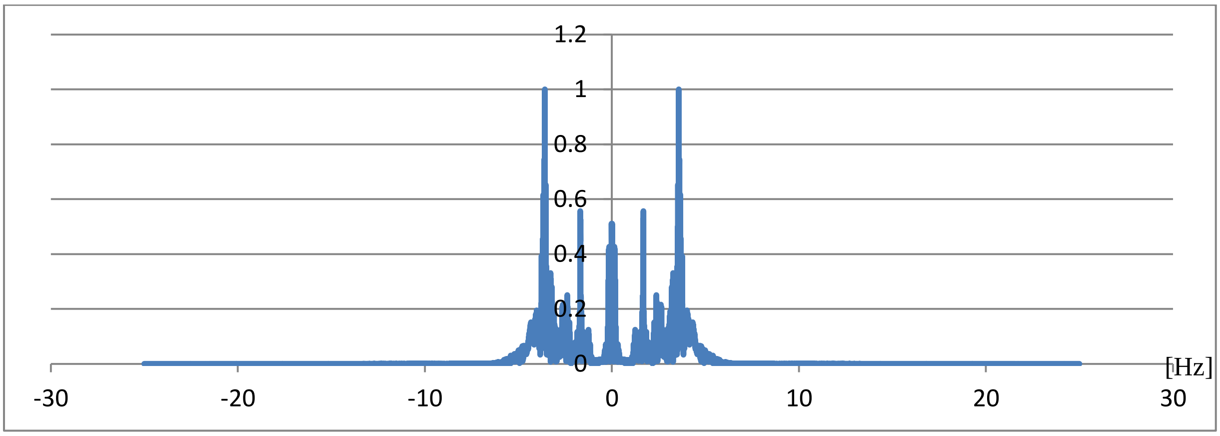


Figure 56 - FFT calculation scaled to 1

The signal is finally scaled to the input signal amplitude in best manner found. The signal is not clearly symmetric, so the average value of maximum and absolute value of minimum is used as basic for scaling of the signal.

$$[FFT(Amp - scaled)] = [FFT(1)] * (max - \frac{max-|min|}{2}) \qquad \text{eq. (5.7.2.6)}$$

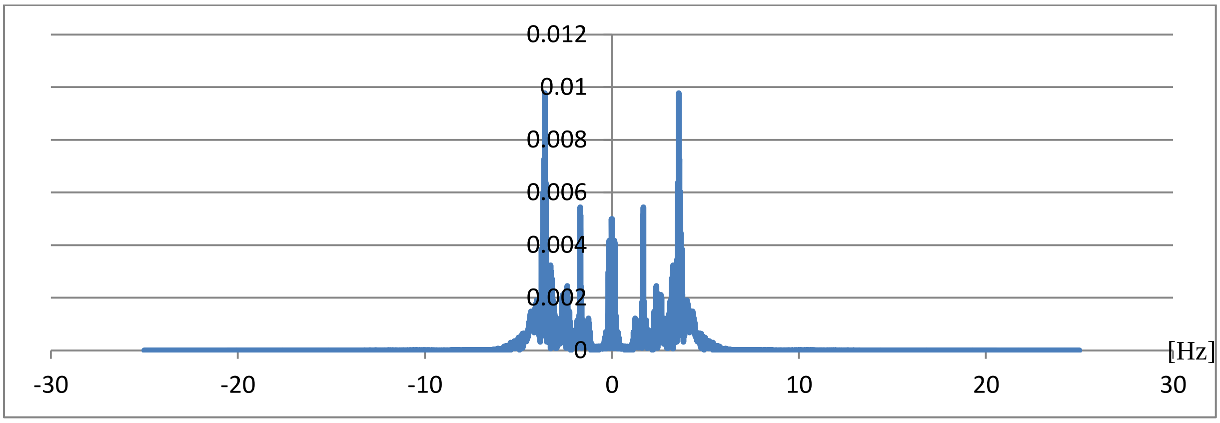


Figure 57 - FFT calculated scaled to input signal

5.7.3 Damage parameter

In order to evaluate the different analysis methods, a fatigue damage parameter was cosen as a parameter for comparison. Same method was used for both sequentially- and fully coupled analysis.

Generating stress results

In order to compute the fatigue damage parameter, the beam end stresses was computed for beam ends shown on the *Figure 58 - Damage parameter - Beam definition* using classical theory for beam crossections. Crossectional properties shown in *Table 33 - Cross section* and *Table 34 – Beam cross section parameters*.

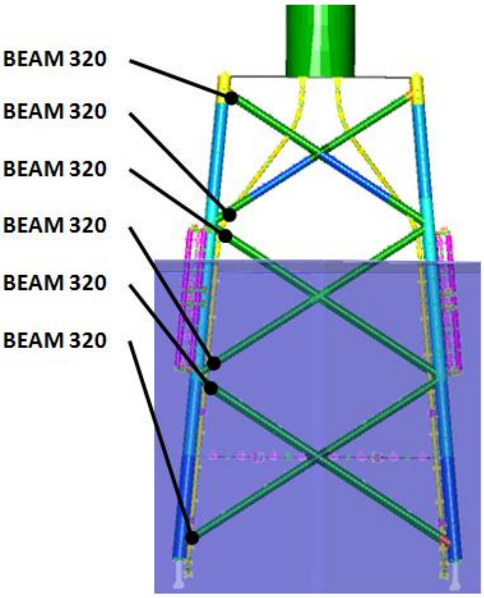


Figure 58 - Damage parameter - Beam definition

Table 33 - Cross section

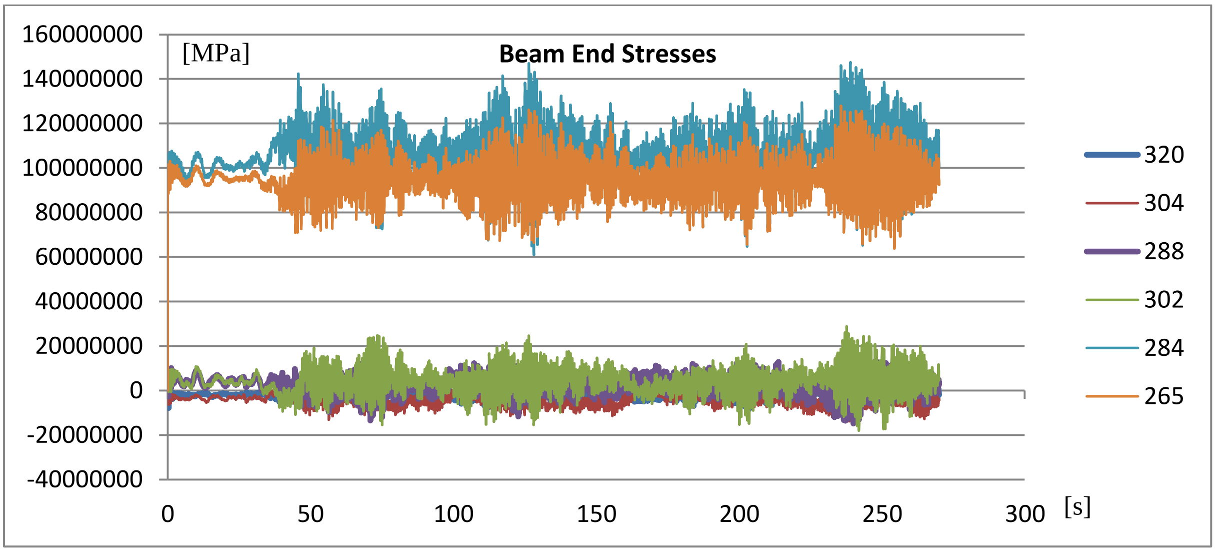
Beam #	Cross section #
320	93
304	6
302	6
288	6
284	6
265	94

Table 34 – Beam cross section parameters

	Beam Cross section		
	93	6	94
Area	0.033866369	0.01893752	0.03386637
I	0.001231555	0.00071116	0.00123155
Y_{max}	0.2795	0.2795	0.2795

From calculations, the stress-time histories was generated from each beam end of interest.

$$\sigma_{total} = \sigma_{axial} + \sigma_{bending} = \frac{F_a}{A} + \frac{M_b}{I * y_{max}} \qquad \text{eq. (5.7.3.1)}$$



Figur 1 Calculated beam end stresses

Simplifications:

The stress was calculated using axial forces and bending moments at the beam ends connected to the jacket legs. This method is a simplification and does not consider the real stresses or the real stress consentration factors or welds that needs to be considered in a real design. These factors are important, but as for this project, where analysis results from two analysis methods are to be compared, it is assumend that these simplifications have no effect as long as the method is equal in both methods.

When calculating fatigue damage, no stress concentration factor or thickness exponents are added due to the same reason as above motioned simplification.

5.7.4 Rainflow Counting

In order to be able to calculate the accumulated damage for each beam end based on the calculated stress, a rainflow counting was done on the stress-time signal. By doing this, the signal is sorted by generating numbers of amplitudes within the same range, allowing calculating the damage afterwards using the Palmgren-Miner rule. The process of performing a rainflow counting is described in the following text.

5.7.5 Rainflow Counting (RFC) Tool in Matlab

To execute the Rainflow Counting (RFC) for the stress time series, the software Matlab was used. The script written to execute the RFC operation is described in the appendix, chapter 11.28.

The first step was to process the output force histories for the bracers in Excel, as explained in chapter 5.7.1 into stress-time histories. The stress histories for the bracers were exported as "*.IRFC" file. The Matlab code calculates the RFC for all files with extension ".IRFC" and extracting the results as "*.ORFC" files. The file is then imported back to Excel for result processing.

RFC is a quite complex operation to simulate in matlab. Of that reason we used a rainflow counting file package written by Adam Nieslony. His file package may be downloaded from MatlabCentral, reference (Matlab Rainflow counting by Adam Nieslony).

Nieslony's file package mainly consists of two operations. The first operation is to convert, or filter, the input signal to only contain the turning points. To illustrate this operation the function 5.7.5.1 is plotted in *Figure 59 - Converting input signal to turning points*. The original signal contains 315 points, while the conveted only contain 5 points in the turn points of the graph:

$$f(x) = x^2 * \sin(x^2)\,, \quad where\ 0 < x < \pi \qquad \text{eq. (5.7.5.1)}$$

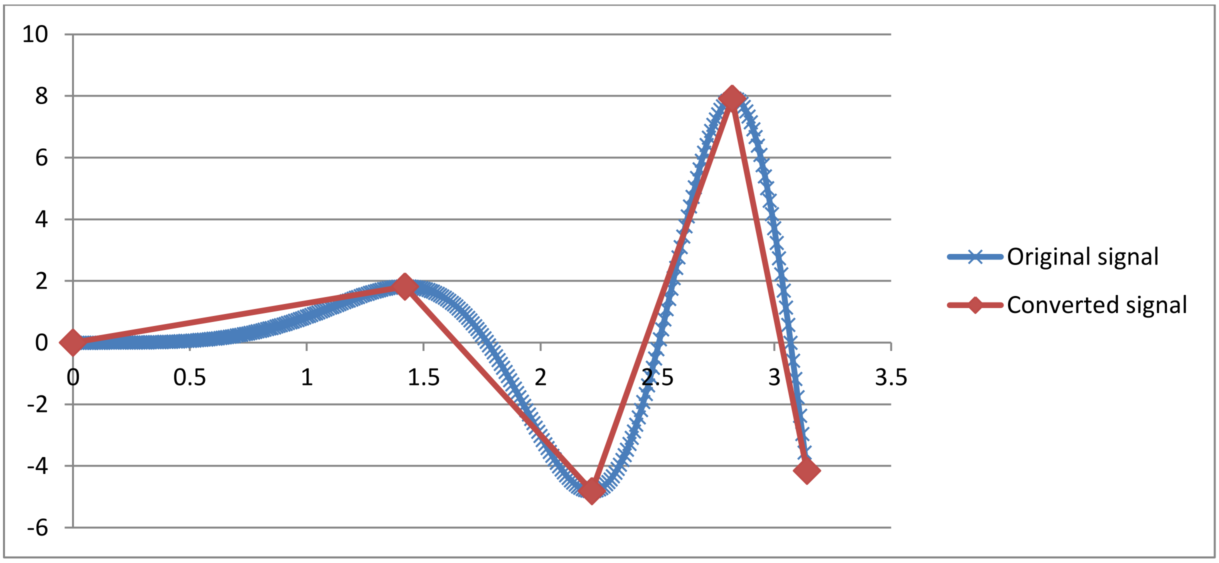


Figure 59 - Converting input signal to turning points

The second operation is the rainflow counting. The RFC method is according to ASTM, see appendix chapter 11.29. The rainflow counting script we used in based on the rainflow counting method illustrated in *Figure 60 - Rainflow counting, ref* .

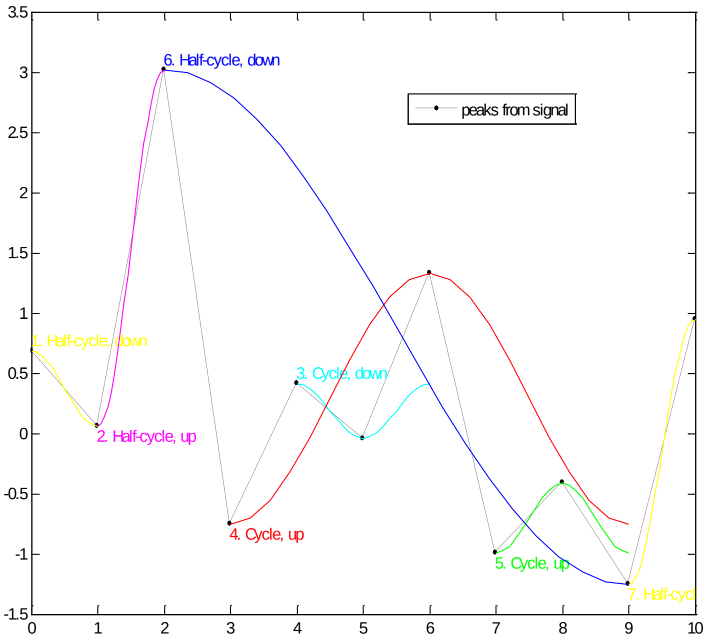


Figure 60 - Rainflow counting, ref (Matlab Rainflow counting by Adam Nieslony)

5.7.6 Calculating damage parameter

The damage parameter was calculated using the Palmgren-Miner rule on the signal generated by the rainflow counting, in combination with S-N curves retrieved from DNV-RP-C203. Mean stress correction on the stress amplitudes was performed using the *Hempel-Morrow* relation for substituting mean stress with a completely reversed amplitude giving the same life as the original stress amplitude including the mean stress. In the calculated stress, the mean stress was variable. A simplification was made, using the average mean stress for each stress-time histories when calculating the stress range when determining the damage.

Determining the fully reversed stress amplitude (σ_{ar}) giving equal life as the stress amplitude (σ_a) including mean stress (σ_m) using the Hempel-Morrow relation:

$$\sigma_{ar} = \frac{\sigma_a}{1-\frac{\sigma_m}{\sigma'_f}} \qquad \text{eq. (5.7.6.1)}$$

The material constant σ'_f is obtained from Table 9.1, mechanical Behaviour of metals, reference (Dowling).

$$\sigma'_f = 1754\ MPa \qquad \text{eq. (5.7.6.2)}$$

Determining numbers of cycles (life) of each block cycle from the rainflow counting:

$$logN = log|a| - m\log\Delta\sigma \qquad \text{eq. (5.7.6.3)}$$

Solving for N:

$$N = 10^{(\log|a| - m\log\Delta\sigma)} \qquad \text{eq. (5.7.6.4)}$$

Where stress range:

$$\Delta\sigma = 2\sigma_{ar} \qquad \text{eq. (5.7.6.5)}$$

The S-N Curve for the damage calculations is illustrated as *Figure 61 – S-N Curve (DNV RP-C203) -2.4.5 S-N curves in seawater with cathodic protection, Curve C2, no thickness exponent*.

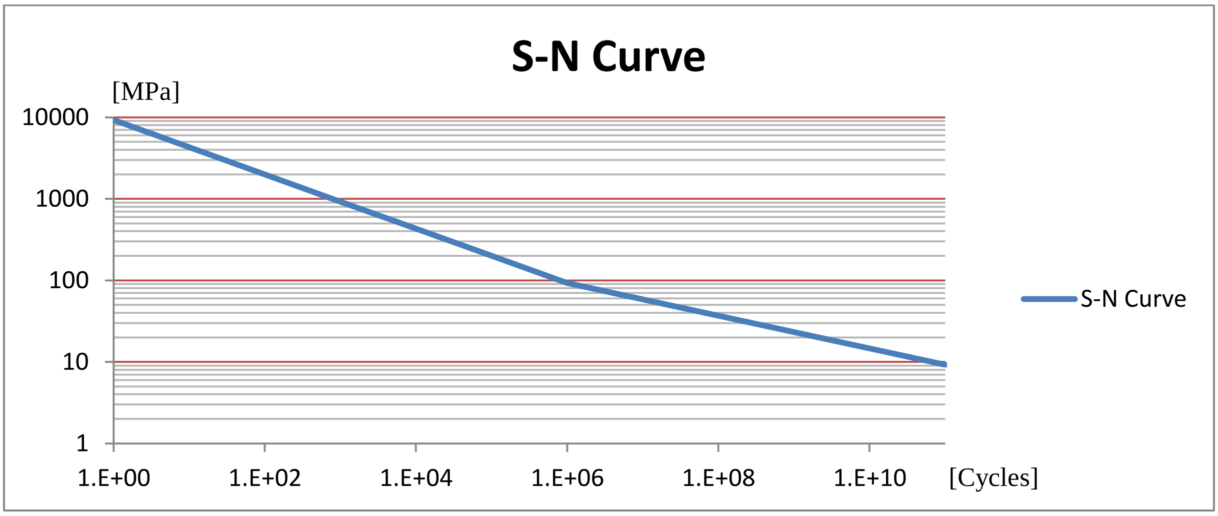


Figure 61 – S-N Curve (DNV RP-C203) -2.4.5 S-N curves in seawater with cathodic protection, Curve C2, no thickness exponent

S-N curve data from DNV RP-C203:

Table 35 - S-N Curve parameters - DNV RP-C203

cycles	Coefficient	Coefficient
N<10e6	m1 = 3	log a1 = 11.901
N>10e6	m2 = 5	log a2 = 15.835

Acumulated damage is calculated by finding the ratio between number of cycles (n_i) and the life (N_i) at the corresponding stress range ($\Delta\sigma$):

$$\sum_{i=1}^{N} \frac{n_i}{N_i} = 1, \; D_i = \frac{n_i}{N_i} \qquad \text{eq. (5.7.6.6)}$$

The sum of damage (total accumulated damage) is the sum of the damage made from each block cycle defined by the rainflow analysis.

The results from the fully coupled analysis in FEDEM are illustrated in *Figure 62 - Results from a Fully Coupled analysis in FEDEM. Accumulated damage and bracer vibrational amplitudes*.

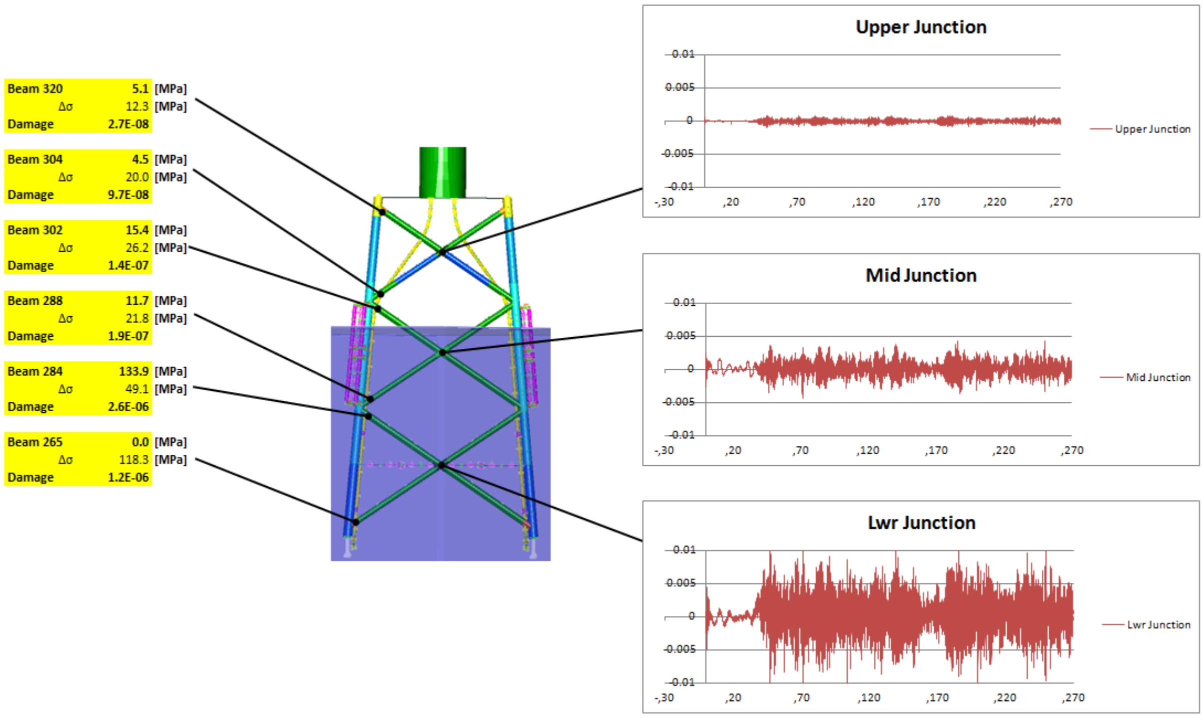


Figure 62 - Results from a Fully Coupled analysis in FEDEM. Accumulated damage and bracer vibrational amplitudes

6 Analysis results

The analyses are mainly based on the fully coupled analysis method in FEDEM and the sequentially coupled analysis method using a combination of USFOS and FEDEM. Results are extracted from similar checkpoints in both analyses, and compared using signal processors as Fast Fourier Transformation (FFT) and Rainflow Counting (RFC). The abbreviations SCA (Sequentially Coupled Analysis) and FCA (Fully Coupled Analysis) is used in this chapter.

In this chapter a selection of the results from the analyses is represented. For the full description and results for the analyses refer to appendix chapter 11.2-11.25. The appendix is divided into five parts per analysis. Part 1 – Description – describes the kind of analysis that is run and how it is set up. Part 2 – Summary – lists the maximum and minimum stress and damage for each K-Joint. Part 3 – Bracer displacement – includes the displacement charts of upper, middle and lower junction. Part 4 – Bracer FFT Analysis – includes the FFT analyses of the displacement of the junctions. The last, part 5 – Beam stresses – both include a chart of the K-Joint stresses and a table of the max, min and delta stresses for the joints.

The first analysis, chapter 6.1, is an Eigen modal analysis of the jacket foundation with tower, nacelle and piles. It is important in order to set focus on in which natural frequencies the bracers operate in.

The chapter 6.2 addresses the fully- and sequentially coupled analysis methods. All three load history events for the two methods are compared to each other. The comparison is divided in two parts, one for fatigue damage using rainflow, and the second for the FFT analysis to analyse the signal frequencies.

The comparison between fully- and sequentially coupled analysis showed significant differences. The chapter 6.3 and 6.4 was considered especially to find out what parameters led to the major difference. For chapter 6.3 the displacement history for the fully- and sequentially coupled analysis in the jacket interface node was examined. In this way the validity of the first part of the sequentially coupled analysis could be specified. In chapter 6.4 the displacement history of the interface node in the fully coupled analysis was transferred to the interface node in the stress recovery analysis of the sequentially coupled analysis. The changes in the methods of the last part could thus be compared.

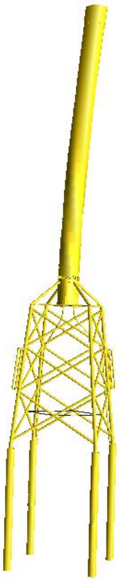
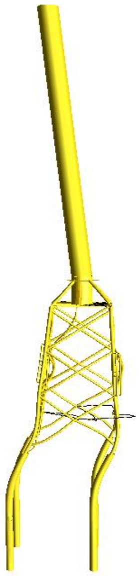
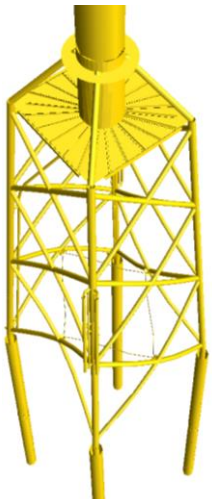
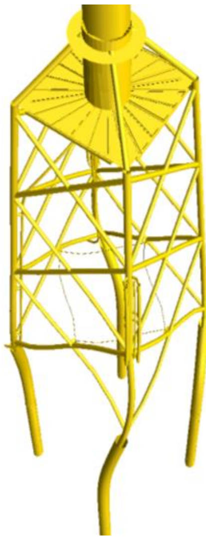
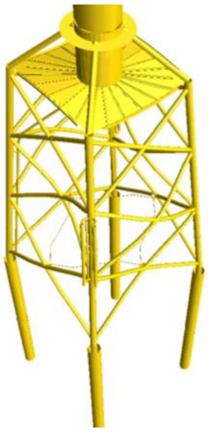
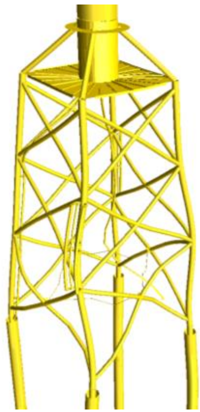
There were some uncertainties of the similarity between FEDEM and USFOS in terms of wave load on the jacket foundation. To be aware of this difference a jacket foundation model only exposed of wave load was executed in both software’s, and is provided in chapter 6.5.

In chapter 6.7, the sensitivity of the mass and damping matrices was examined for the sequentially coupled analysis. It was desired to examine if the matrices would have any impact on the frequency band given by the FFT analyses.

The last, chapter 6.8, is a study where the analysis time increment is varied. This project includes both fully- and sequentially coupled analysis method. The time increment differs in the methods, and it is of interest to see if the difference has an effect on the predicted result.

6.1 Eigen modes of complete structure – USFOS

Eigen frequencies have been extracted for the structure including tower and nacelle masses. Rotor blades and their inertial masses are not included. The eigenmodes are extracted for comparison with results obtained from the dynamic analyses using both the fully- and sequentially coupled method.

 Mode 0.29Hz Global sway	 Mode 1.63Hz Global sway	 Mode 1.87Hz Lower bay bracers OOP	Mode 2.32Hz  Global Twist
 Mode 3.25Hz Mid bay bracers OOP	 Mode 3.77 Hz Mid bay bracers OOP	 Mode 6.05Hz Lower bay bracers OOP	

OOP = Out Of Plane (movement)

Table 36 - Predicted eigenfrequencies on bracers

Bracer Junction	Lower bay bracers	Middle bay bracers	Upper bay bracers
Reported Frequency	1.9Hz, 3.2Hz, 6Hz	1.9Hz, 3.2Hz, 3.8Hz, 6Hz	1.9Hz, 3.2 Hz,

6.2 Comparison of FCA versus SCA

The comparison of fully- versus sequentially coupled analysis is mainly done by comparing the estimated joint damage and bracer out of plane vibration.

6.2.1 Estimated K-Joint damage

The *Figure 64* to *Figure 66* represents the estimated damage of the K-Joints of three of the bracers. The bracers are defined by *Figure 63*.

There are three different analysis cases compared in the diagrams. The “FCA” analysis is the fully coupled analysis performed in FEDEM Windpower. The estimated damage is not expressed by the structure lifetime, but by a ratio. The fully coupled FEDEM analysis is defined as two unit factors. The “SCA_D” and “SCA_M” damage is defined by the ratio to “FCA”.

There are two sequentially coupled analyses. The analysis “SCA_D” is where the full displacement time history for the interface node is used. The analysis “SCA_M” has the z-direction replaced by a nodemass during the USFOS recovery analysis in the final run.

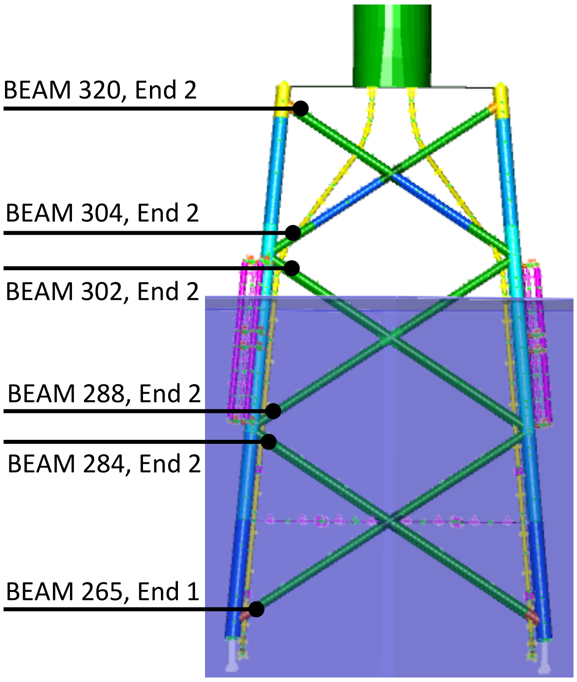


Figure 63 - K-Joint definition for Jacket foundation

Results shows very high damage accumulated for sequentially coupled analysis, much higher than expected, and several orders of magnitudes above the results from FCA. Note that the y-axis is given by exponential representation, and the results are given by a factor of 2.

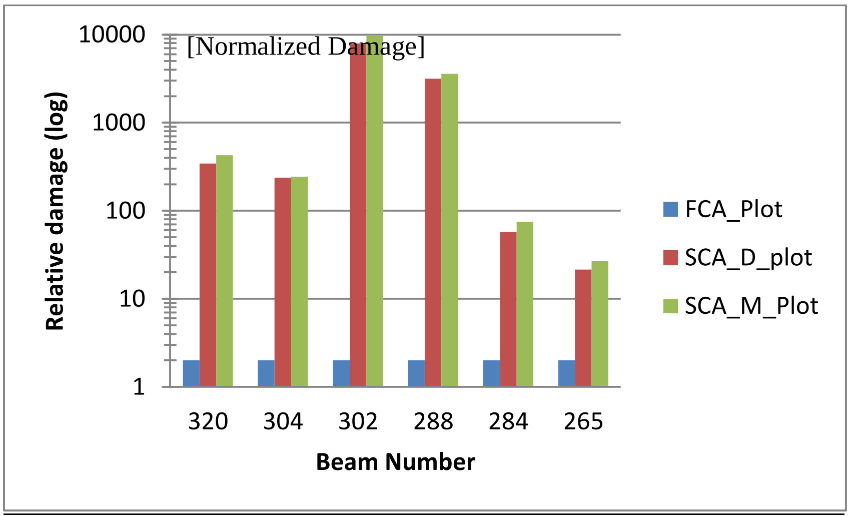


Figure 64 - FCA versus SCA damage - Event 1

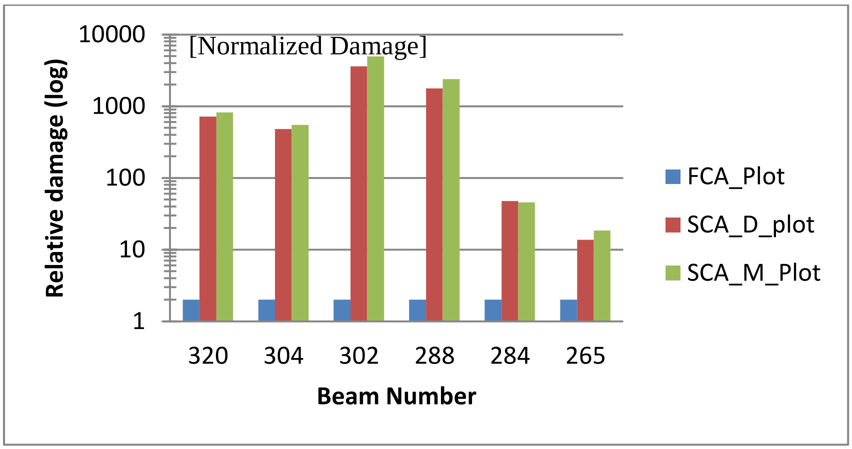


Figure 65 - FCA versus SCA damage - Event 2

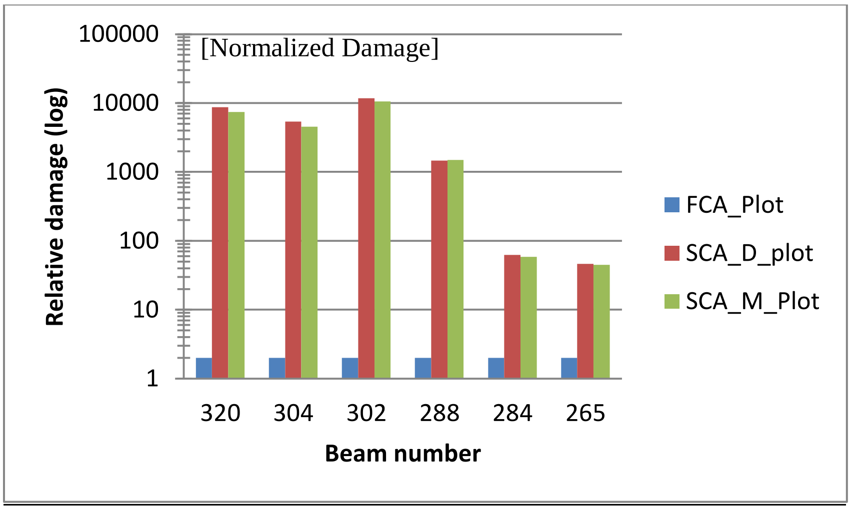


Figure 66 - FCA versus SCA damage - Event 3

6.2.2 Bracer FFT analyse – Event 1

Figure 67 to *Figure 69* represents the FFT analysis of the deflections in the centre of the upper, middle and lower jacket foundation bracer of event 1. When the results are interpreted, it is important to avoid looking to much at the frequency amplitude, as it may be misleading. When looking at the upper and middle junction, SCA have peaks at 1.75, 3, and a major at 3.9 Hz. The FCA has peaks at 1.75 and major at 2.5 Hz. At the lower junction FCA has the major at 2.6, but high energy fluctuations at a range at 3-4 Hz. The predicted frequencies from the SCA fit quite well with the predicted eigenfrequencies from USFOS.

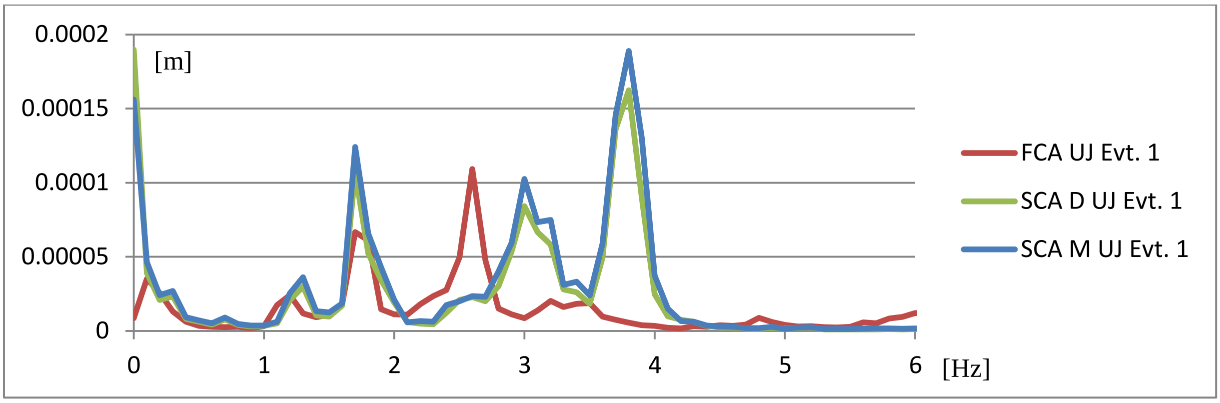


Figure 67 - Upper Junction - Event 1 - Analyse comparison

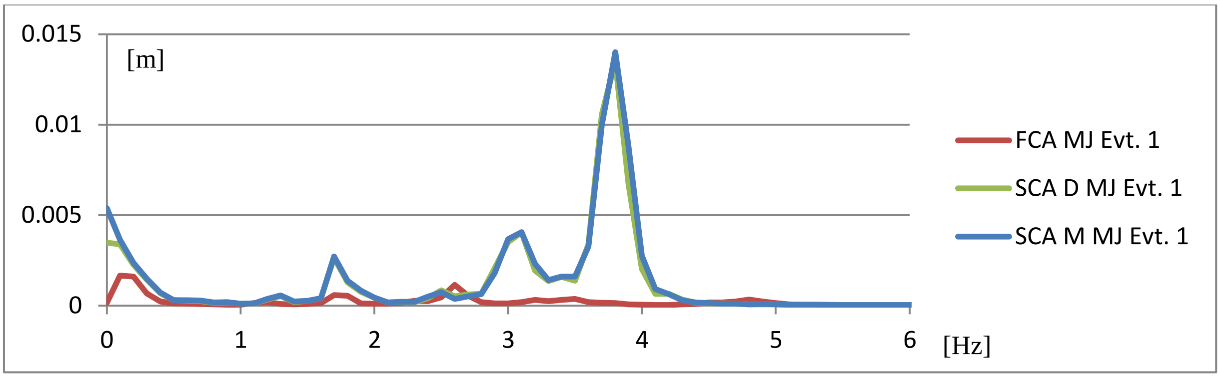


Figure 68 - Middle Junction - Event 1 - Analyse comparison

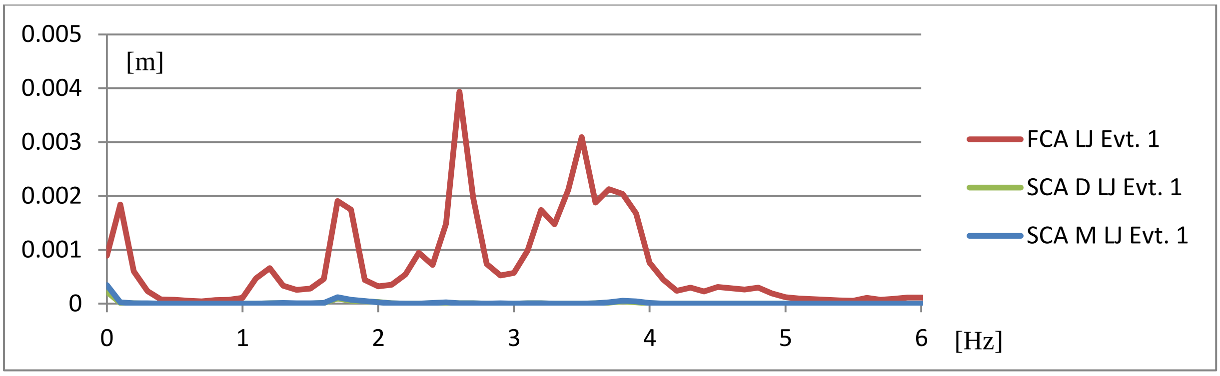


Figure 69 - Lower Junction - Event 1 - Analyse comparison

6.2.3 Bracer FFT analyse – Event 2

For event 2 the FCA has almost the same characteristics as FCA at event 1. By comparison with event 1, the frequency has shifted 4 % higher, while the amplitude is 2 to 4 times larger in value. The SCA still has the same peaks, but the curve has a lower slopes. Generally, the peaks are blunt, but wide. Also here the predicted frequencies from the SCA fit quite well with the predicted eigenfrequencies from USFOS.

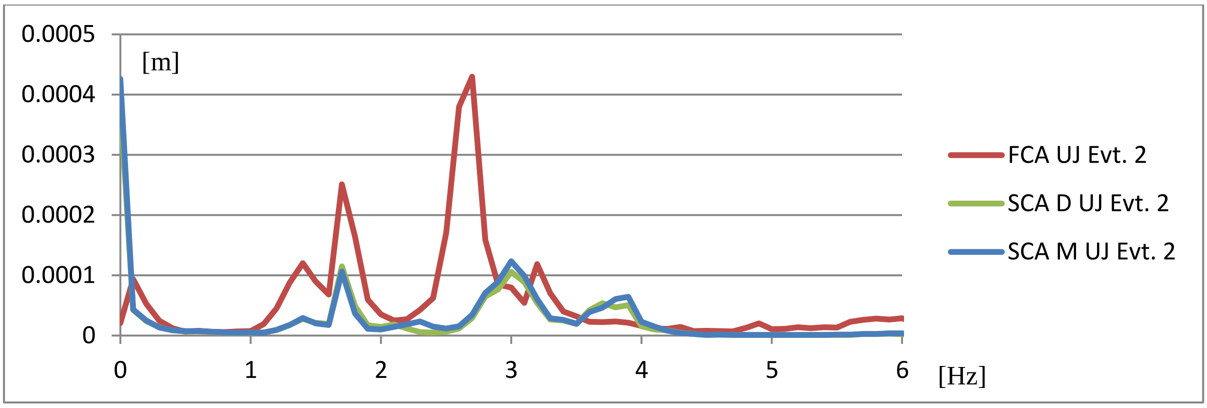


Figure 70 - Upper Junction - Event 2 - Analyse comparison

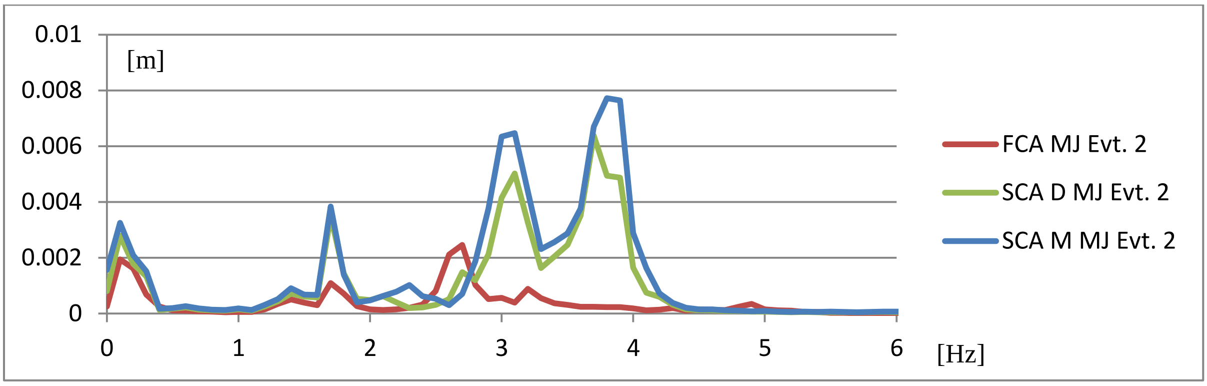


Figure 71 - Middle Junction - Event 2 - Analyse comparison

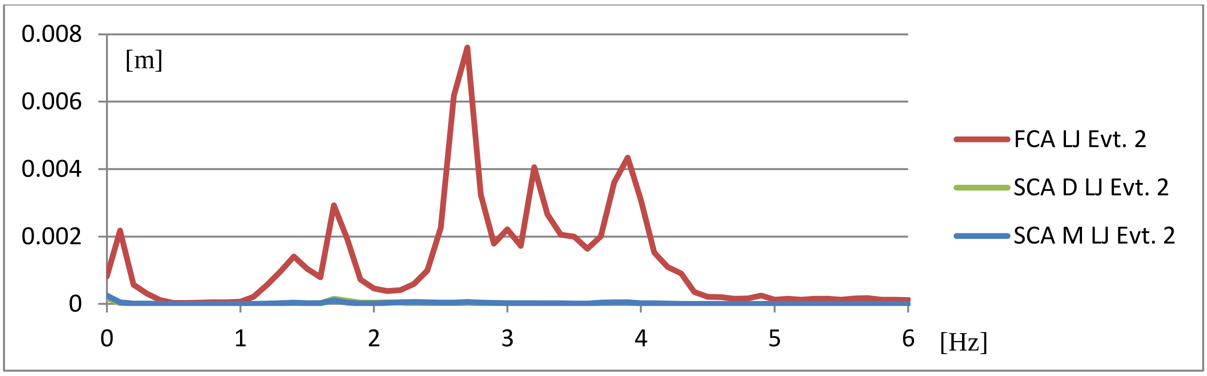


Figure 72 - Lower Junction - Event 2 - Analyse comparison

6.2.4 Bracer FFT analyse – Event 3

The characteristics for FCA are the same to event 3, as event 2 is for event 1. For SCA there were problems with buckling of the jacket foundation. Of that reason the result of the analyses should be considered as invalid. However, it should be noted that the trend we see in event 2 also applies to the results in event 3 for the SCA. The predicted frequencies from the SCA fit quite well with the predicted eigenfrequencies from USFOS.

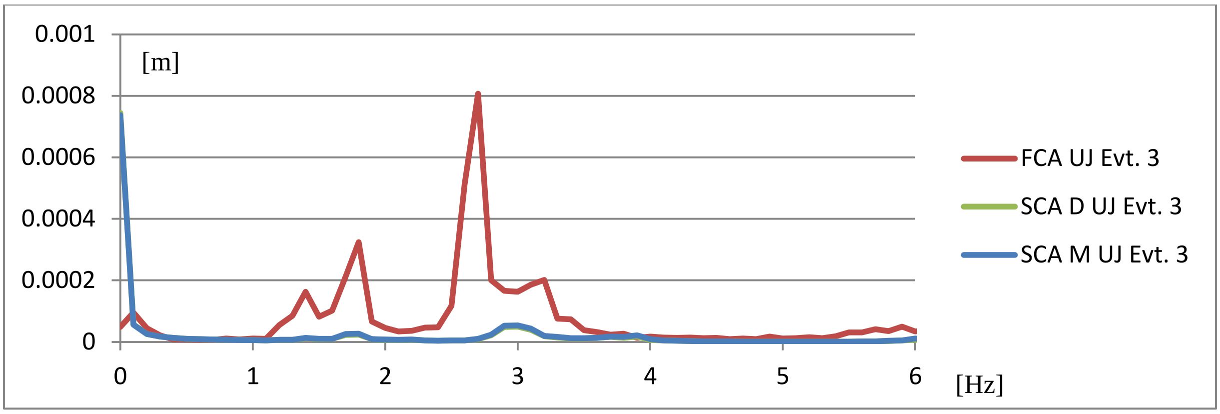


Figure 73 - Upper Junction - Event 3 - Analyse comparison

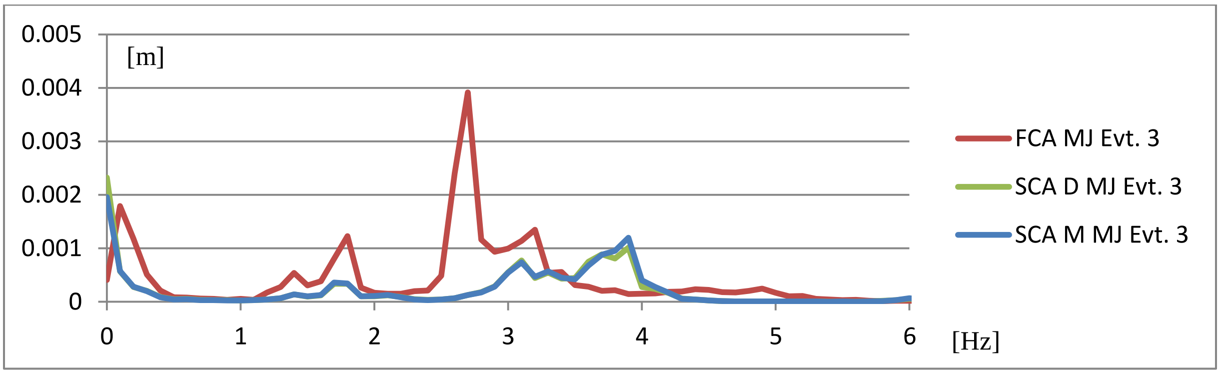


Figure 74 Middle Junction - Event 3 - Analyse comparison

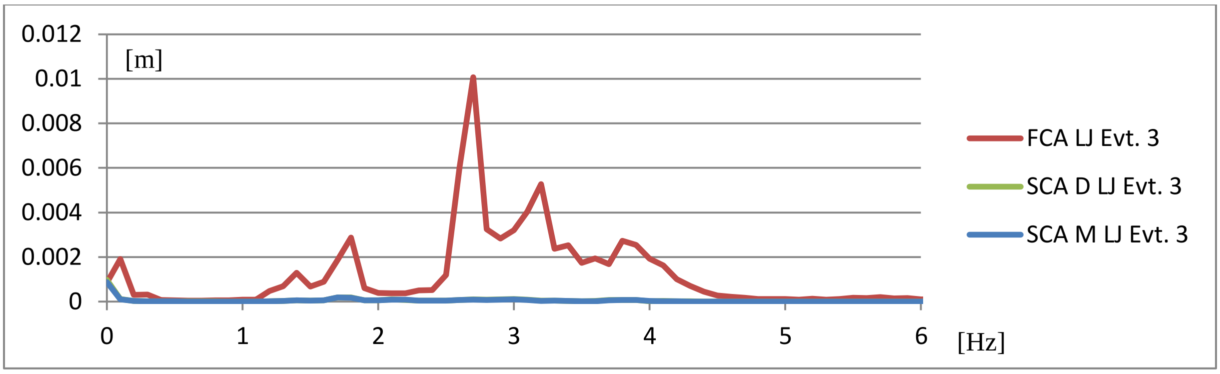


Figure 75 - Lower Junction - Event 3 - Analyse comparison

6.2.5 Debugging

Due to large differences between SCA and FCA it is necessary to find the cause. Input files, scripts and output files were checked, without finding any errors. The conclusion had to be that the cause lay within analysis methods. The very low fatigue life of the SCA compared to the FCA indicated that the cause was likely to be found within the SCA method or the SCA analysis.

The table *Table 37* lists the debugging process and the chapter 6.3 to 6.5 shows the results in detail.

Table 37 - Debugging process for SCA

Nr.	**Possible cause of error - or parameter to check**	**Result**
1.	Wrong input to FEA on SCA and FCA?	Checked - OK
2.	Wrong cross-sectional data (for beams) in excel?	Checked - OK
3.	Wrong references to formulas in excel when calculating beam stresses?	Checked - OK
4.	Errors in damage calculations in excel sheets for SCA?	Checked - OK
5.	Check output beam end forces and moments in SCA (4)	Checked – discrepancy USFOS >> FEDEM – WHY?
6.	Difference in wave loads FEDEM/USFOS?	See pt. 11
7.	K-Joints in USFOS stiffer than node-to-node joints in FEDEM?	K-Joints might be stiffer Possible influence?
8.	SCA stiffness matrix – Rotational stiffness in K matrix?	Possible influence?
9.	Run USFOS Event 1 with displacements from N8000 FEDEM FCA	If equal to FCA; then Krz must be reason for difference
10.	Run USFOS Event 1 with forces from N8000 in FEDEM FCA	
11.	Compare FEDEM & USFOS results without Windturbine (loads and weight) – Waves only	FEDEM: Waves have small impact on structural response USFOS: More impact on structure than FEDEM
12.	Rainflow counting on Signal (Disp Node 1/8000) – correlation FCA SCA	rZ – much higher in SCA than FCA (low Krz? In SCA). See pt. 9 & 11

6.3 Analysis - Interface node displacement, FCA versus SCA

To examine what makes the major difference between the FCA and SCA, the interface node displacement for the analyses for event 1 were compared.

6.3.1 Displacement comparison

Generally the amplitude for FCA and SCA is quite similar. For X-direction and Y-rotation the mean deflection for FCA is a little higher, but has the same characteristics as SCA. For Y-direction, X-rotation and Z-rotation the mean deflection is the same, but the SCA amplitude is a little higher. However, the results from the displacement comparison may hardly explain the large fatigue difference between the two analysis methods.

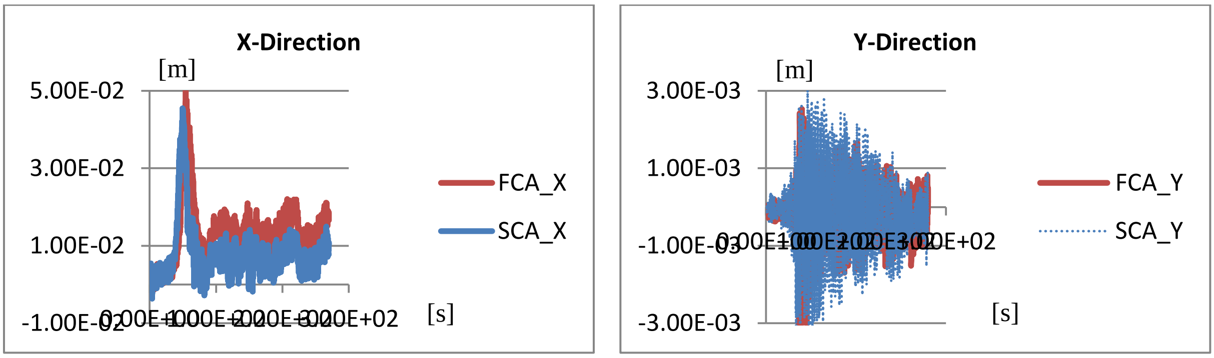


Figure 76 - Displacement of XY-direction of n1

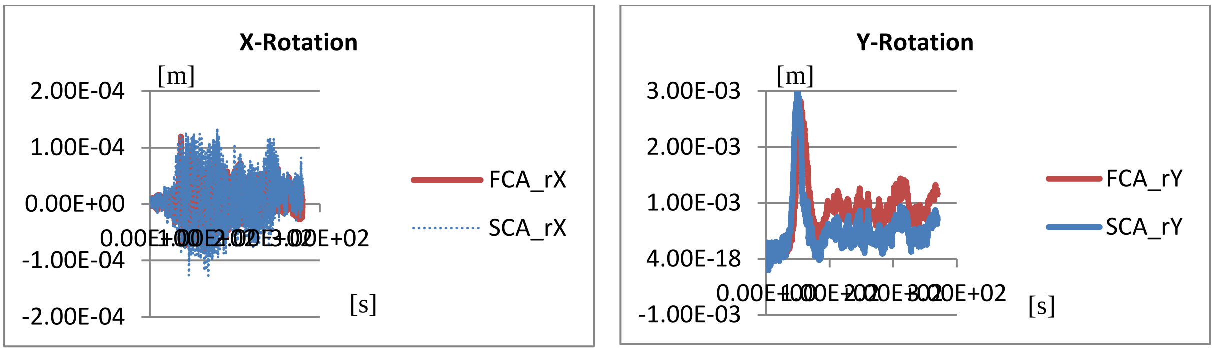


Figure 77 - Displacement of XY-rotation of n1

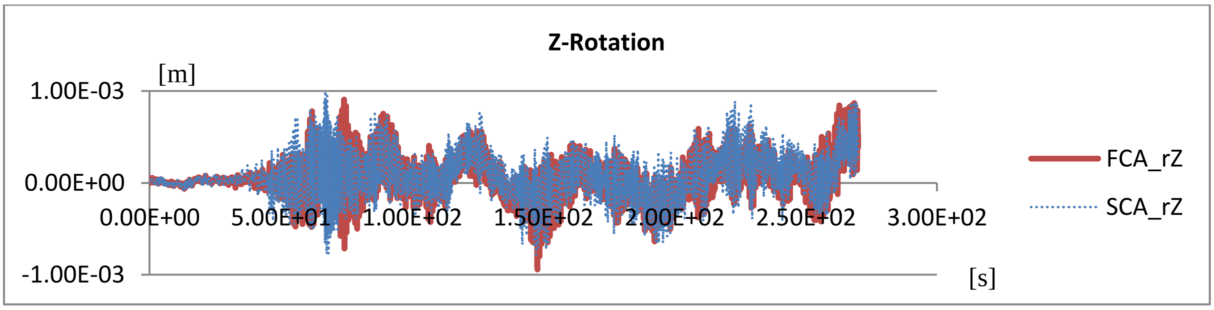


Figure 78 - Displacement of Z-rotation of n1

6.3.2 Rainflow analysis

To be able to examine the interface node displacement more closely, a rainflow counting was run. The rainflow counting was divided into 100 bins. The X-direction and X-rotation have approximately equal values for FCA versus SCA. The Y-direction has some more fluctuations for FCA at low amplitudes, while SCA had little more fluctuations than FCA for Y-rotation. For Z-rotation the SCA cannot be compared with FCA. The SCA has 10-20 times more fluctuations than FCA across the full spectrum. In figurative sense it means that the frequency of rotation about z is much higher for SCA than for FCA, and is in this case the weakest link.

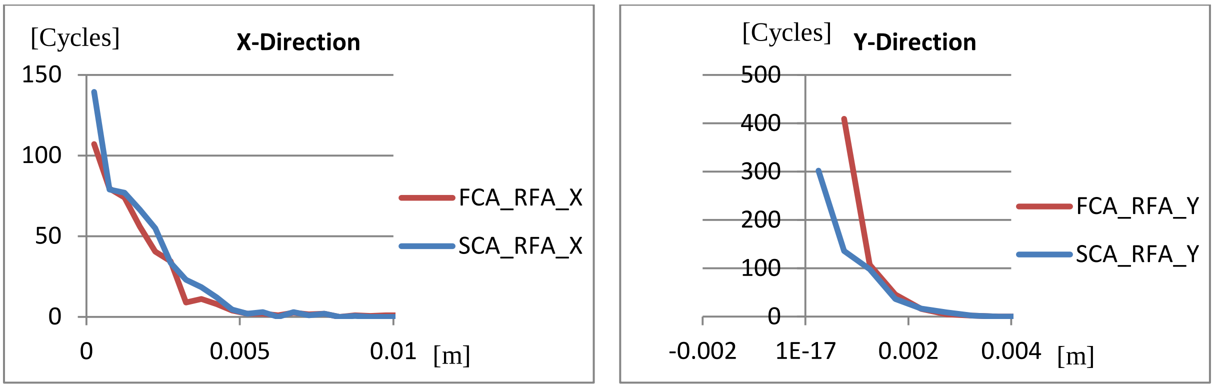


Figure 79 - RFA of XY-direction n1

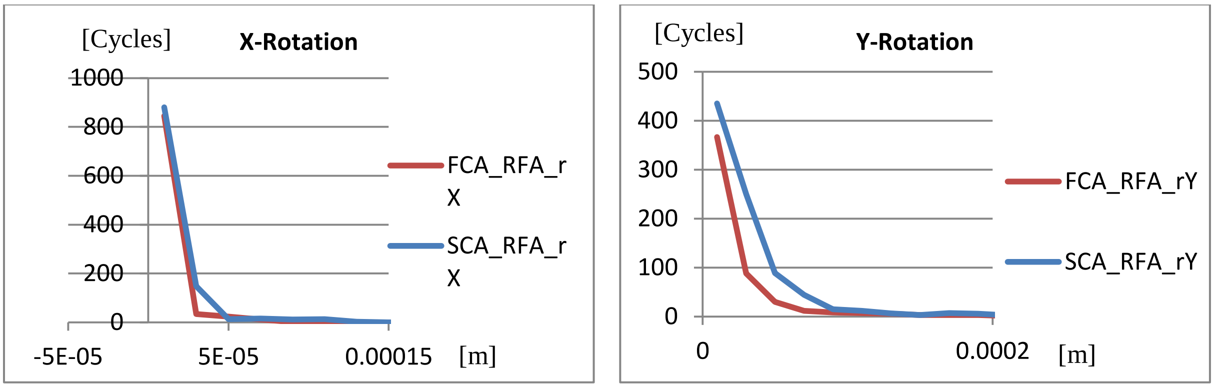


Figure 80 - RFA of XY-rotation n1

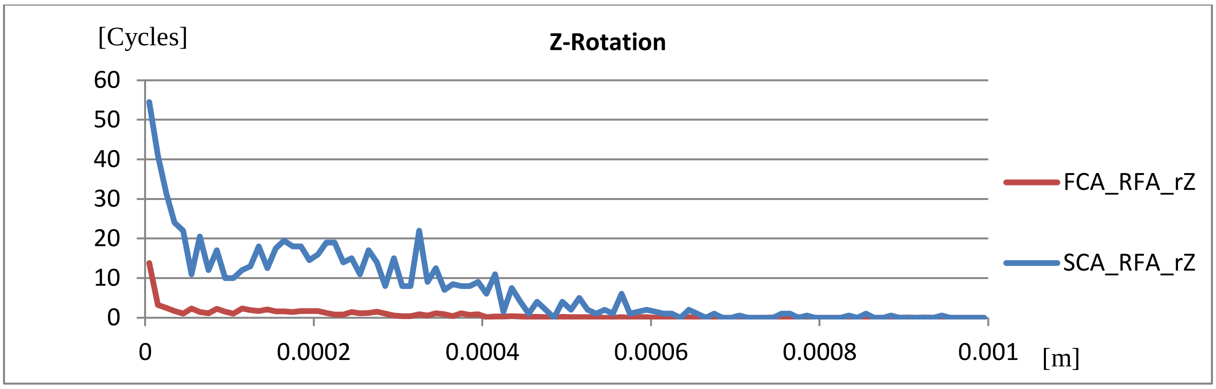


Figure 81 - RDA of Z-rotation n1

6.4 Time history from interface node in FCA used in SCA

There are many parameters which can influence the results for the sequentially coupled analysis. To be able to filter out some uncertainties of the analysis, the stress recovery step of the SCA was run by applying the interface node displacement from the fully coupled analysis. In this way the stress recovery in USFOS was tested against the FCA from FEDEM by itself. Both the estimated K-Joint damage and the FFT analysis were compared with FCA.

6.4.1 Estimated K-Joint damage

According to the fatigue calculations, this analysis gave significantly better result compared to the FCA, see chapter 6.2. In the *Figure 82* the FCA for beam 320, 304, 284, and 265 is scaled to 1, and for beam 302 and 288 FCA is scaled to 0.1. For the upper junction (beam 320 and 304) the SCA have 2 times higher damage than for FCA. These results are more similar to previous studies. For the middle junction (beam 302 and 288) the damage by SCA is 80 and 62 times higher than for FCA. For the lower junction the SCA damage is 0.42 and 0.18 times the FCA damage.

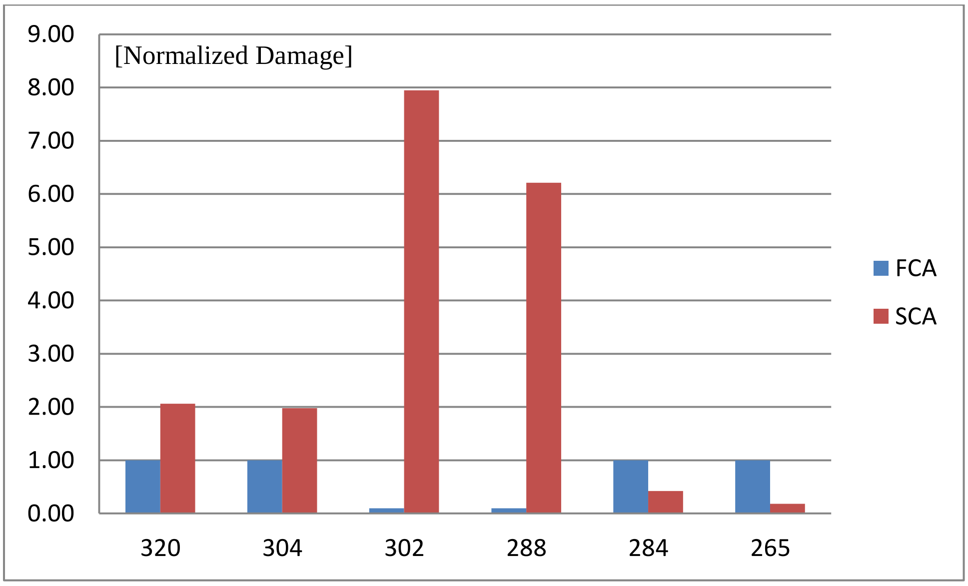


Figure 82 - FCA versus SCA with FCA n1 displacement history - Event 1

6.4.2 Bracer FFT analyse – Event 1

For the upper junction there are still major differences between FCA and SCA in terms of the frequency band where the fluctuations occur. The FCA has a significant amplitude at 2.6 Hz and at 1.7 Hz. The SCA has the highest amplitude around 3.6 and 1.8. The SCA has a fairly even distribution, and has a cut-off at 4 Hz.

For the middle junction the shape of the signal for SCA is similar to the upper junction. The differences are its more significant characteristic from 3.5 to 4 Hz and at 0.1 Hz.

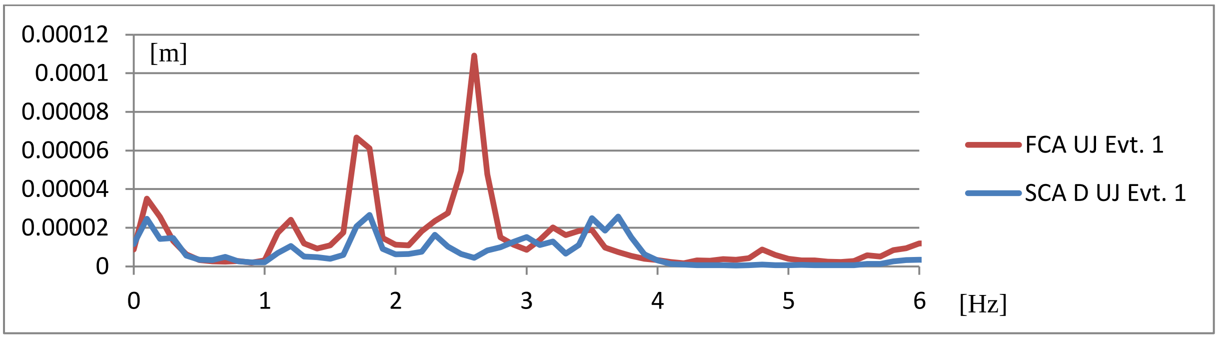


Figure 83 -Upper Junction - Event 1 – SCA with FCA n1 displacement history

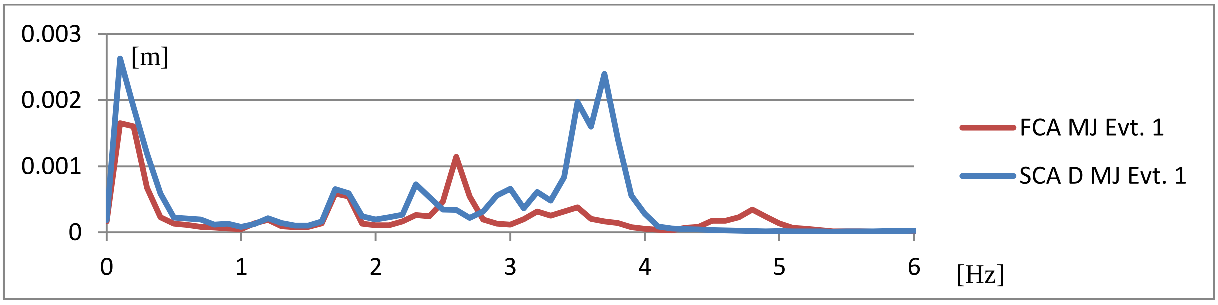


Figure 84 -Middle Junction - Event 1 – SCA with FCA n1 displacement history

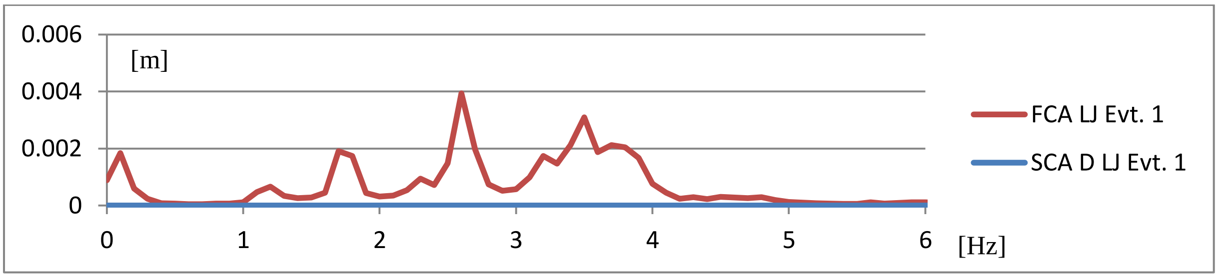


Figure 85 -Lowe Junction - Event 1 – SCA with FCA n1 displacement history

6.5 SCA by FEDEM and USFOS – Time history displacement from FCA

This analysis was executed to get an impression of the differences between the FEA model in FEDEM and USFOS. The USFOS model is used as it is, with linear soil pile springs and wave loading. For the FEDEM model the turbine and the tower was disconnected to prepare it for a sequentially coupled analysis. The displacement time history loaded in the interface node for these analyses is provided by the fully coupled analysis.

To use the fully coupled analysis displacement, then it is also possible to compare the last sequence term of SCA to FCA in the same analysis software. Both the estimated K-Joint damage and the FFT analysis were compared.

6.5.1 Estimated K-Joint damage

According to the fatigue calculations, there are variable differences between the two analysis models. For the first joint of the upper junction (beam 320) there are little differences between USFOS and FEDEM. The damage in USFOS is 0.85 times the damage in FEDEM. For the second joint USFOS has only 0.11 times FEDEM damage. For the bracer in the wave zone USFOS damage is 36 and 22 times larger. In the wire pre-tension bracer the USFOS damage is only 0.01 times the damage for FEDEM.

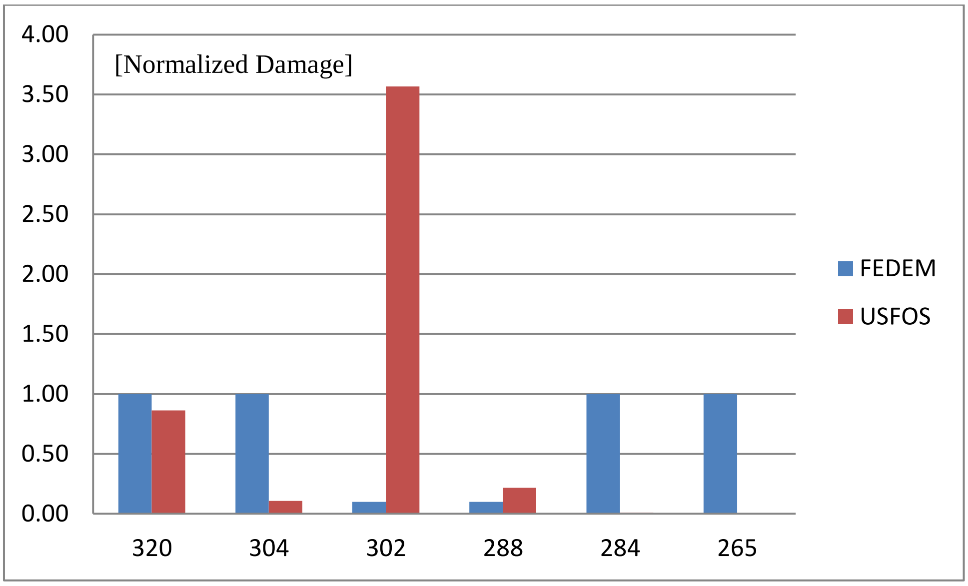


Figure 86 - SCA in FEDEM and USFOS - Time history displacement from FCA

6.5.2 Bracer FFT analyse – Event 1

For the upper junction the fluctuation frequencies of the FEDEM analysis are approximately similar as in the FCA. The major differences are that the amplitudes are higher than FCA. The fluctuation frequency at ~2.9 Hz has a wider distribution. For the middle junction the fluctuations for both analyses are located at 0.1 and 1.7 Hz. What should be noted is that the frequency range of 3.7 Hz for USFOS has shifted to 2.9 Hz for FEDEM. For the lower junction the fluctuations at 3.9 for FEDEM have been amplified in relation to FCA.

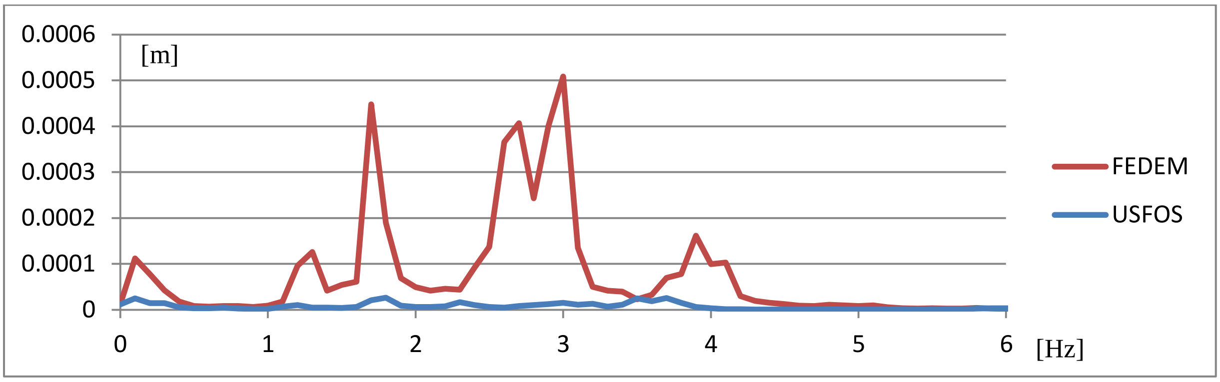


Figure 87 - Upper Junction - Event 1 - SCA with FEDEM and USFOS using FCA displacement

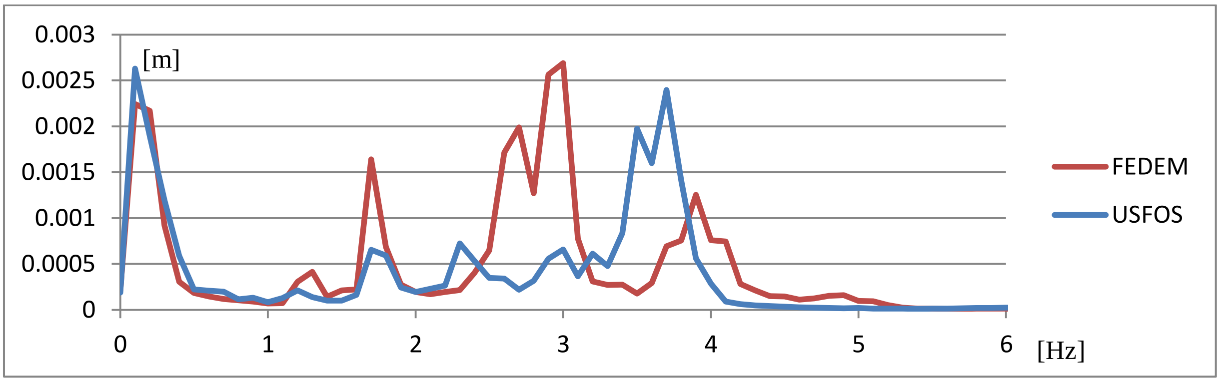


Figure 88 - Middle Junction - Event 1 - SCA with FEDEM and USFOS using FCA displacement

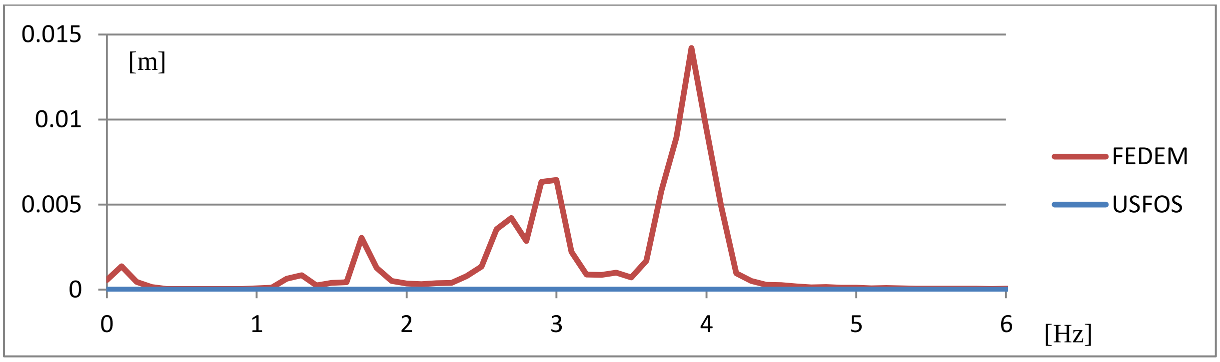


Figure 89 - Lower Junction - Event 1 - SCA with FEDEM and USFOS using FCA displacement

6.6 Significance of jacket wave load

To be able to compare FCA and SCA it was important to know the difference between the influence of wave load in USFOS and FEDEM. In this analysis the jacket foundation has undergone the wave load alone in both FEA cedes.

6.6.1 Estimated K-Joint damage

The damage in the bracer is significantly higher in USFOS (SCA) than in FEDEM (FCA). For the upper junction it is 10 and 30 times higher for SCA. For the middle junction, 139 and 180 times higher. For the lower junction, 3 and 1 times higher.

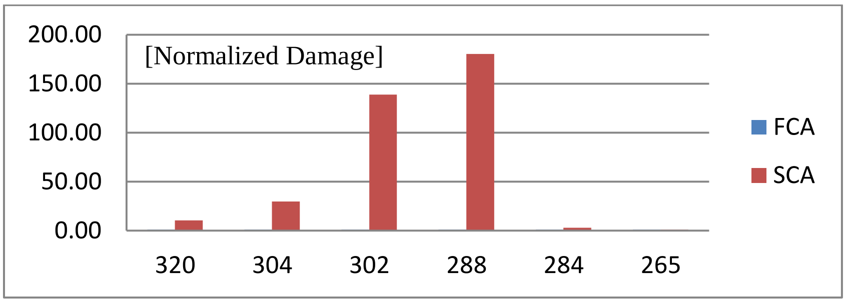


Figure 90 - FCA versus SCA wave load damage - Event 1

6.6.2 Bracer FFT analyse – Event 1

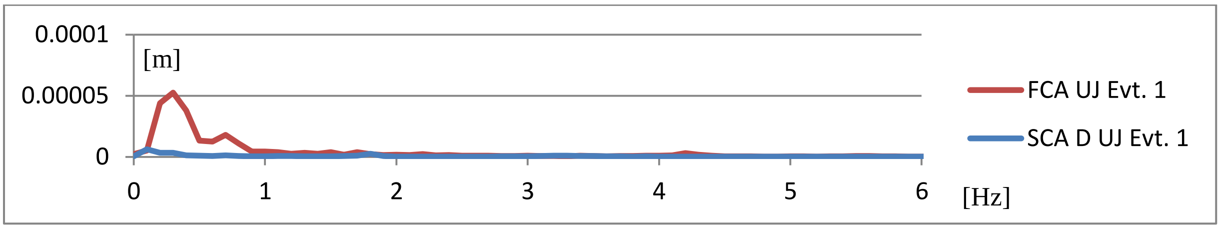


Figure 91 – Upper Junction Wave load only

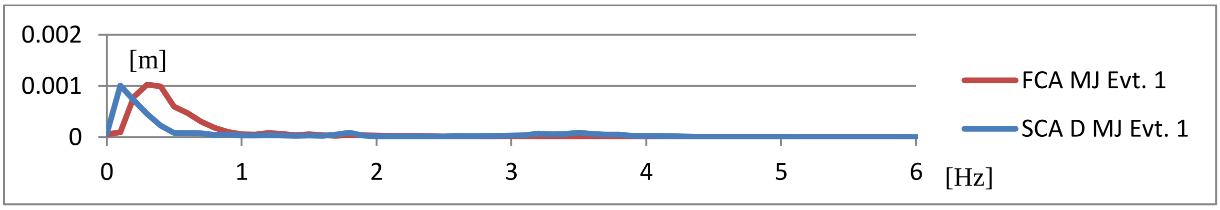


Figure 92 – Middle Junction Wave load only

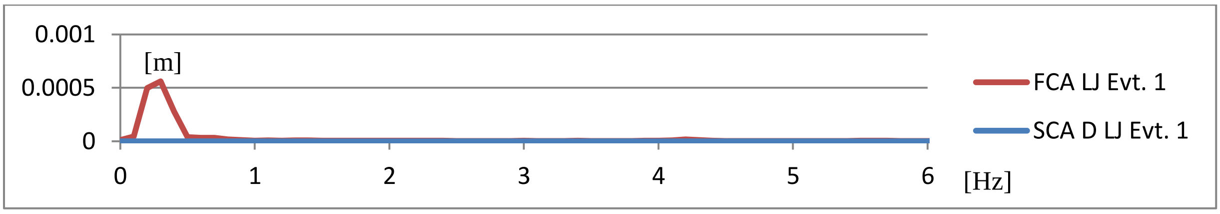


Figure 93 – Lower Junction Wave load only

6.7 Sensitivity analysis on mass- and damping matrices used in SCA

A sensitivity analysis for the damping and mass matrix was carried out to investigate the importance of the matrices. Analyses with different values for damping and mass were compared for event 1. The values tested were 10 and 20 % higher/lower than the original values for the analysis.

6.7.1 Estimated K-Joint damage - Damping matrix sensitivity analysis – Event 1

For the damping matrices there were not measured any influence of damage for the bracer.

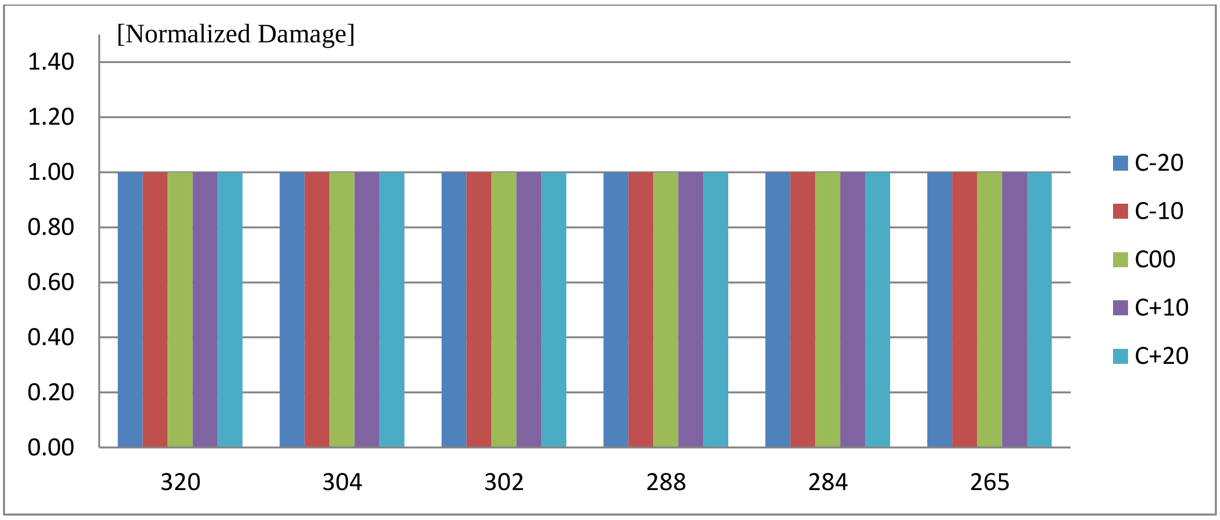


Figure 94 - Estimated K-Joint damage - Damping matrix sensitivity analysis – Event 1

6.7.2 Estimated K-Joint damage - Mass matrix sensitivity analysis – Event 1

The mass matrix had high influence of the damage for the bracer joints. To get fairly consistent results the increase and decrease ratio of the values was too high.

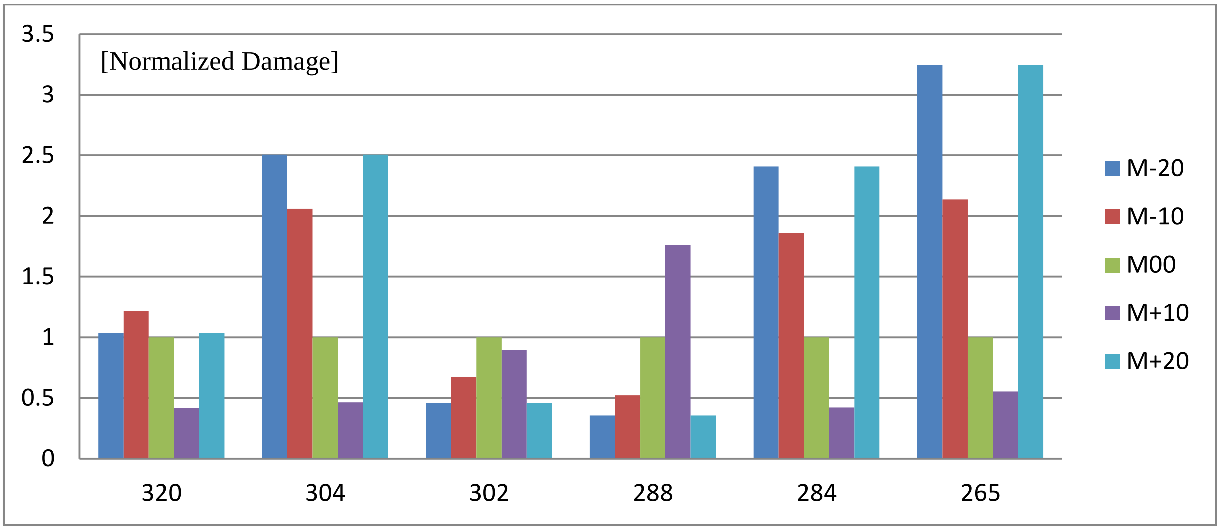


Figure 95 - Estimated K-Joint damage - Mass matrix sensitivity analysis – Event 1

6.7.3 Bracer FFT analyse – Damping matrix sensitivity analysis – Event 1

As for the K-Joint damage measured, there is no big difference between the FFT analyses for the different damping matrices.

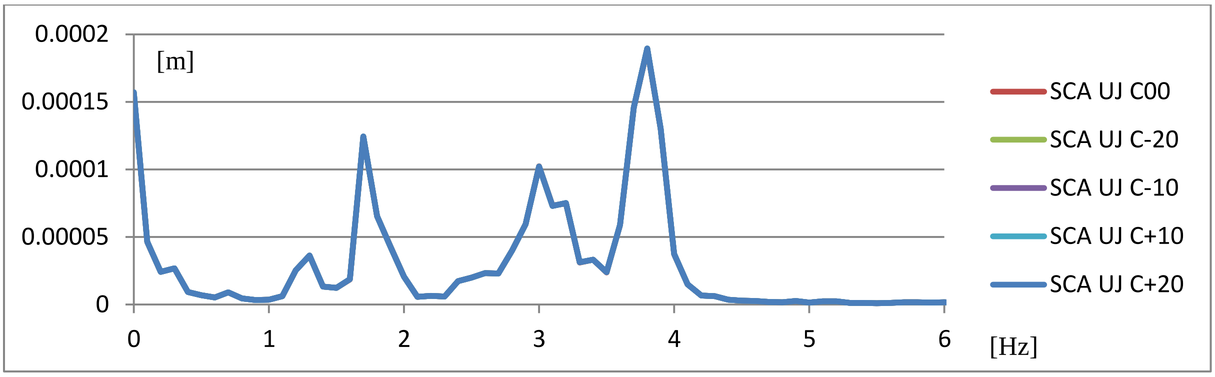


Figure 96 - Upper Junction - Event 1 - C matrix sensitivity analysis

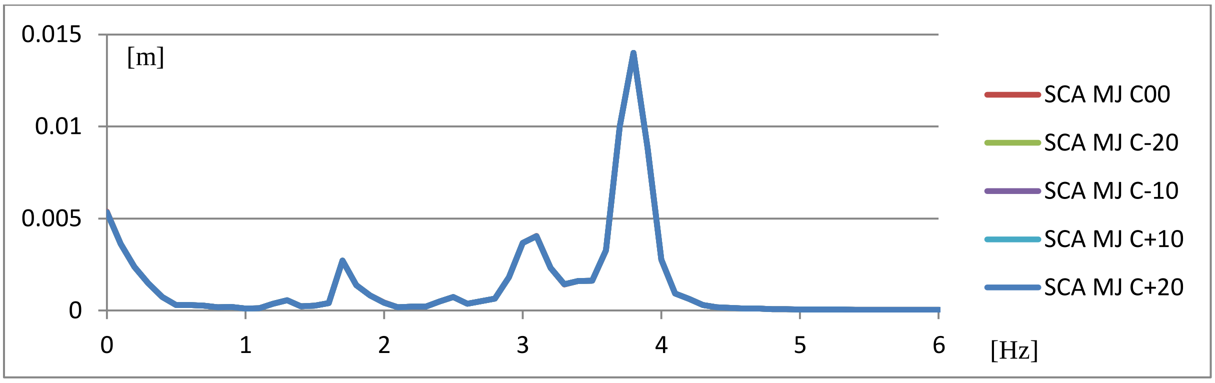


Figure 97 - Middle Junction - Event 1 - C matrix sensitivity analysis

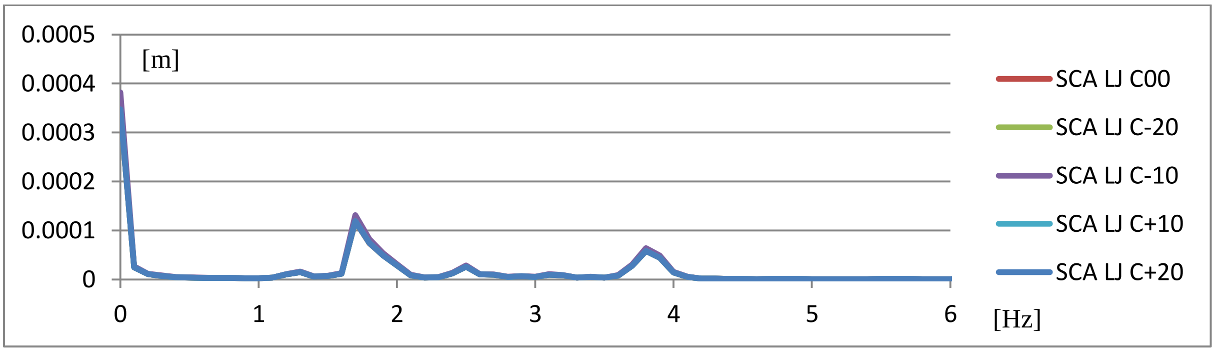


Figure 98 - Lower Junction - Event 1 - C matrix sensitivity analysis

6.7.4 Bracer FFT analyse – Mass matrix sensitivity analysis – Event 1

For the difference in mass matrix the characteristic of the FFT chart for the upper, middle and lower junction is fairly the same. There is only a small shift in frequency values for the amplitude peaks. The only significant difference is the amplitude of the fluctuations of the junctions.

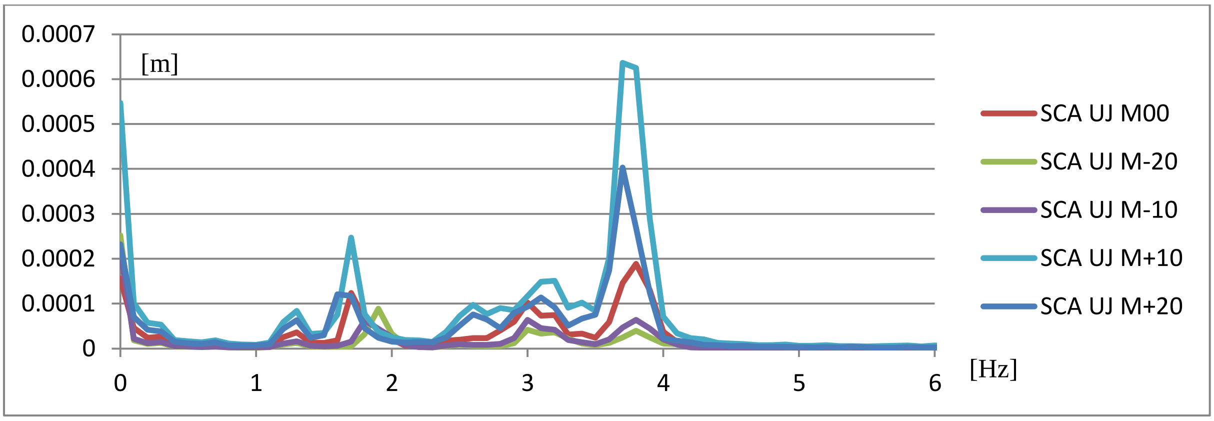


Figure 99 - Upper Junction - Event 1 - M matrix sensitivity analysis

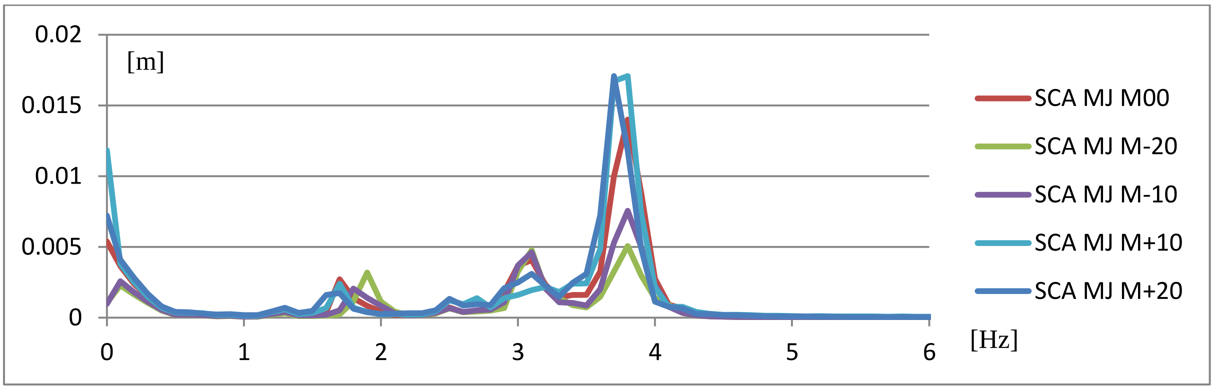


Figure 100 - Middle Junction - Event 1 - M matrix sensitivity analysis

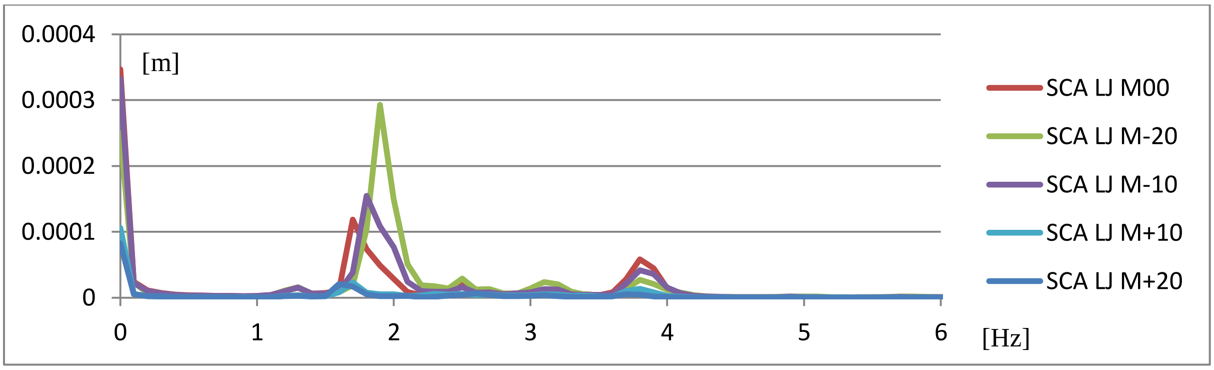


Figure 101 - Lower Junction - Event 1 - M matrix sensitivity analysis

6.8 Variation of analysis increment time and predicted damage

The fully coupled and sequentially coupled analyses considered in this master project were compared using a damage parameter. Both analysis methods included the windturbine loading from FEDEM. The SCA method utilized an analysis time increment at 0.01 seconds while in the FCA method the time increment was 0.02 seconds. Information given from TDA showed time increments 0.01 for SCA and 0.04 for FCA regarding in-house analysis.

It was of interest to see if the different time increments would affect the predicted damage in the analyses. The analyses were done using the FEDEM FCA, ruling out disturbances, if any, introduced by the SCA method. Three analyses were performed with time increments of 0.01, 0.02 and 0.04 seconds. The predicted damage from the FCA using $dt = 0.02s$ was scaled by a scale factor to unity, and the results from the analyses using $dt = 0.01s\ \ and\ 0.04s$ were scaled using the same scale factor.

The analysis setup was as described for a regular fully coupled analysis, with the exception of changed time increment. It is worth noticing that the same turbulent wind-file was used for all three analyses to maintain equal wind load. The resolution of the turbulent wind –file was $dt = 0.02s$.

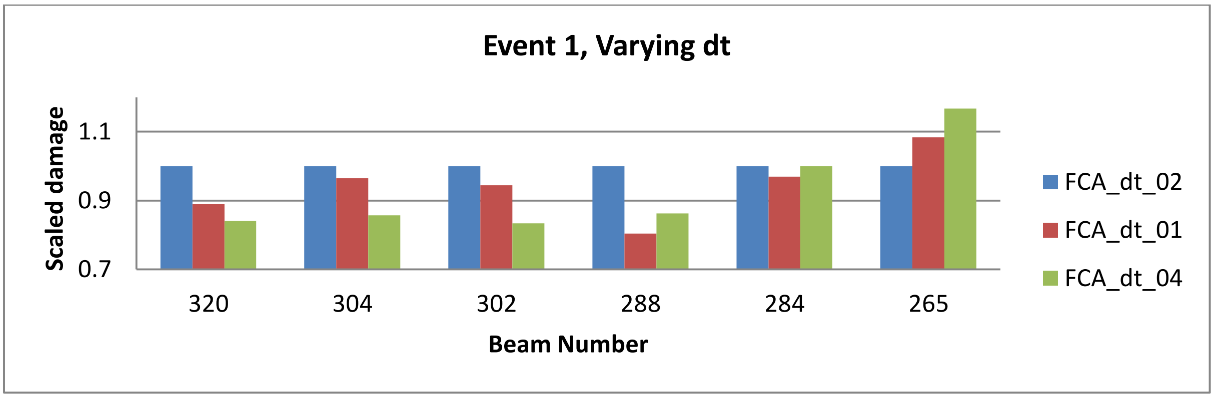


Figure 102 - FCA increment time variation

From the results of these analyses with varying increment time, it was observed a difference of up to 20% in the predicted damage compared to the analysis using $dt = 0.02s$. The study shows an increase in damage if the time increment is increased above $dt = 0.01s$. However, the analysis using $dt = 0.02s$ predicts more damage than the analysis using $dt = 0.04s$.

7 Discussion

7.1 Observations made from the analysis results

For the reader’s convenience, following abbreviations are used in the remains of this chapter:

SCA: Sequentially Coupled Analysis **FCA**: Fully Coupled Analysis

1) Eigen-frequencies for the jacket structure were extracted using USFOS. This was not done in FEDEM due to hardware related problems. The eigen-frequencies found were generally in agreement with the vibration frequencies predicted in USFOS using the SCA method.

2) The first results observed in this project showed that SCA overpredicted the damage with a factor of up to 10000 compared to the FCA (ref chapter 6.2.1). The expected tendency was that SCA would generally over-predict damage compared with FCA.

The cause of the large deviance had to be found. Debugging of the analysis and analysis tools indicated that the cause of deviance lay within the SCA analysis.

a) When the interface node displacement was compared between the FCA and SCA it became clear that the twisting motion around the z-axis for the SCA was much higher than for FCA. This can indicate that values of the **K**, **C** and **M** matrices affecting the rotational degree of freedom in z- direction causing the effect.

b) Comparison between SCA and FCA with a setup where interface node displacement from the FCA is used as input to SCA showed that the SCA in combination with USFOS predicted more damage than FCA in FEDEM.

c) Comparison between USFOS and FEDEM analysis setup with wave loads only on the jacket structure alone shows differences. USFOS predicted generally more damage than FEDEM when comparing wave loads.

d) A sensitivity study was done on the SCA where the **M-** and **C-** matrices were scaled respectively to study the influence on the predicted damage. The observed result was that the **M**- matrix variations contributed to some differences in damage, although not proportional to the scaling factor. An influence of changing the **C** matrix was not observed although expected.

e) A study was done in FEDEM to investigate if different time increments affected the predicted results. It was observed that there were small but significant differences between the predicted damage from the analysis due to the variation in time increments. The observed increase in damage was not proportional to the time increment.

Possible causes of these deviations will be discussed, and might also be a part of the effect observed, giving rise to more damage on the sequentially coupled analysis method. Please note the numbering, since they will be referred to in the following discussion.

7.1.1 Eigenfrequencies

The eigenfrequencies are not compared between USFOS and FEDEM, since they could not be extracted in FEDEM. However, an observation made was that FEDEM would generally predict a small frequency offset – approximately 1Hz towards the lower frequencies (ref 11.1). To what extent this has an influence is not known, apart from predicting that energy appears at lower frequencies, indicating less high frequent vibrations.

7.1.2 Making the needed K, C and M matrix for the SCA method (Guyan reduction)

Related to point 2 a)

To run a SCA, a Guyed reduction (ref literature study, chapter 4.4) has to be performed. Here the stiffness-, mass- and damping matrixes are extracted from the jacket structure analytically. This was done in order to give representative properties to a full spring element representing the whole jacket in FEDEM when doing the aero-elastic analysis. The extraction of the matrices was complex, especially the damping matrix that needed results from dynamic analysis. The complexity induces a possibility for errors in the results, and also reflects the vulnerability of the method.

It was observed for the SCA that the values affecting the rotational degree of freedom in z-direction was contributing to severe twisting of the complete structure, giving more dynamics in the bracers, leading to more damage. This can be directly related to the **K**, **C** and **M** matrices used for the aero elastic analysis within FEDEM, were only the **K** matrix was a full 6x6 matrix. The **C-** matrix were represented by diagonal values and the **M-** matrix was represented by four diagonal values – one describing mass and the last three describing rotational inertia.

It is also stated that the Guyan reduction is only accurate for static loads.

The above mentioned are thought to be the main reasons to the large discrepancies in the main results 2).

7.1.3 Comparison with FCA interface node displacement in SCA

Related to point 2 b)

An SCA was done using the interface node displacement from an FCA. Using this input on the SCA would remove a potential error generated by the Guyan reduction (extraction of **K**, **C** and **M** matrices) from the analysis. Discrepancies between the FCA and SCA would most likely relate to other differences between the analysis models. The findings were that the SCA predicted significantly higher damage compared to the FCA. The observed differences were that the SCA overestimated the damage with a factor of up to 80 compared to FCA (ref chapter 6.4.1), which is considerable lower than the results including the Guyan reduction.

This observation strengthens the suspicion on the Guyan reduction being the cause of the large deviations between the analysis methods. This due to the fact that the damage is 125 times higher when including the Guyan reduction in the SCA.

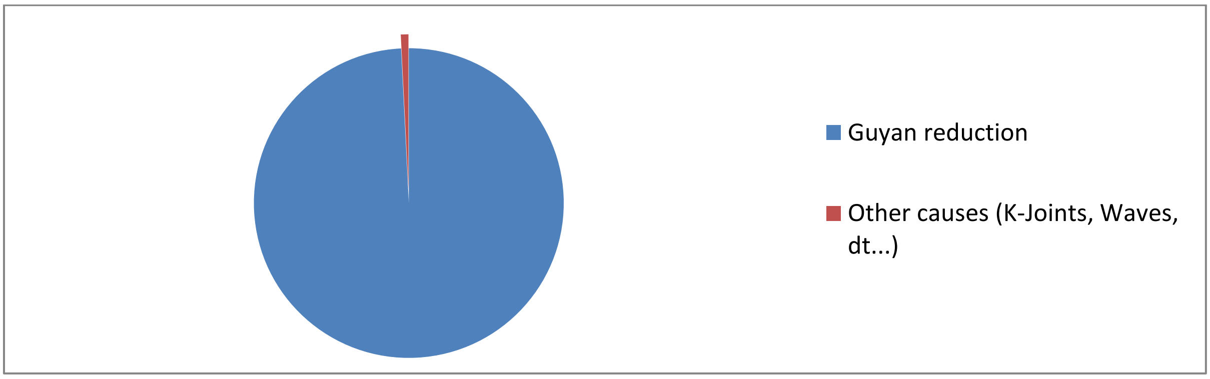


Figure 103 - Causes of deviance between FCA and SCA

In addition to the test mentioned, a similar test was made. Here the displacements from a FCA was added to a jacket model in FEDEM, and compared to the results from the USFOS analysis run with the same displacement input. The results from this comparison predicted differences between the models. It was observed that for the bracers above water FEDEM overpredicted the damage slightly compared to USFOS. The damage on the bracers lying in the splash zone were overestimated by USFOS. The damage predicted on the submerged bracers were highly overestimated by FEDEM. This observation might support a suspected difference in the wire pre-tension method between USFOS and FEDEM.

7.1.4 Comparison between response to waves in FEDEM and USFOS

Related to point 2 c)

In both USFOS and FEDEM a setup was made to allow for a dynamic analysis with irregular wave loading from event1 on the jacket structure. No other loads were imposed on the structure. The findings from the analyses showed that USFOS overestimates the damage compared to FEDEM with a factor of up to 180 (ref chapter 6.2.1). Some possible causes are listed here, and are discussed later:

- Increased K-Joint stiffness in USFOS due to modelling method
- Overestimation of mass in model in USFOS
- Wave discretization upon beam elements in FEDEM
- Different time increment in used the analysis codes

Note that the damage ratio mentioned here cannot be compared with damage ratios calculated from analysis including wind turbine loads. This is due to that observed damage resulting from wave loads are comparably small to the accumulated damage due to wind turbine loads (ref chapter 11.1-11.3 and 11.12).

7.1.5 Sensitivity study on C, and M matrices

A sensitivity study on the C and M matrices were performed to study the effect of relatively small changes (ref chapter 6.7). These analyses were conducted using USFOS and the setup for the last SCA step. It was observed that the **M**- matrix variations contributed to some differences in damage, although not proportional to the scaling factor. An influence of changing the **C** matrix was not observed, although it was expected. These results were surprising, and might be caused by human error, or using wrong method for measuring the difference (accumulated damage). The analyses were made “last minute”, and results are presented even if they were inconsistent, knowing that they cannot be used in arguments in this discussion.

7.1.6 Different time increments in SCA and FCA

Related to point 2 e)

Using different time increments (dt) in an FCA was observed to have small but noticeable effect. The observed difference may be related to the resolution of the turbulent wind field, which is the same for all three analyses considered. It might also relate to the time increment which was varied. However, the deviance observed by varying time increment is small, and not thought to be of considerable size with regards to the cause of the large deviance shown in the main results 2).

7.2 Possible sources of error

The project has been complex in many aspects, and many sources of error may exist. In the following, some possible causes of errors and factors related deviances in the results are discussed. There are probably other factors contributing to errors in addition to those mentioned here.

7.2.1 Two different FEA codes are used to produce the SCA

When the different FEA analysis were defined, much care was taken to make sure that the same input was used for both methods, knowing that differences in input will make differences in the output. The purpose was to end up with comparable results from two different analysis methods. Since the analyses software programs are different, there might be sources of deviance that have an effect on the results. The known differences in the software used are relating to:

Overestimation of damage due to waves

The accumulated damage from wave load on the structure is compared between USFOS and FEDEM without including wind (no errors induced from the SCA method). It is observed that

USFOS predicts much higher damage compared to FEDEM (ref *Figure 90*). The reason for this is unknown, but might be due one or both of the following factors:

- The discretization of wave forces upon beam elements might be quite different between the codes, and contributing to the effect. However, this effect will probably not be the cause of the large deviances found.
- In USFOS the K-joints are modelled differently from those in FEDEM, utilizing the function of nodal offset on beam elements. This is done in USFOS as a work-around to achieve the correct geometry at the same time as avoiding short beam elements. These K-Joints might be acting stiffer, and result in higher stresses. If this is the case, then the stresses will not be comparable between the two FEA codes, and other beam elements which are not directly linked to the K-joints should have been used for results extraction.

Overestimation on weight

There is a small overestimation of weight due to overlapping members at joints where beam elements connect at the intersection point of the centreline. This also leads to an overestimation of marine growth and hydrodynamic mass. The magnitude of the overestimation is not calculated, but is calculated to ~9% for structural mass and marine growth, and ~4.5% for hydrodynamic added mass for a similar structure, ref (NREL). This effect is not checked, but assumed to be equal in both FEA codes – thus not contributing to the large differences in 2).

A reported issue in USFOS and ASAS (ASAS is used by TDA) is that they to some degree overestimate hydrodynamic mass compared to FEDEM, ref (NREL). This can have an influence on the dynamic response on the structure, and may be a source of difference in damage predictions from USFOS and FEDEM. It is believed that the effect of this is small compared to other contributing factors to 2).

Performing SCA on a mix of Linear and Nonlinear code

USFOS is a nonlinear code while FEDEM is handling the jacket structure as if it was a linear code (no nonlinearities in the material for imported space frame structures). Combining linear and nonlinear codes when using a sequentially coupled analysis method is not optimal (ref chapter 4.5). The effect of this combination is not known.

7.2.2 The human factor

All work and analyses made during the project are complex and strongly dependant on human interaction, even though many tasks are handled by simple programs and scripts. It is therefore possible that errors caused by a human factor could exist in the results.

7.3 Differences due to analysis effects

7.3.1 Wire pre-tension on lower bracings

The wire pre-tension between the lower bracer junctions introduces high forces to the bracers. This give arise to certain effects, making it difficult to do correlations with the other bracer junctions. This is an observation and does not relate to the deviances in 2).

7.3.2 Correlation between out of plane vibrations and measured damage

It is thought that relatively high damage can be predicted without finding large out of plane vibration amplitudes in the bracer junctions. This might be due to twisting of the complete structure. The twisting movement is not necessarily generating much out of plane vibrations in centre of the bracer junction since it produces a circular motion around the structure’s vertical centre line. However, the end of the bracers, connected to the jacket legs using K-joints, might have relatively large rotational movements leading to both increased axial loads and bending moments on the beam elements. This results in higher predicted damage, without being able to measure significant vibrations in the bracer junctions.

This master thesis revolves around the theme of out of plane vibrations in the context of damage, but due to above mentioned observations, it is thought that analysing local vibrations alone will not tell the whole truth regarding the stress states.

7.4 Further work on the subject

If this master thesis was to continue, it would be interesting to find out following aspects:

7.4.1 FEA Code Validation

The FEA codes should have been thoroughly validated. One possible setup would have been to use only the jacket model and no waves or wind, introducing the same time-displacement signal on the structure in both FEDEM and USFOS. The differences could have been captured by the predicted damage from each code. The difference between the results from the codes would have indicated if the codes were comparable or not, that is if no other sources of differences were participating.

7.4.2 Wave comparison FEDEM versus USFOS

The wave definitions in FEDEM and USFOS can be defined by slightly different parameters. It is of interest to see if the wave energy is different from both codes although they are probably almost identical. This could have been investigated comparing the contents of the wave signals produced by both codes.

8 Conclusion

The given task addresses the question whether there are out of plane vibrations of 2-5 Hz with amplitudes ranging from 0.1-1% of the bracer diameter, and if these are real. The task also involves the question if a sequentially coupled method will overestimate the out of plane vibrations compared to a fully coupled analysis.

This master thesis generated results predicting these out of plane vibrations in the relevant frequency range. However, the results by the fully coupled analysis were more probable compared to results from the sequentially coupled analysis method, since the sequentially coupled analysis method predicted improbably high accumulated damage in a very short time period.

Factors contributing to the discrepancy between fully- and sequentially coupled analysis method have been studied. Several factors contributed to the difference, but it was observed that the factor contributing the most to the discrepancy was the method of generating the needed **K**, **C** and **M** matrices. These matrices are defining how the simplified jacket structure responds in the aero-elastic analysis when exerted to loads. The properties of the simplified structure in the aero-elastic analysis are very important since the dynamics from this analysis in turn prescribes the displacement of the interface node at the last sequential analysis step. Errors in the matrices will cause erroneous input, and thereby create discrepancies in the results from the sequentially coupled analysis.

The findings made in this master thesis highlights the vulnerability lying in the sequentially coupled analysis method when an analytical procedure is used to extract the **K**, **C** and **M** matrices, which are essential to the validity of the method.

9 Symbol list

Symbols	**Description**	**Unit**
- a	Acceleration	$[m/s^2]$
- A	Cross sectional area	$[m]$
- Cd	Damping coefficient	$[-]$
- Cm	Added mass coefficient	$[-]$
- D	Diameter	$[m]$
- dt	Analysis time increment	$[s]$
- F	Force	$[N]$
- Fa	Axial force	$[N]$
- fs	Frequency range (in FFT)	$[s^{-1}]$
- fv	Ocilatory frequency	$[s^{-1}]$
- g	Gravity constant	$[m/s^2]$
- Hs	Significant wave height	$[m]$
- I	Moment of inertia	$[m^4]$
- M	Moment	$[Nm]$
- m	Mass	$[kg]$
- N	Number of increments in FFT signal	$[-]$
- St	Strouhal number	$[-]$
- $Tmin$	Minimum period of irregular waves	$[s]$
- $Tmax$	Maximum period of irregular waves	$[s]$
- Tp	Spectral peak period	$[s]$
- U_∞	Velocity far upstream	$[m/s]$
- y_{max}	Maximum distance from centerline to membrane	$[m]$
- $\emptyset$	Diameter	$[m]$
- $\Delta\sigma$	Stress range	$[MPa]$
- λ	Wave length	$[m]$
- σ_a	Stress amplitude	$[MPa]$
- σ_{ar}	Fully reversed stres amplitude	$[MPa]$
- σ_{axial}	Axial stress	$[MPa]$
- $\sigma_{bending}$	Bending stress	$[MPa]$
- $\sigma_{f'}$	Material constant for use in fatigue calculations	$[MPa]$
- σ_m	Mean stress	$[MPa]$
- σ_{total}	Total stress due to axial force and moment	$[MPa]$

10 Literature references

(u.d.). Hentet fra Matlab Rainflow counting by Adam Nieslony: www.mathworks.com/matlabcentral/fileexchange/3026-rainflow-counting-algorithm

ASAS (Non-Linear) User Manual - Version 14.0. (2011).

AA242B: MECHANICAL VIBRATIONS - Direct Time-Integration Methods. (u.d.).

About us: TDA A/S Organization chart. (u.d.). Hentet fra TDA A/S - Website: www.tda.as/about_us/company/content/image_034eddd4-4890-4f69-8cfb-cb5b81fdd754/1354177884512.MaxSize.msw-w.w-756.msh-h.h-424.Save.img

ASAS Software - Website. (u.d.). Hentet fra ANSYS - Website: http://www.ansys.com/Products/Other+Products/ANSYS+ASAS

Cimbala, J. M. (2010). *Fourier Transforms, DFTs, and FFTs.* Penn State University.

DNV-OS-J101 – DESIGN OF OFFSHORE WIND TURBINE STRUCTURES. (u.d.).

DNV-RP-C205 - ENVIRONMENTAL CONDITIONS AND ENVIRONMENTAL LOADS. (u.d.).

Dowling, N. E. (u.d.). *AISI 4340.*

Faltinsen, O. M. (1998). *SEA LOADS ON SHIPS AND OFFSHORE STRUCTURES.* Cambridge: Cambridge University Press.

FEDEM Windpower - Website. (u.d.). Hentet fra FEDEM - Website: http://www.fedem.com/programvare/fedem-windpower-7.0/

Geradin, M., & Rixen, D. (u.d.). *Mechanical Vibrations: Theory and Applications to Structural Dynamics.* Wiley, John & Sons.

Jonkman, J., Butterfield, S., Musial, W., & Scott, G. (2009). *Definition of a 5-MW Reference Wind Turbine for Offshore System Development.*

Jonkman, J., Butterfield, S., Musial, W., & Scott, G. (u.d.). *Wind Turbine for Offshore System Development.*

Main Page: TDA A/S - Webside. (u.d.). Hentet fra TDA A/S - Webside: www.tda.as

Main Page: USFOS. (u.d.). Hentet fra USFOS - Website: http://usfos.com/

MSC Nastran Reference Manual. (u.d.). Hentet fra MSC Software: http://simcompanion.mscsoftware.com/resources/sites/MSC/content/meta/DOCUMENTATION/9000/DOC9188/~secure/refman.pdf?token=WDkwz5Q6v7LTw9Vb5p+nwkbZMJAxZ4rU6BoR7AHZFxi2Tl1QdrbVvWj00qmcC4+S3fnbL4WUa5ovbpBwGDBt+zFPzsGyYC13zvGPg0j/5SrMF6bnWrQoTGyJb8ho1ROYUMvA0u

Musculus, M., & NTNU-Trondheim. (u.d.). *The full-height lattice tower.* 9th Deep Sea Offshore Wind R&D Seminar, Trondheim, Norway.

Muskulus, M. (2012). The full-height lattice tower concept.

Muskulus, M., Zwick, D., Schafhirt, S., Brommundt, M., Sampathkumar, N., Mechineau, J., & Haugsøen, P. B. (u.d.). *Comparison of different approaches to load calculation for the OWEC Quattropod (R) jacket support structure.*

NREL. (u.d.). *Offshore Code Comparison Collaboration Continuation (OC4).*

Offshore wind: NREL Website. (u.d.). Hentet fra NREL Website: http://www.nrel.gov/wind/offshore_wind.html

Ormonde Jacket - *Website*. (u.d.). Hentet fra http://lorc.e-kvator.com/cgi-bin/lorc/uploads/images/Knowledge/Wind/Support%20structures/Jacket-terminology_large.jpg

Ormonde Midsection - *Website*. (u.d.). Hentet fra http://www.vattenfall.co.uk/en/ormonde-inauguration/ormonde-offshore-wind-farm.htm

Ormonde Wind Turbine - *Website*. (u.d.). Hentet fra http://subseaworldnews.com/2012/07/12/global-marine-provides-subsea-cables-to-ormonde-wind-farm-uk/global-marine-provides-subsea-cables-to-ormonde-wind-farm-uk/

Seidel, M., & Ostermann, F. (u.d.). *Validation of Offshore load simulations using measurement data from the DOWNVInD project.*

Seidel, M., von Mutius, M., Rix, P., & Steudel, D. (u.d.). *Integrated analysis of wind and wave loading for complex support structures of Offshore Wind Turbines.*

11 ATTACHMENTS

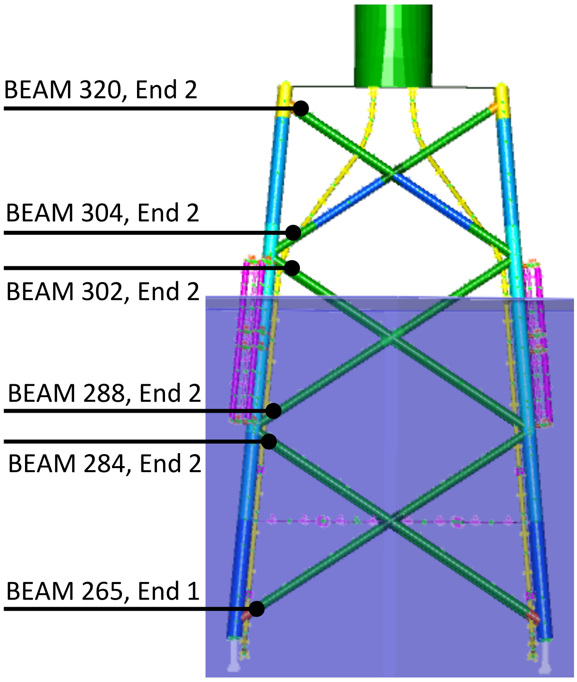
11.1 Fully Coupled Analysis- Event 1 - Fedem

11.1.1 Description

- Fully coupled Fedem analysis
- K, C, and M matrices replacing the soil piles properties
- Turbine setup according to “Definition of a 5-MW Reference Wind Turbine for Offshore System Development”
    - J. Jonkman, S. Butterfield, W. Musial, and G. Scott
- Event 1

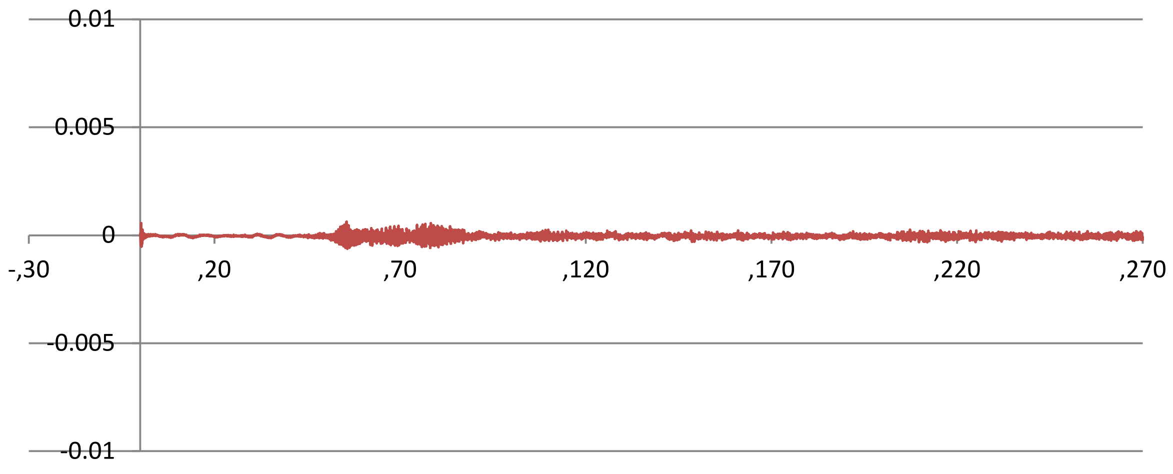
11.1.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	2.452E+00
Stress range	$\Delta\sigma$ [MPa]	9.244E+00
Damage		5.516E-09
Beam 304		
Max stress	σ_{max} [MPa]	5.039E+00
Stress range	$\Delta\sigma$ [MPa]	1.576E+01
Damage		2.703E-08
Beam 302		
Max stress	σ_{max} [MPa]	1.170E+01
Stress range	$\Delta\sigma$ [MPa]	1.931E+01
Damage		2.673E-08
Beam 288		
Max stress	σ_{max} [MPa]	1.163E+01
Stress range	$\Delta\sigma$ [MPa]	1.659E+01
Damage		3.952E-08
Beam 284		
Max stress	σ_{max} [MPa]	1.228E+02
Stress range	$\Delta\sigma$ [MPa]	3.426E+01
Damage		2.249E-07
Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	1.062E+02
Damage		1.218E-07

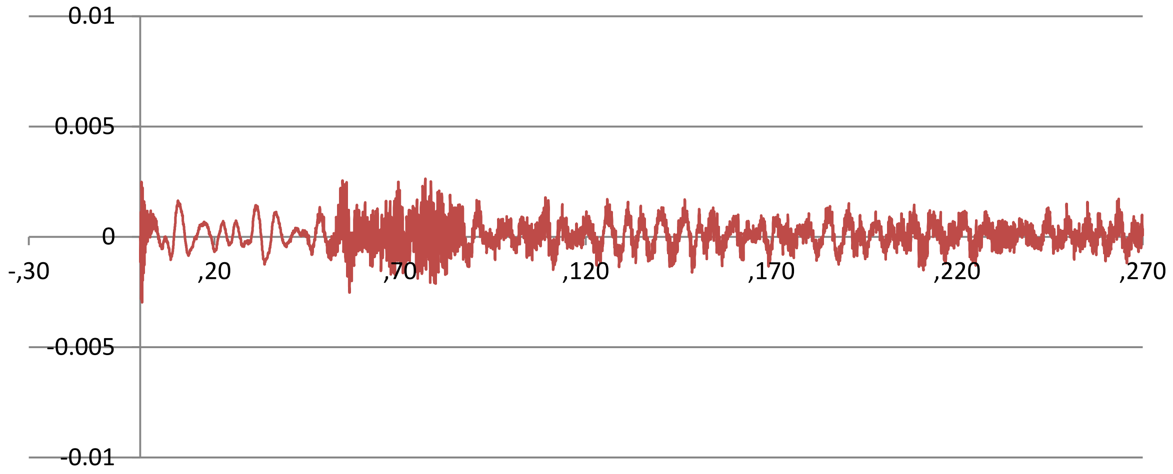


11.1.3 Bracer displacement

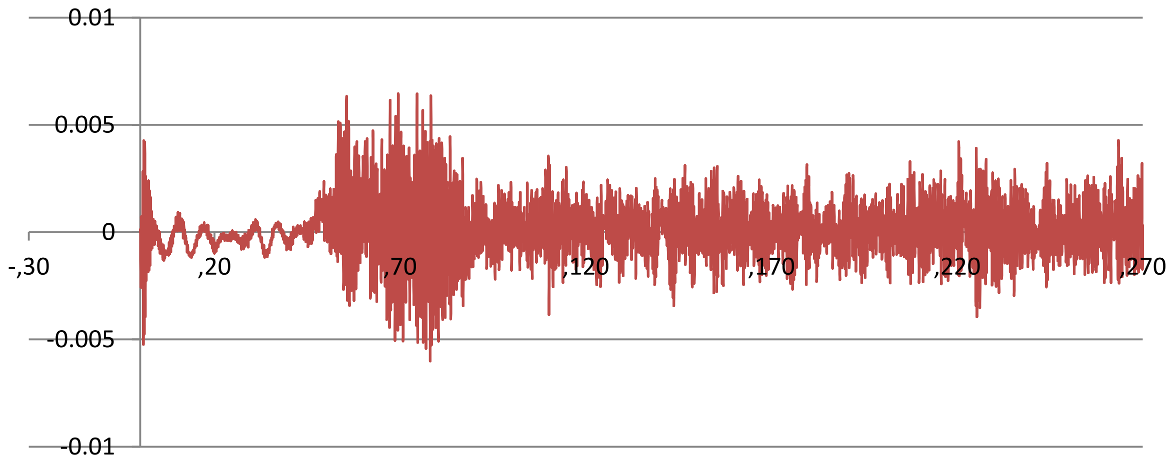
Upper Junction



Mid Junction

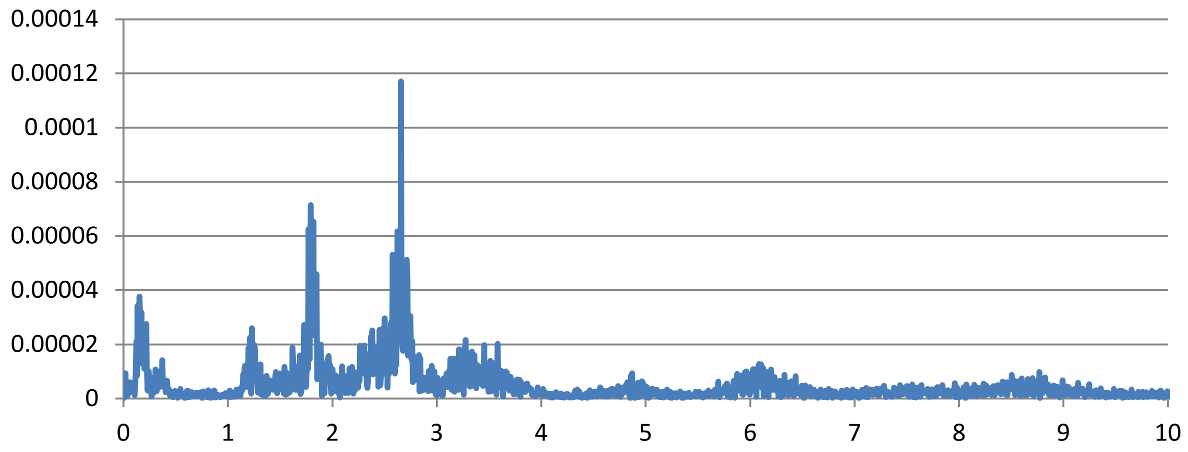


Lwr Junction

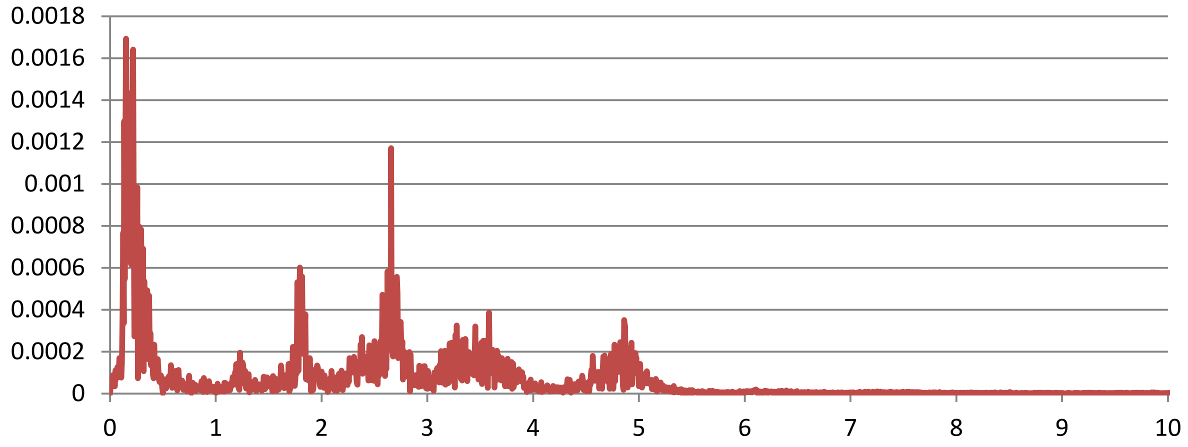


11.1.4 Bracer FFT Analysis

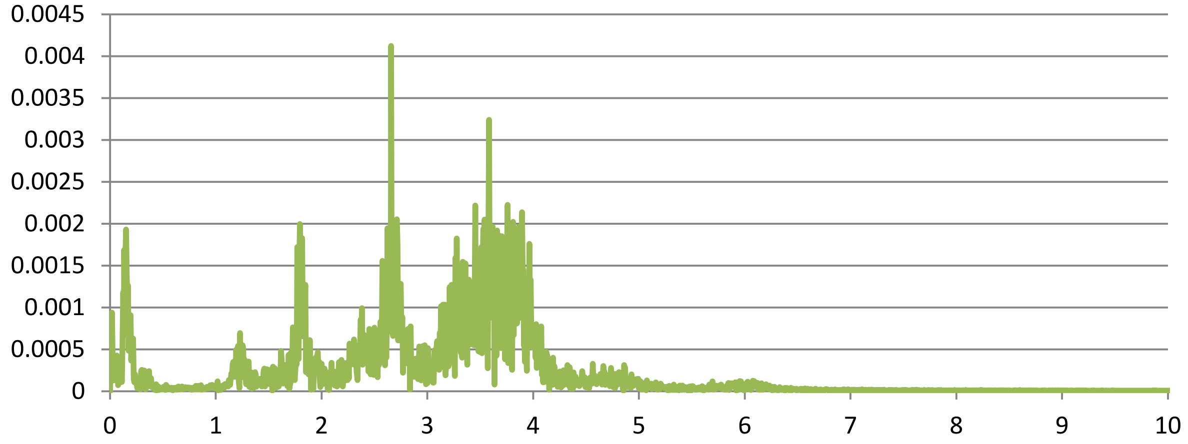
Upper Junction



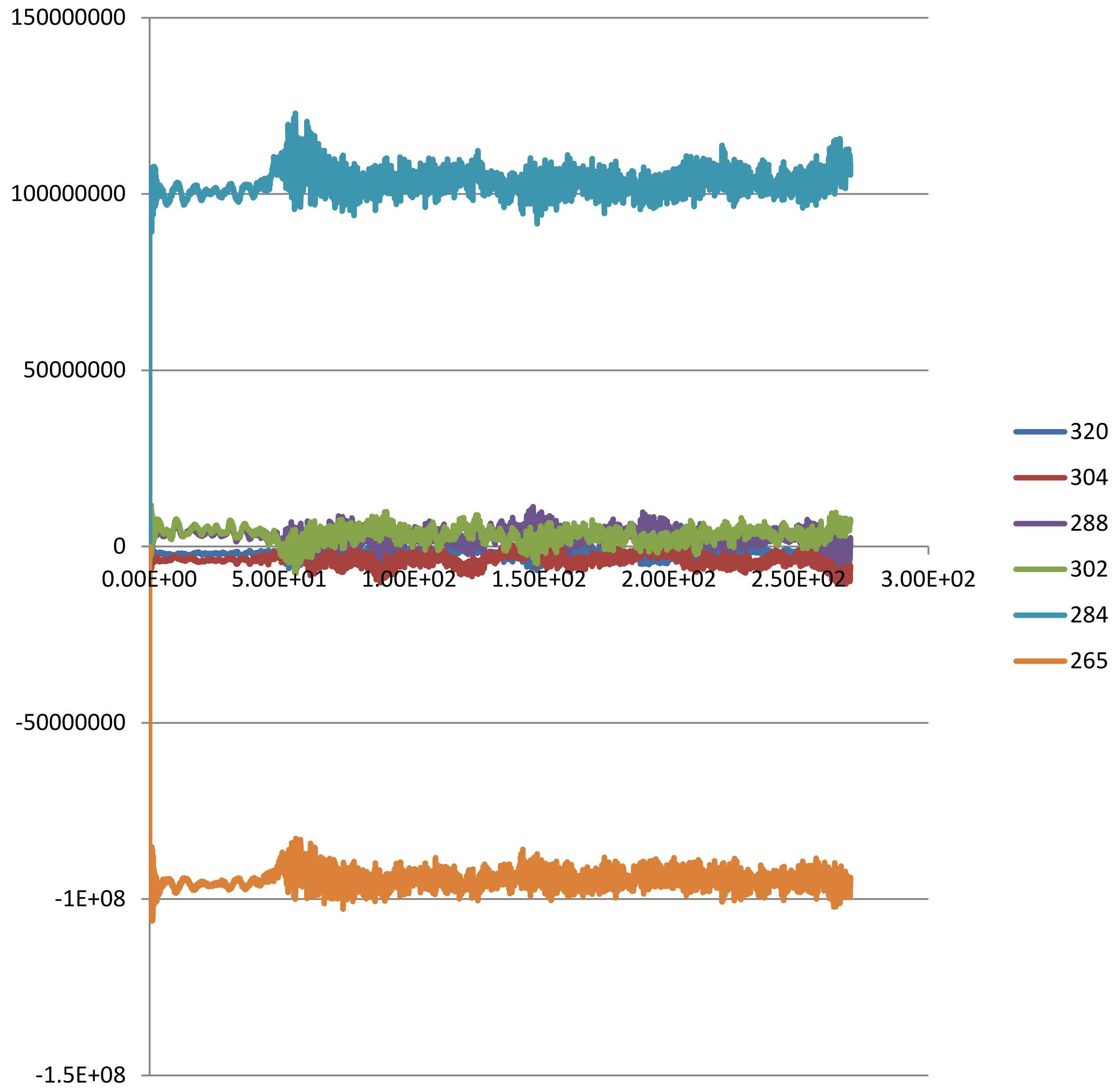
Mid Junction



Lwr Junction



11.1.5 Beam stresses



Max Beam Stresses

320	304	302	288	284	265	
2.45E+06	5.04E+06	1.17E+07	1.16E+07	1.23E+08	2.65E+02	*[Pa]*
2.452	5.039	11.698	11.634	122.842	0.000	*[MPa]*

Min Beam Stresses

320	304	302	288	284	265	
-6.79E+06	-1.07E+07	-7.61E+06	-4.96E+06	8.86E+07	-1.06E+08	*[Pa]*
-6.792	-10.723	-7.614	-4.956	88.580	-106.227	*[MPa]*

Delta sigma

320	304	302	288	284	265	
9.244	15.762	19.312	16.590	34.262	106.227	*[MPa]*

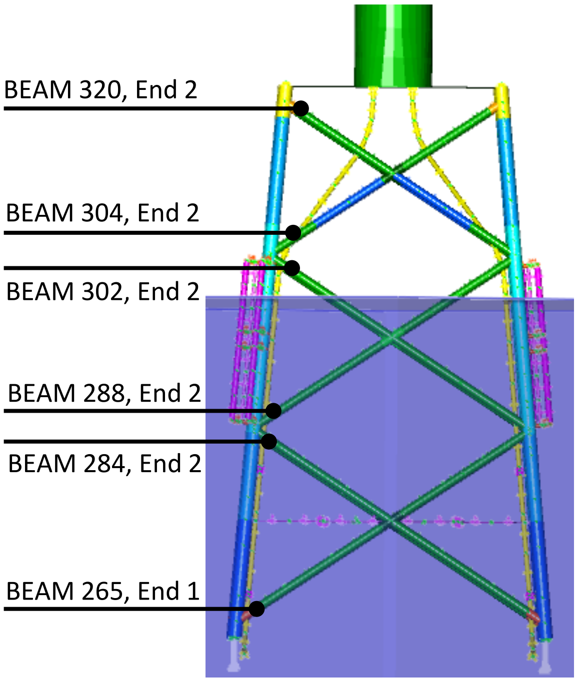
11.2 Fully Coupled Analysis- Event 2 - Fedem

11.2.1 Description

- Fully coupled Fedem analysis
- K, C, and M matrices replacing the soil piles properties
- Turbine setup according to “Definition of a 5-MW Reference Wind Turbine for Offshore System Development”
    - J. Jonkman, S. Butterfield, W. Musial, and G. Scott
- Event 2

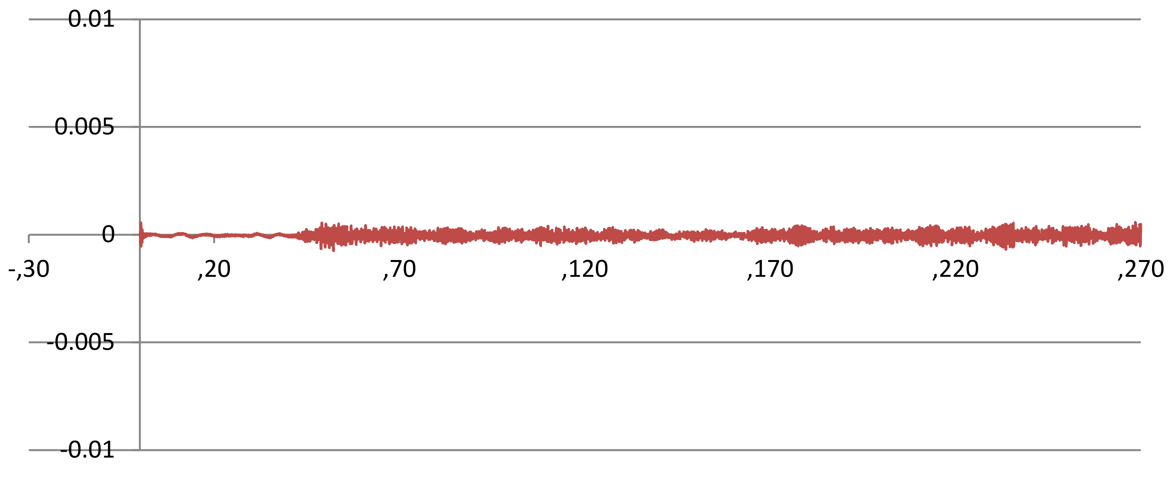
11.2.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	3.425E+00
Stress range	$\Delta\sigma$ [MPa]	1.038E+01
Damage		1.117E-08
Beam 304		
Max stress	σ_{max} [MPa]	5.105E+00
Stress range	$\Delta\sigma$ [MPa]	1.584E+01
Damage		4.456E-08
Beam 302		
Max stress	σ_{max} [MPa]	1.144E+01
Stress range	$\Delta\sigma$ [MPa]	2.024E+01
Damage		5.805E-08
Beam 288		
Max stress	σ_{max} [MPa]	1.189E+01
Stress range	$\Delta\sigma$ [MPa]	1.699E+01
Damage		8.079E-08
Beam 284		
Max stress	σ_{max} [MPa]	1.222E+02
Stress range	$\Delta\sigma$ [MPa]	3.377E+01
Damage		7.785E-07
Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	1.056E+02
Damage		4.506E-07

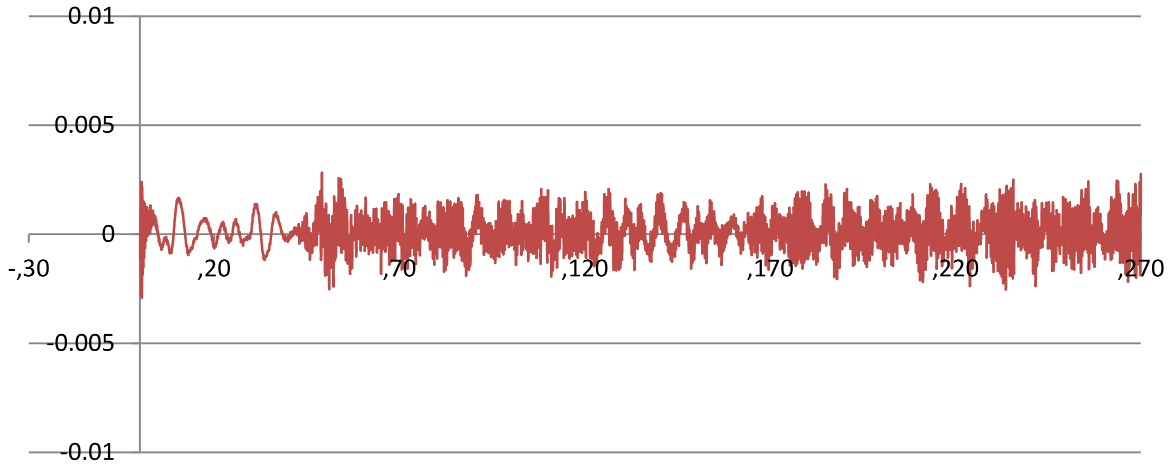


11.2.3 Bracer displacement

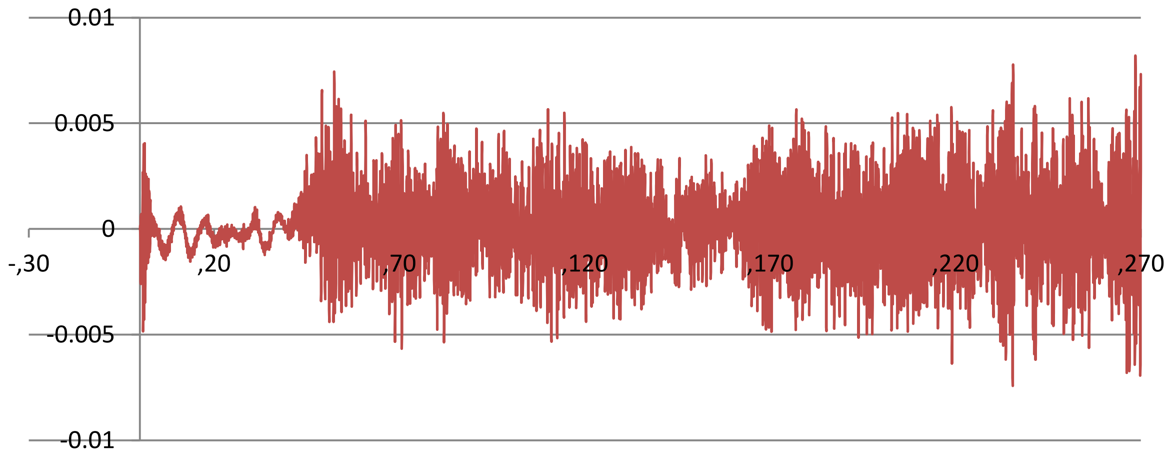
Upper Junction



Mid Junction

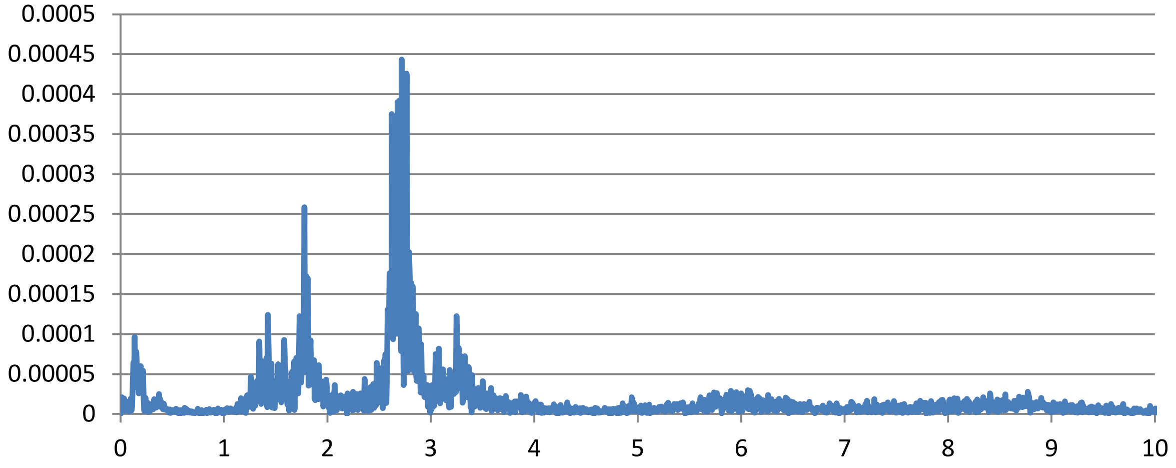


Lwr Junction

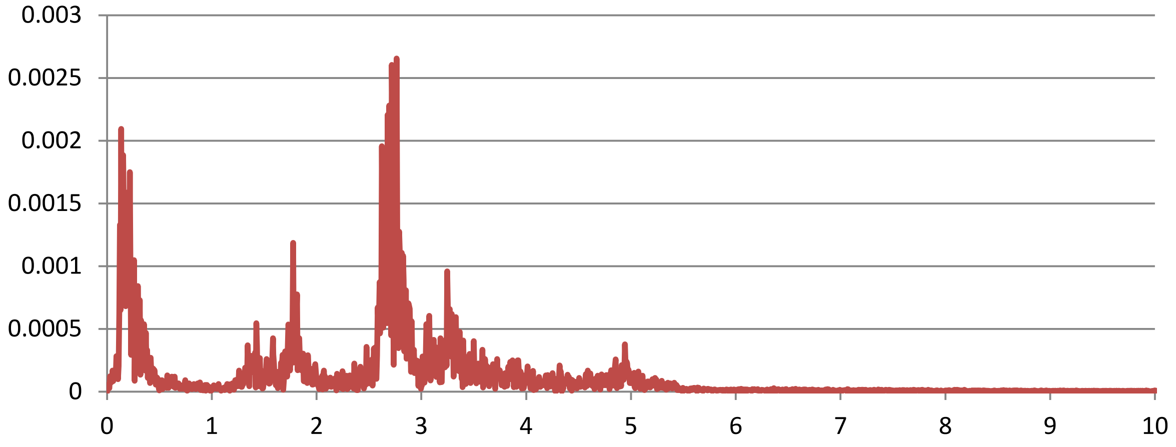


11.2.4 Bracer FFT Analysis

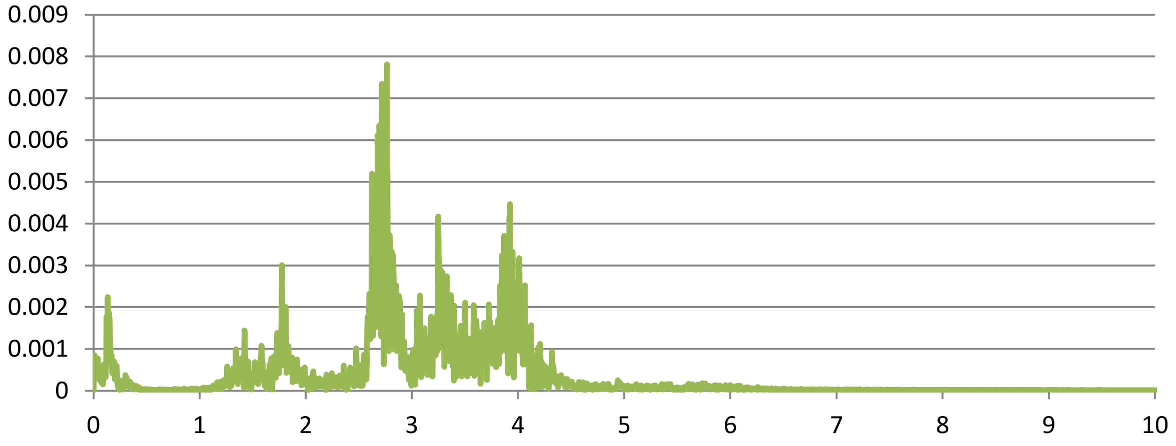
Upper Junction



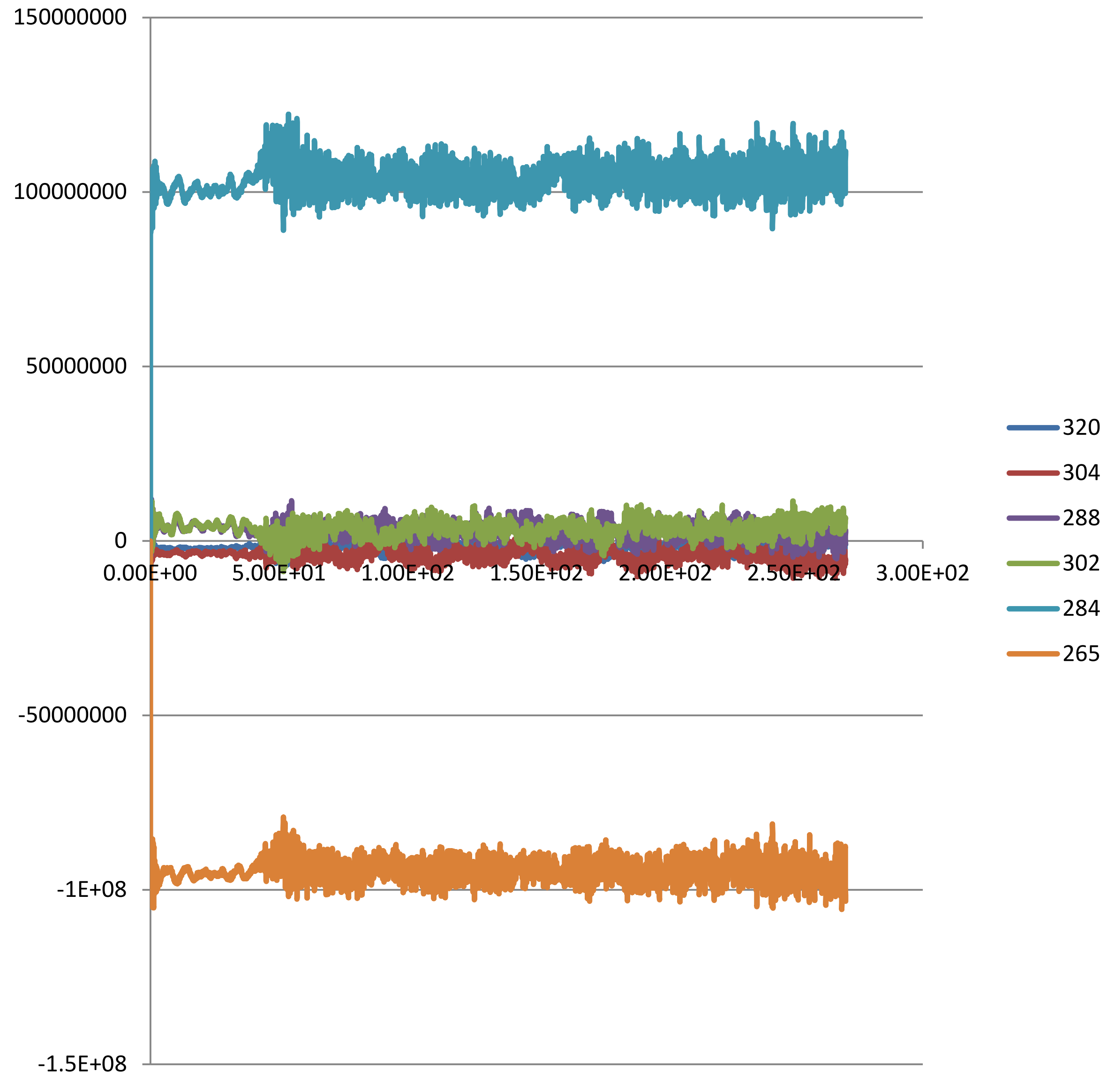
Mid Junction



Lwr Junction



11.2.5 Beam stresses



Max Beam Stresses

320	304	302	288	284	265	
3.42E+06	5.10E+06	1.14E+07	1.19E+07	1.22E+08	2.65E+02	*[Pa]*
3.425	5.105	11.438	11.888	122.214	0.000	*[MPa]*

Min Beam Stresses

320	304	302	288	284	265	
-6.95E+06	-1.07E+07	-8.80E+06	-5.10E+06	8.84E+07	-1.06E+08	*[Pa]*
-6.952	-10.731	-8.802	-5.100	88.448	-105.634	*[MPa]*

Delta sigma

320	304	302	288	284	265	
10.377	15.836	20.240	16.988	33.766	105.634	*[MPa]*

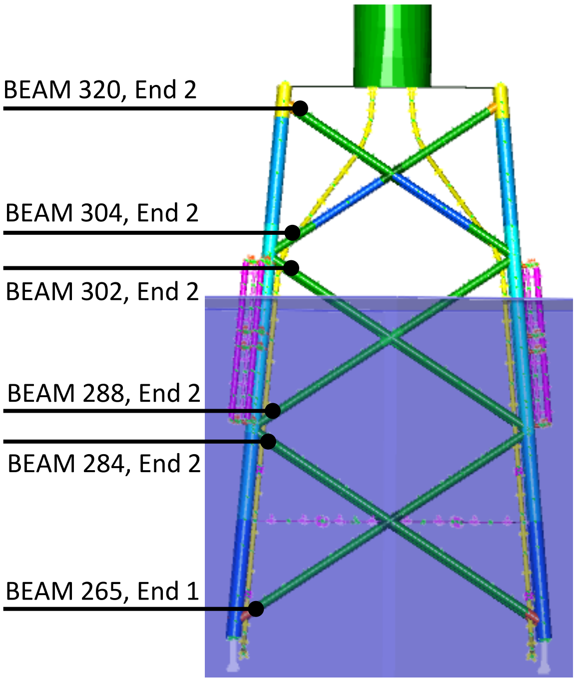
11.3 Fully Coupled Analysis- Event 3 - Fedem

11.3.1 Description

- Fully coupled Fedem analysis
- K, C, and M matrices replacing the soil piles properties
- Turbine setup according to “Definition of a 5-MW Reference Wind Turbine for Offshore System Development”
    - J. Jonkman, S. Butterfield, W. Musial, and G. Scott
- Event 3

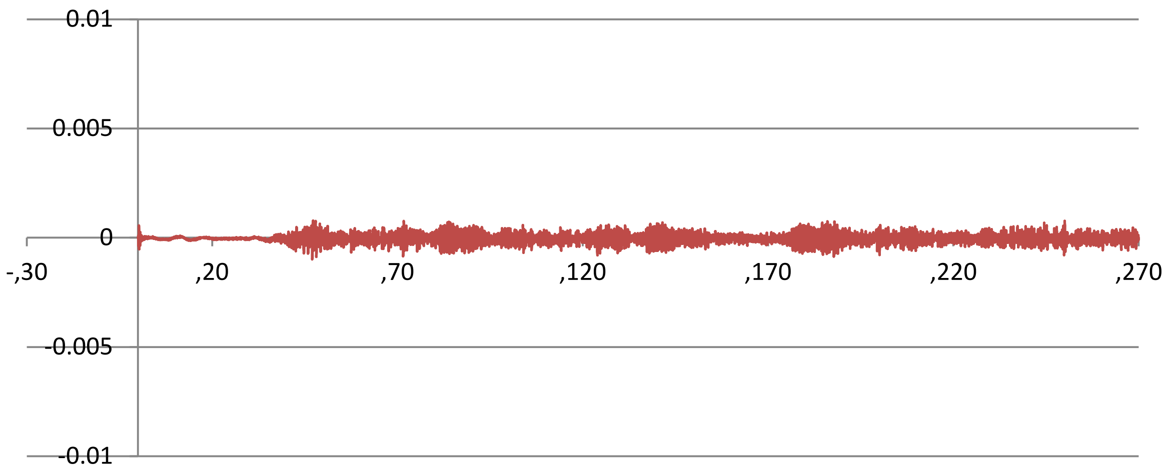
11.3.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	5.067E+00
Stress range	$\Delta\sigma$ [MPa]	1.227E+01
Damage		2.730E-08
Beam 304		
Max stress	σ_{max} [MPa]	4.472E+00
Stress range	$\Delta\sigma$ [MPa]	2.004E+01
Damage		9.594E-08
Beam 302		
Max stress	σ_{max} [MPa]	1.542E+01
Stress range	$\Delta\sigma$ [MPa]	2.620E+01
Damage		1.435E-07
Beam 288		
Max stress	σ_{max} [MPa]	1.164E+01
Stress range	$\Delta\sigma$ [MPa]	2.187E+01
Damage		1.861E-07
Beam 284		
Max stress	σ_{max} [MPa]	1.337E+02
Stress range	$\Delta\sigma$ [MPa]	4.909E+01
Damage		2.191E-06
Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	1.185E+02
Damage		1.379E-06

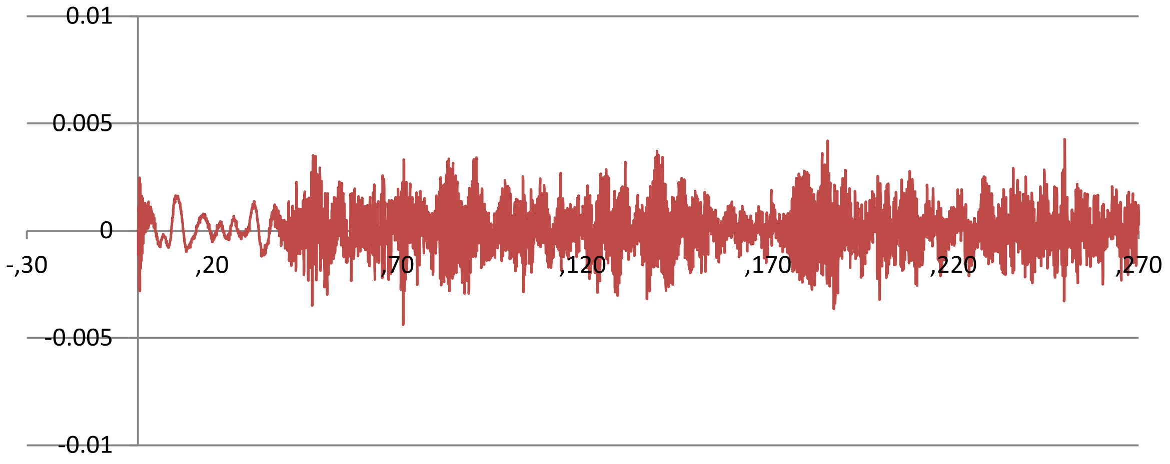


11.3.3 Bracer displacement

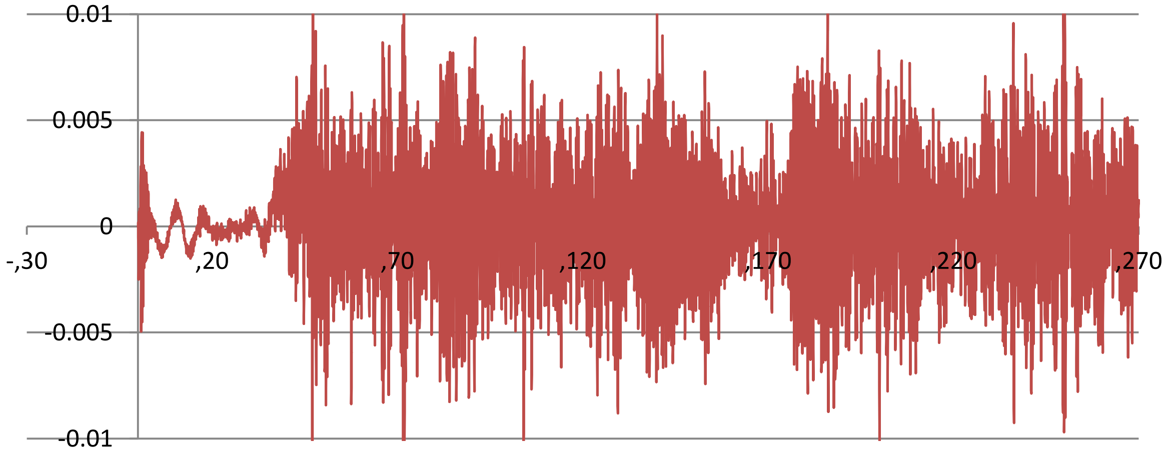
Upper Junction



Mid Junction

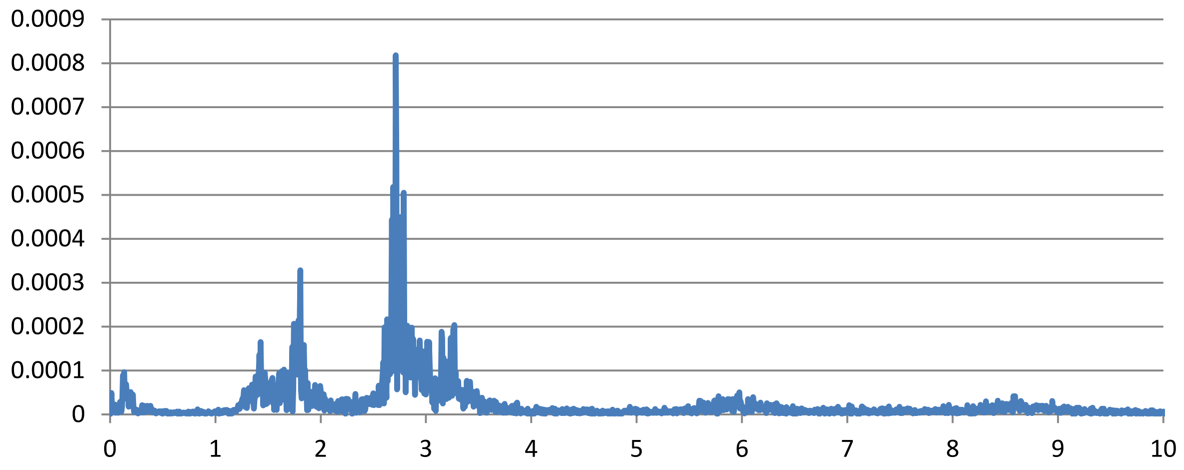


Lwr Junction

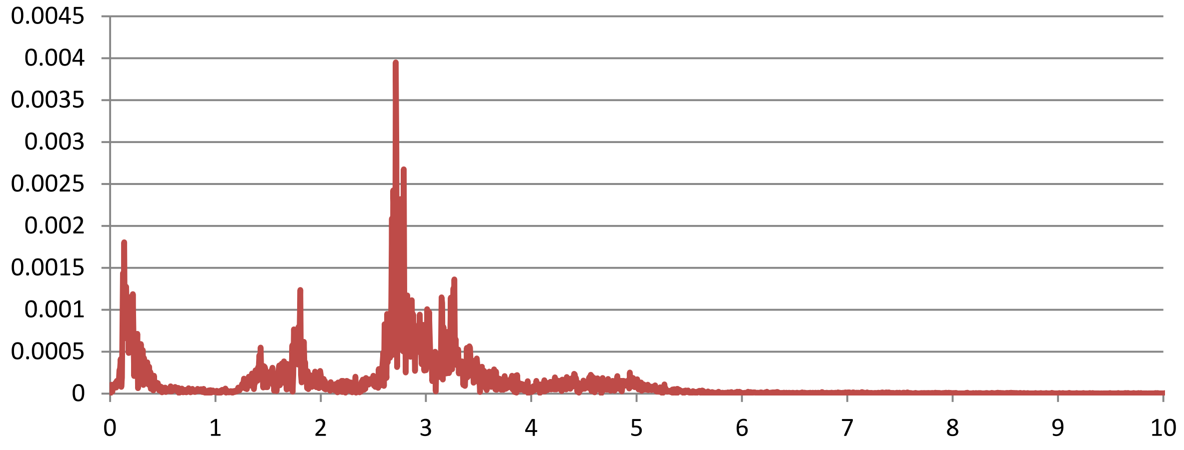


11.3.4 Bracer FFT Analysis

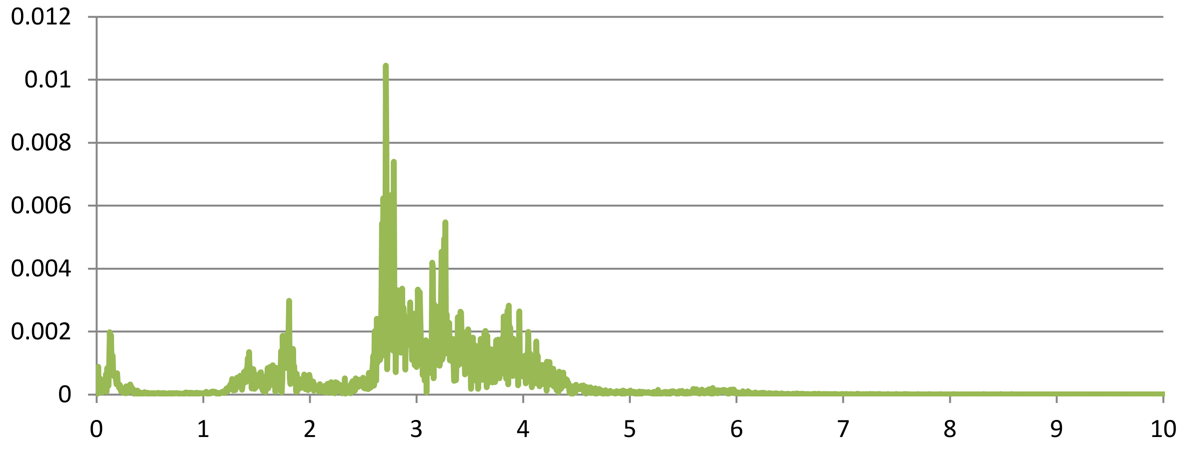
Upper Junction



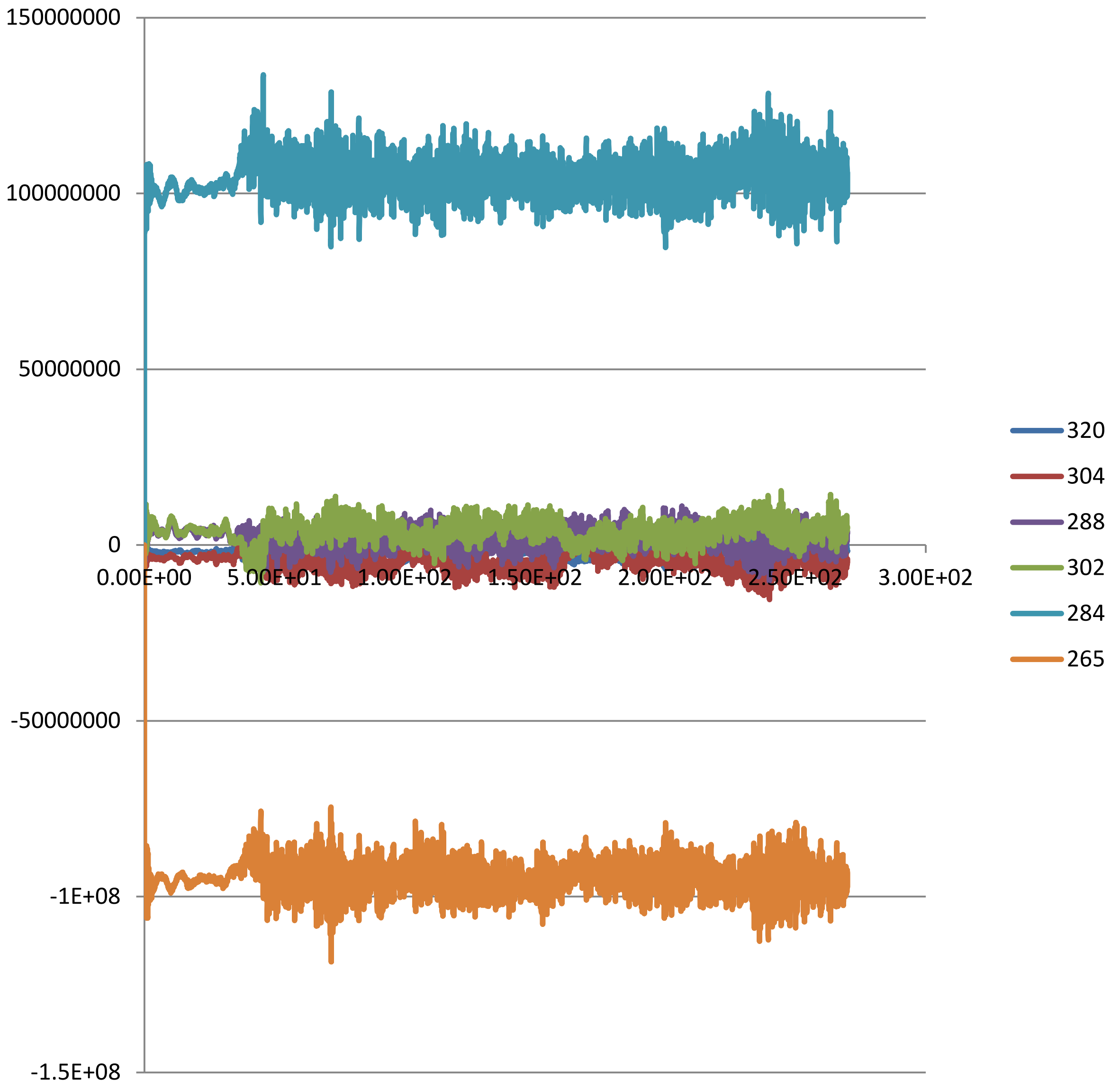
Mid Junction



Lwr Junction



11.3.5 Beam stresses



Max Beam Stresses

320	304	302	288	284	265	
5.07E+06	4.47E+06	1.54E+07	1.16E+07	1.34E+08	2.65E+02	*[Pa]*
5.067	4.472	15.420	11.640	133.697	0.000	*[MPa]*

Min Beam Stresses

320	304	302	288	284	265	
-7.21E+06	-1.56E+07	-1.08E+07	-1.02E+07	8.46E+07	-1.19E+08	*[Pa]*
-7.207	-15.567	-10.778	-10.226	84.604	-118.516	*[MPa]*

Delta sigma

320	304	302	288	284	265	
12.274	20.039	26.198	21.866	49.093	118.516	*[MPa]*

11.4 Sequentially Coupled Analysis- Event 1-All DOF Free-Usfos/Fedem

11.4.1 Description

- Fully coupled Usfos/Fedem analysis
- K, C, and M matrices replacing the interface node for Fedem analysis
    - For all DOF
- Linear springs along soil piles for soil properties
- Turbine setup in Fedem according to “Definition of a 5-MW Reference Wind Turbine for Offshore System Development”
    - J. Jonkman, S. Butterfield, W. Musial, and G. Scott
- Event 1

11.4.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	2.126E+01
Stress range	$\Delta\sigma$ [MPa]	3.598E+01
Damage		9.445E-07

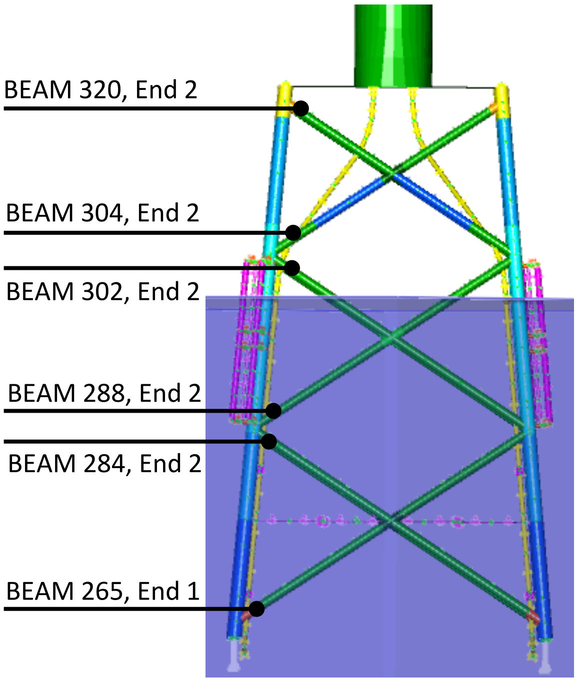
Beam 304		
Max stress	σ_{max} [MPa]	3.252E+01
Stress range	$\Delta\sigma$ [MPa]	5.738E+01
Damage		3.206E-06

Beam 302		
Max stress	σ_{max} [MPa]	5.371E+01
Stress range	$\Delta\sigma$ [MPa]	1.218E+02
Damage		1.058E-04

Beam 288		
Max stress	σ_{max} [MPa]	4.357E+01
Stress range	$\Delta\sigma$ [MPa]	1.066E+02
Damage		6.225E-05

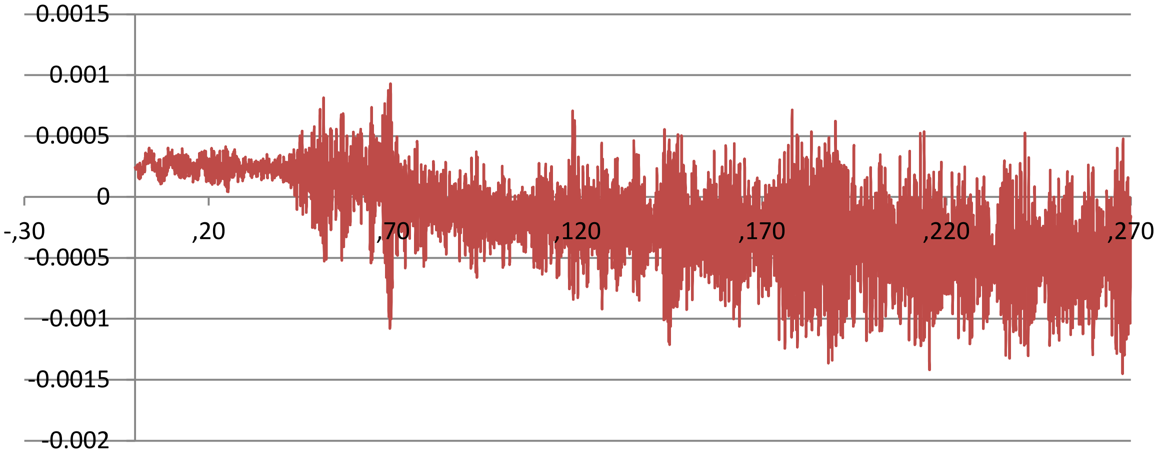
Beam 284		
Max stress	σ_{max} [MPa]	2.840E-04
Stress range	$\Delta\sigma$ [MPa]	2.071E+02
Damage		6.452E-06

Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	1.287E+02
Damage		1.306E-06

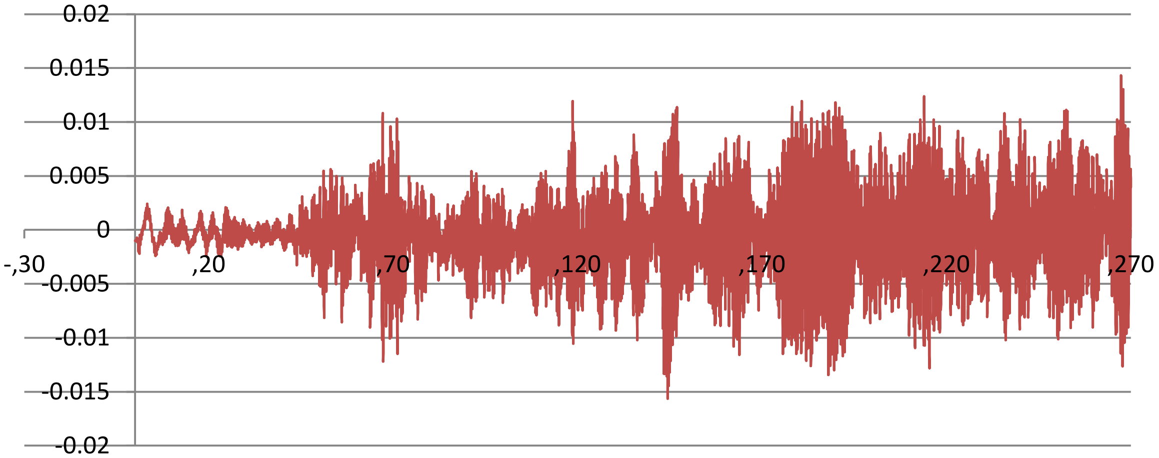


11.4.3 Bracer displacement

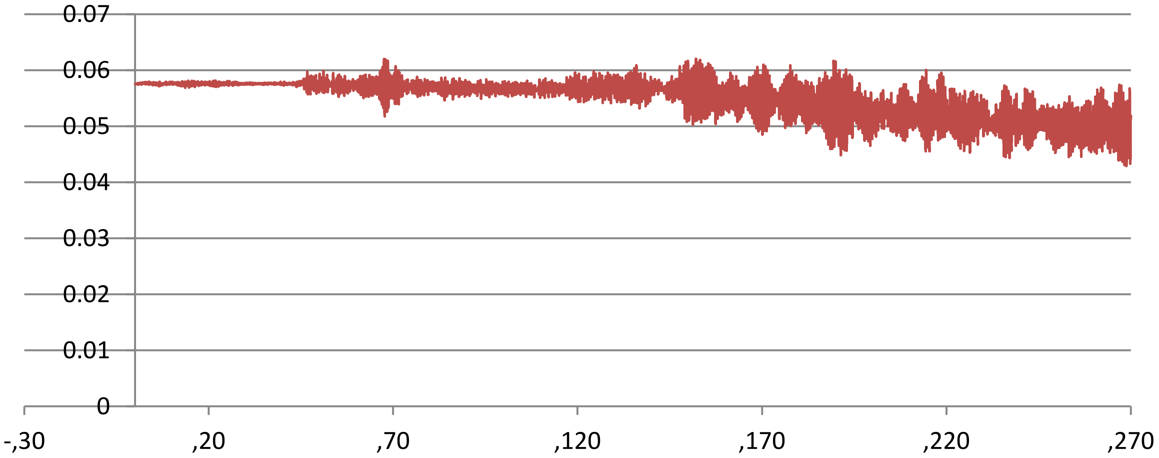
Upper Junction



Mid Junction

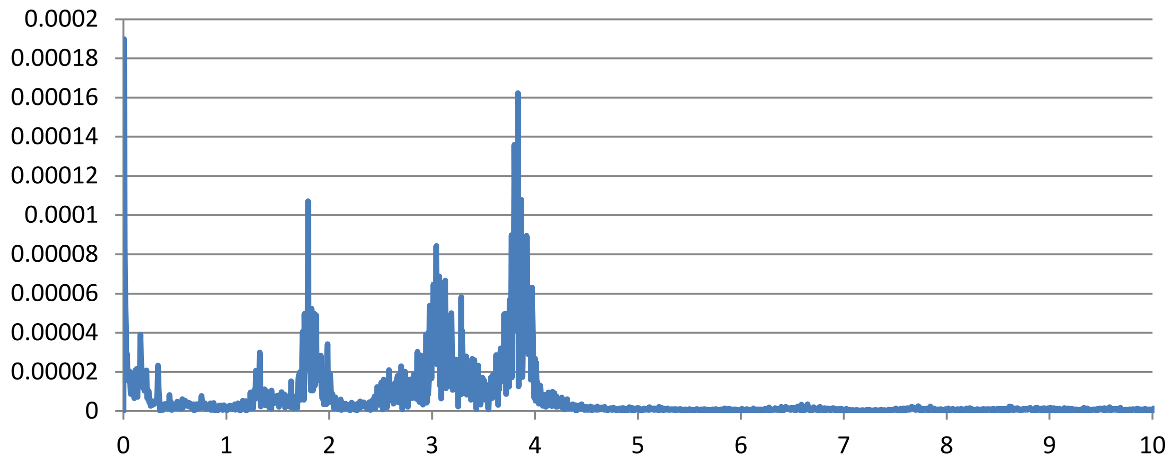


Lwr Junction

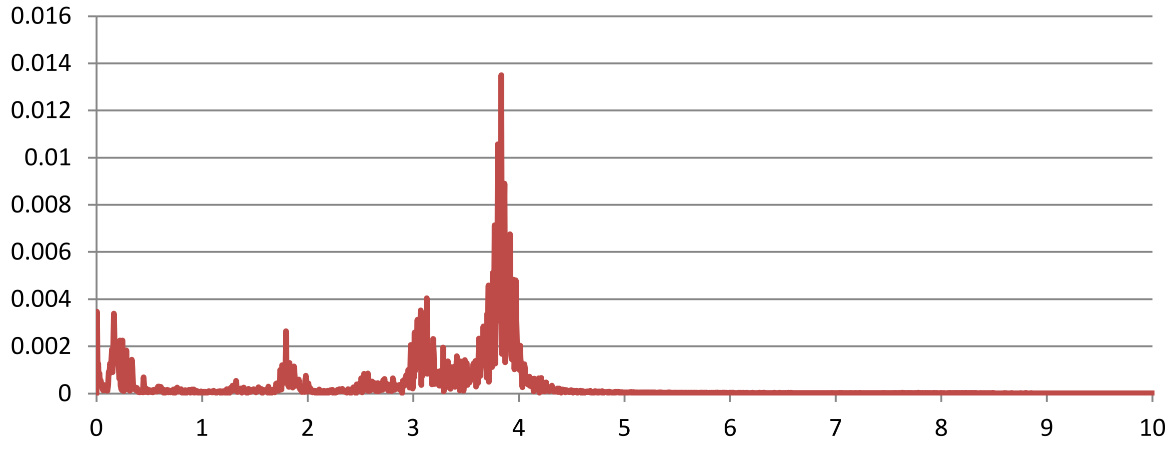


11.4.4 Bracer FFT Analysis

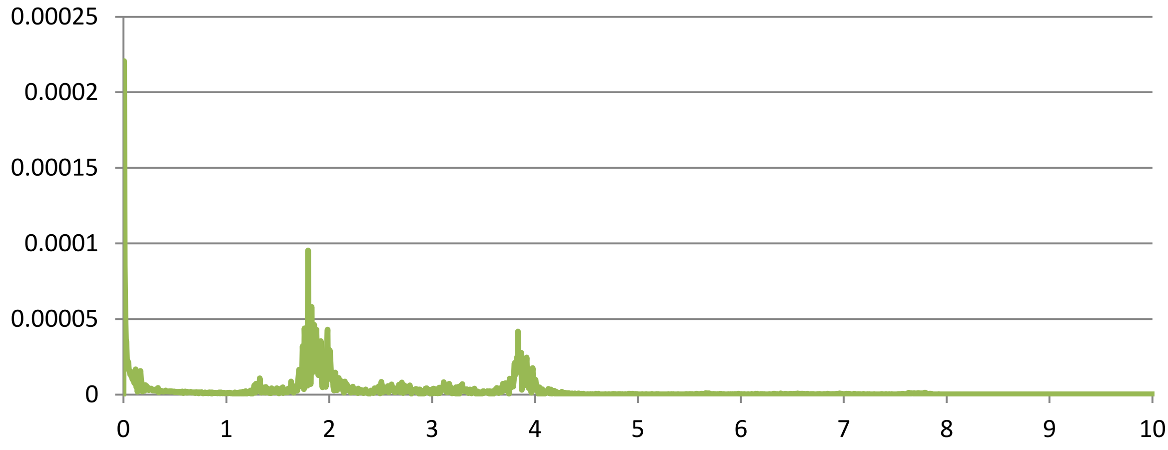
Upper Junction



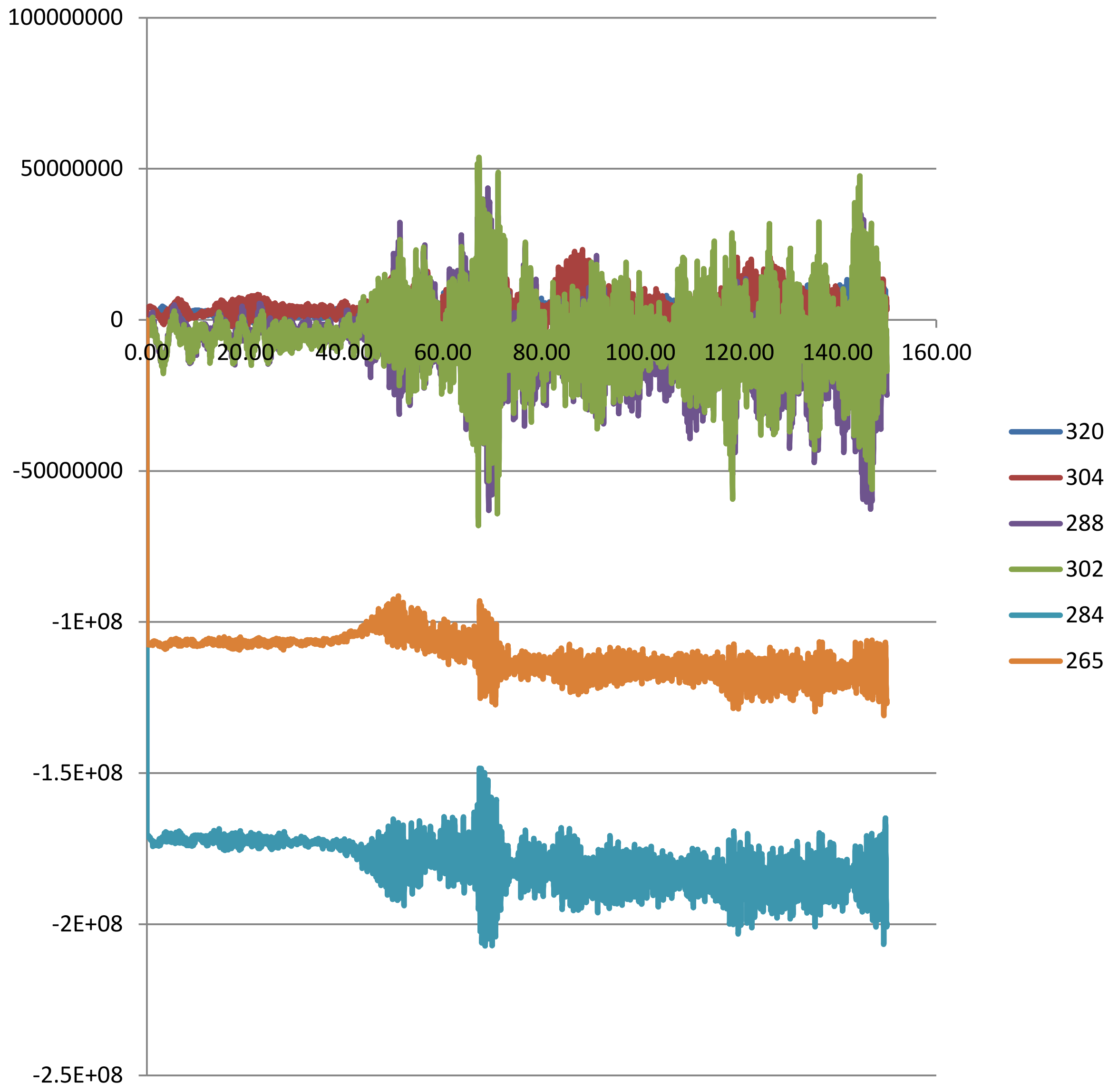
Mid Junction



Lwr Junction



11.4.5 Beam stresses



Max Beam Stresses

320	**304**	**302**	**288**	**284**	**265**	
2.13E+07	3.25E+07	5.37E+07	4.36E+07	2.84E+02	2.65E+02	*[Pa]*
21.255	32.516	53.710	43.569	0.000	0.000	*[MPa]*

Min Beam Stresses

320	**304**	**302**	**288**	**284**	**265**	
-1.47E+07	-2.49E+07	-6.81E+07	-6.31E+07	-2.07E+08	-1.29E+08	*[Pa]*
-14.729	-24.859	-68.067	-63.058	-207.148	-128.731	*[MPa]*

Delta sigma

320	**304**	**302**	**288**	**284**	**265**	
35.984	57.375	121.777	106.627	207.149	128.731	*[MPa]*

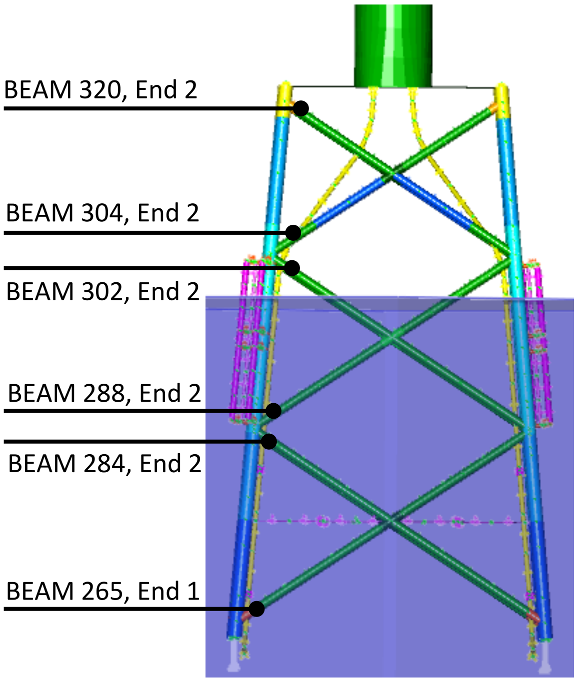
11.5 Sequentially Coupled Analysis- Event 2-All DOF Free-Usfos/Fedem

11.5.1 Description

- Fully coupled Usfos/Fedem analysis
- K, C, and M matrices replacing the interface node for Fedem analysis
    - For all DOF
- Linear springs along soil piles for soil properties
- Turbine setup in Fedem according to “Definition of a 5-MW Reference Wind Turbine for Offshore System Development”
    - J. Jonkman, S. Butterfield, W. Musial, and G. Scott
- Event 2

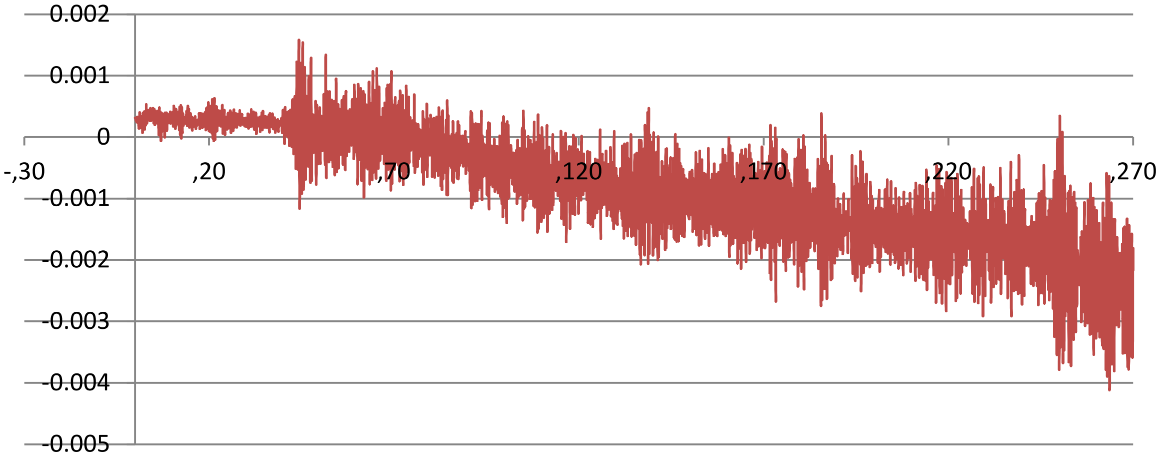
11.5.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	4.170E+01
Stress range	$\Delta\sigma$ [MPa]	5.627E+01
Damage		4.000E-06
Beam 304		
Max stress	σ_{max} [MPa]	5.194E+01
Stress range	$\Delta\sigma$ [MPa]	8.641E+01
Damage		1.073E-05
Beam 302		
Max stress	σ_{max} [MPa]	5.425E+01
Stress range	$\Delta\sigma$ [MPa]	1.177E+02
Damage		1.044E-04
Beam 288		
Max stress	σ_{max} [MPa]	3.727E+01
Stress range	$\Delta\sigma$ [MPa]	1.018E+02
Damage		7.129E-05
Beam 284		
Max stress	σ_{max} [MPa]	2.840E-04
Stress range	$\Delta\sigma$ [MPa]	2.148E+02
Damage		1.843E-05
Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	1.328E+02
Damage		3.093E-06

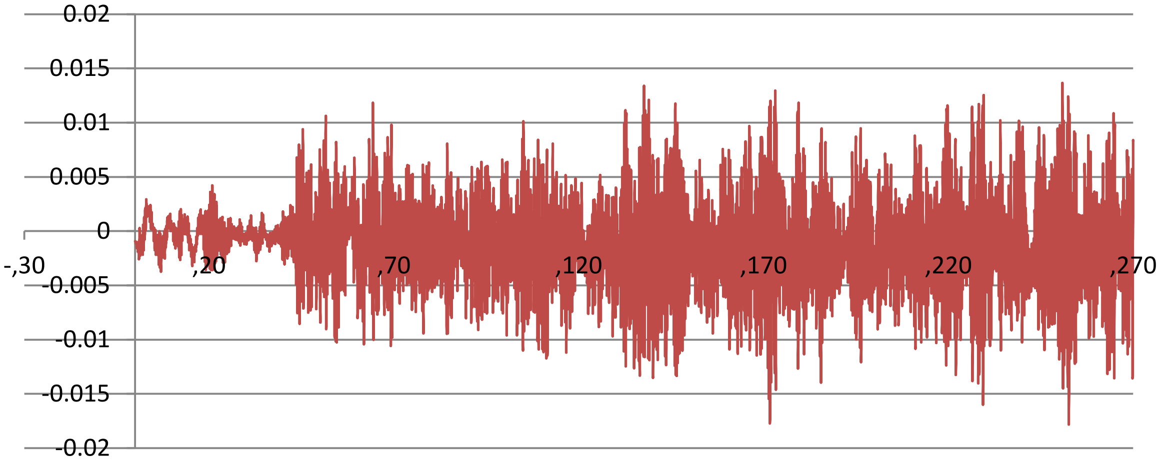


11.5.3 Bracer displacement

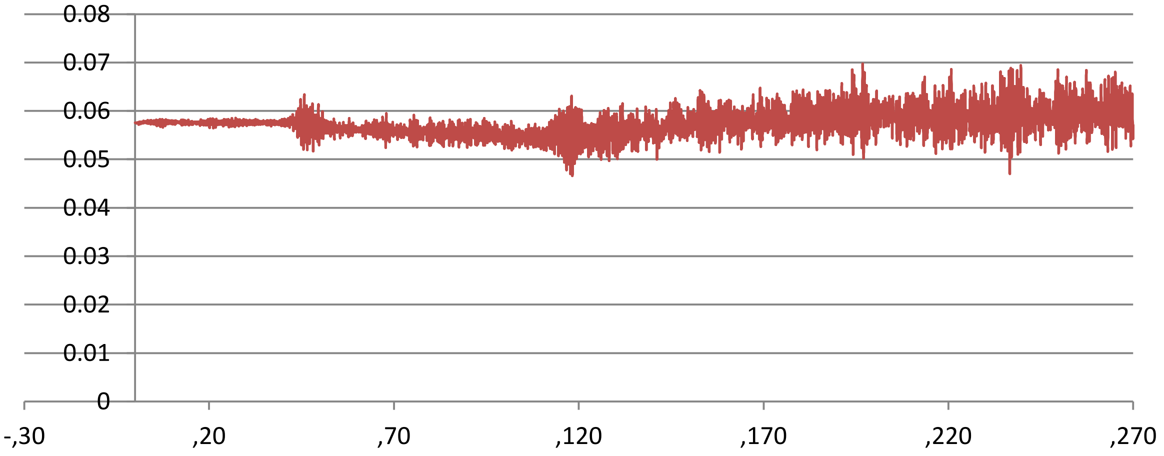
Upper Junction



Mid Junction

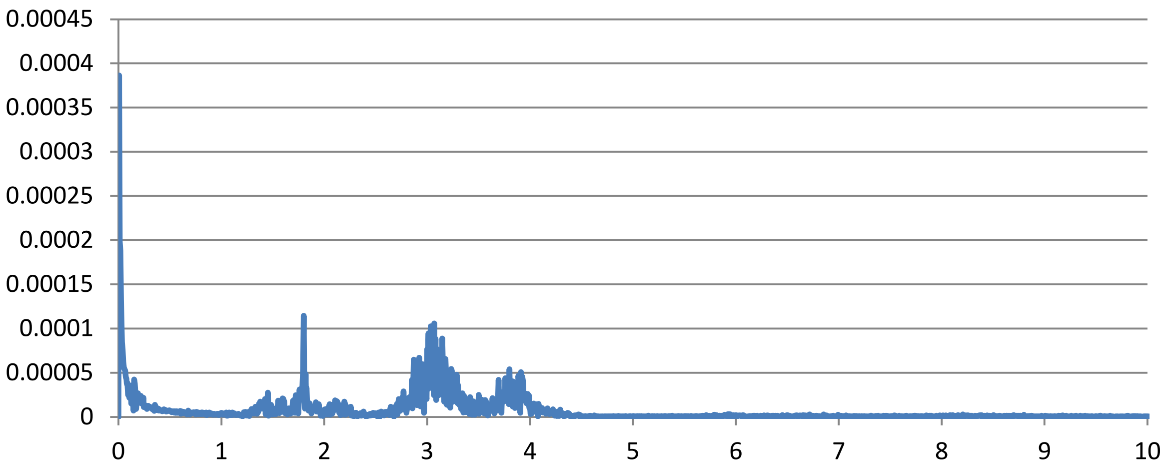


Lwr Junction

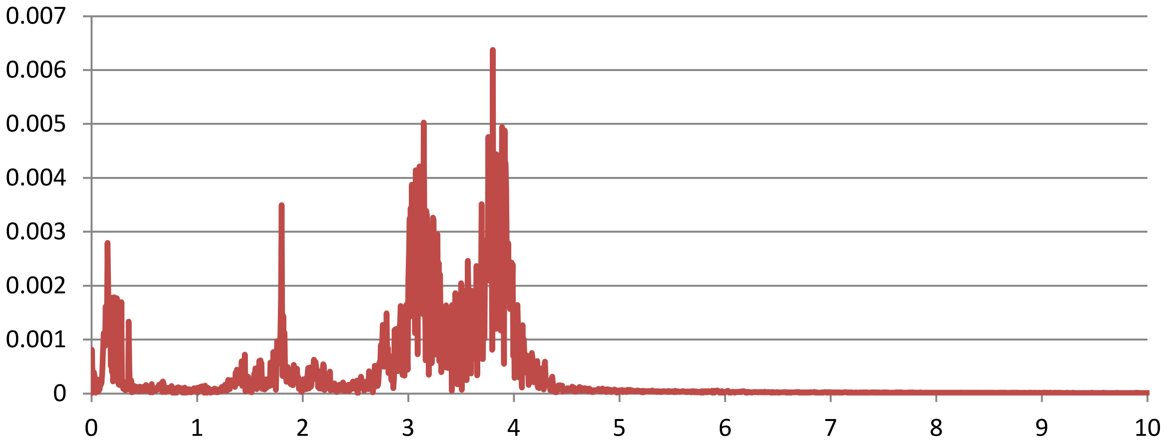


11.5.4 Bracer FFT Analysis

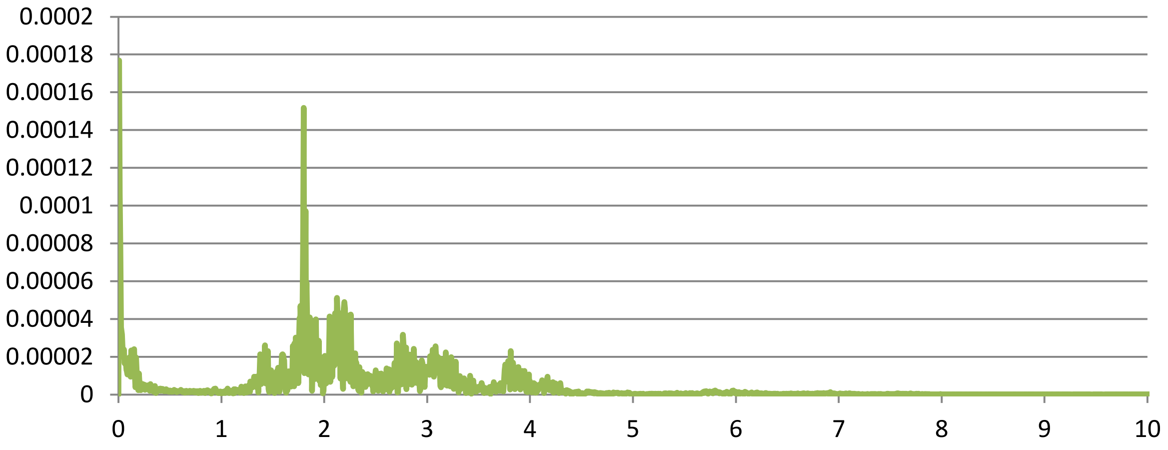
Upper Junction



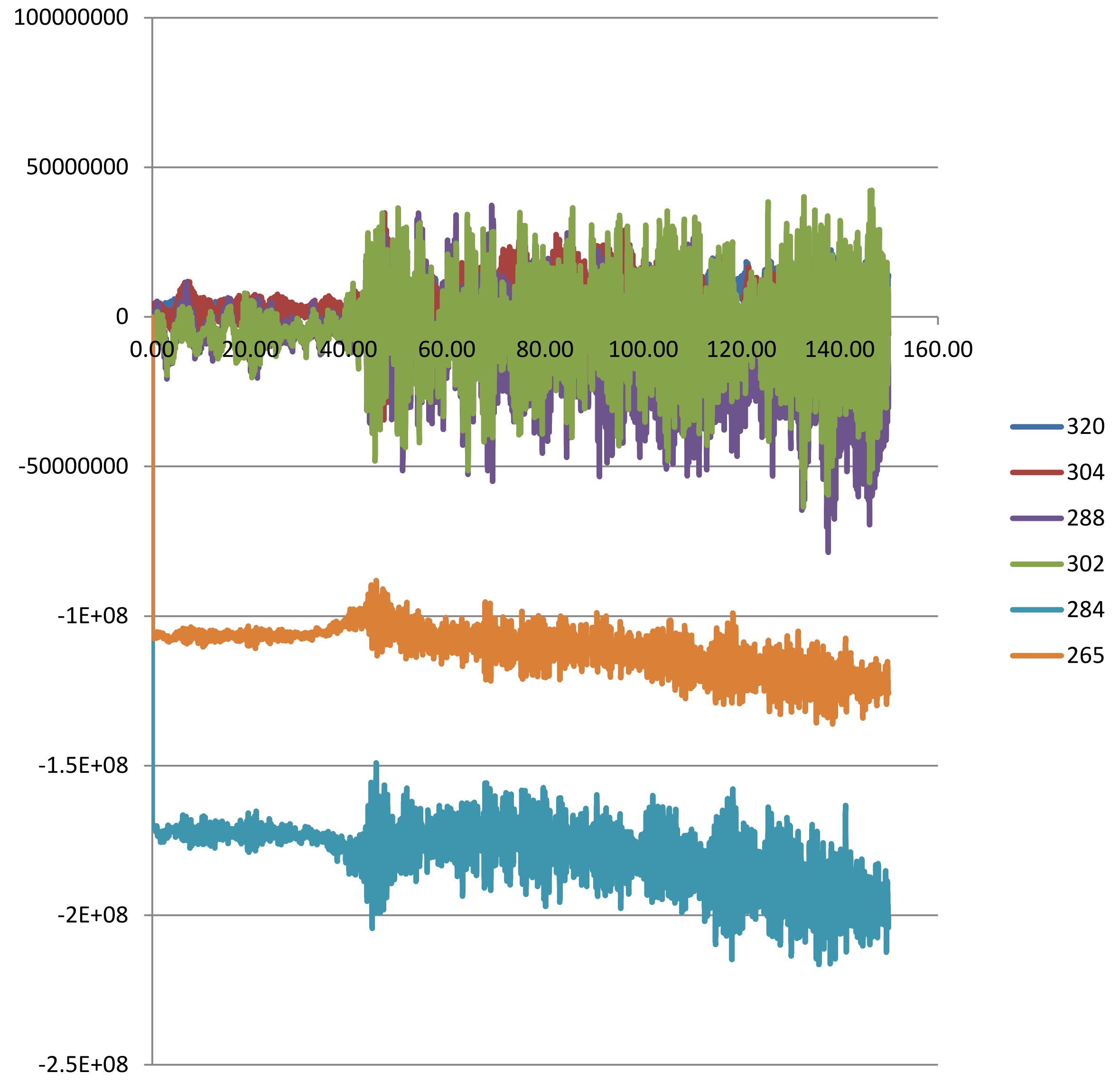
Mid Junction



Lwr Junction



11.5.5 Beam stresses



Max Beam Stresses

320	**304**	**302**	**288**	**284**	**265**	
4.17E+07	5.19E+07	5.43E+07	3.73E+07	2.84E+02	2.65E+02	*[Pa]*
41.700	51.936	54.252	37.274	0.000	0.000	*[MPa]*

Min Beam Stresses

320	**304**	**302**	**288**	**284**	**265**	
-1.46E+07	-3.45E+07	-6.35E+07	-6.46E+07	-2.15E+08	-1.33E+08	*[Pa]*
-14.573	-34.473	-63.483	-64.554	-214.839	-132.835	*[MPa]*

Delta sigma

320	**304**	**302**	**288**	**284**	**265**	
56.273	86.409	117.735	101.828	214.840	132.835	*[MPa]*

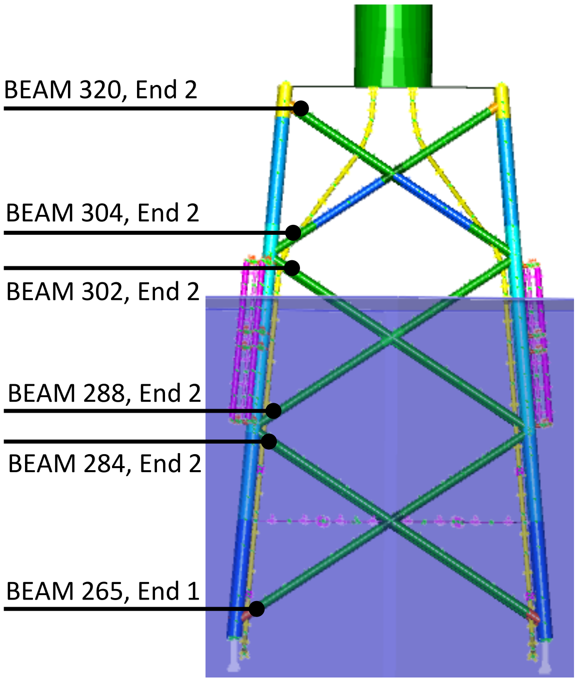
11.6 Sequentially Coupled Analysis- Event 3-All DOF Free-Usfos/Fedem

11.6.1 Description

- Fully coupled Usfos/Fedem analysis
- K, C, and M matrices replacing the interface node for Fedem analysis
    - For all DOF
- Linear springs along soil piles for soil properties
- Turbine setup in Fedem according to “Definition of a 5-MW Reference Wind Turbine for Offshore System Development”
    - J. Jonkman, S. Butterfield, W. Musial, and G. Scott
- Event 3

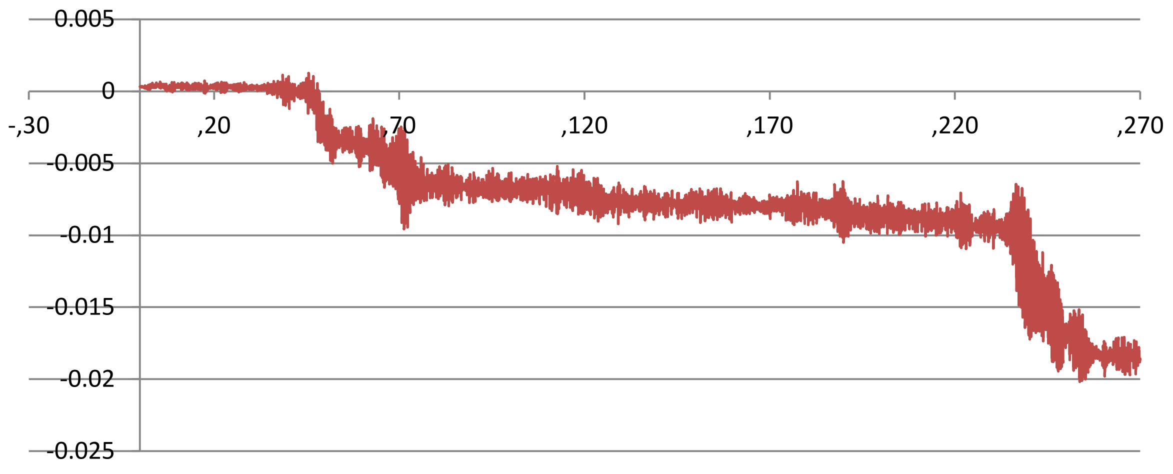
11.6.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	1.396E+02
Stress range	$\Delta\sigma$ [MPa]	1.877E+02
Damage		1.184E-04
Beam 304		
Max stress	σ_{max} [MPa]	1.574E+02
Stress range	$\Delta\sigma$ [MPa]	2.559E+02
Damage		2.591E-04
Beam 302		
Max stress	σ_{max} [MPa]	1.445E+02
Stress range	$\Delta\sigma$ [MPa]	2.929E+02
Damage		8.414E-04
Beam 288		
Max stress	σ_{max} [MPa]	9.971E+01
Stress range	$\Delta\sigma$ [MPa]	2.507E+02
Damage		1.363E-04
Beam 284		
Max stress	σ_{max} [MPa]	2.840E-04
Stress range	$\Delta\sigma$ [MPa]	3.791E+02
Damage		6.842E-05
Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	2.525E+02
Damage		3.202E-05

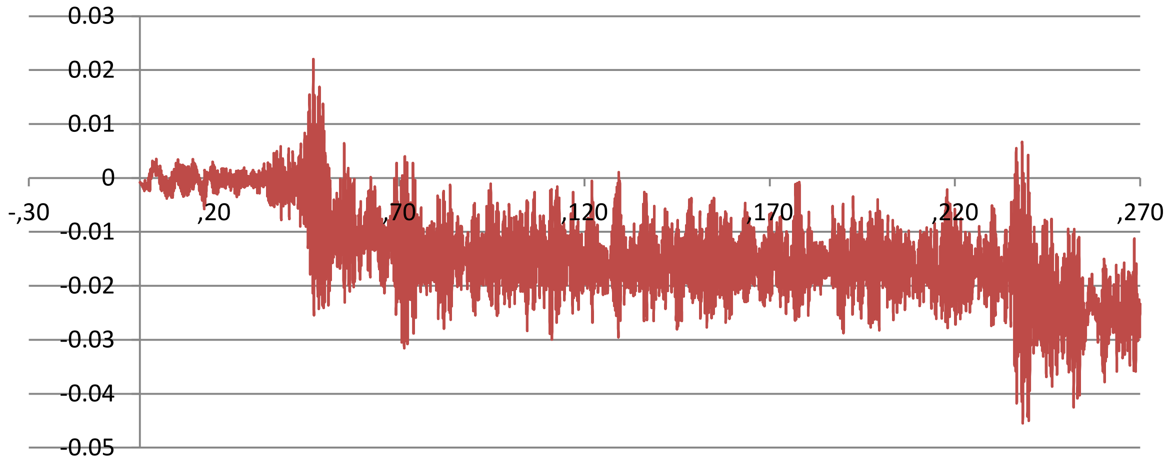


11.6.3 Bracer displacement

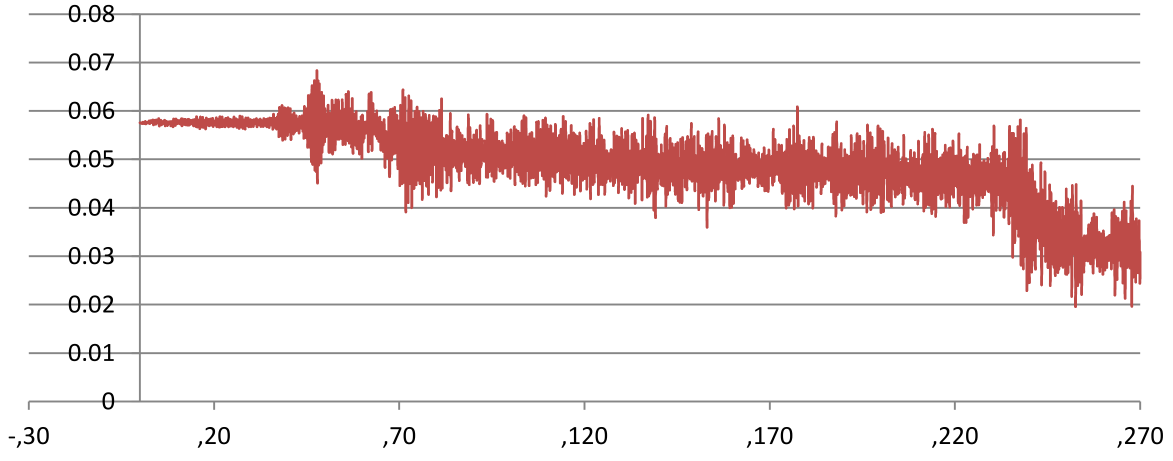
Upper Junction



Mid Junction

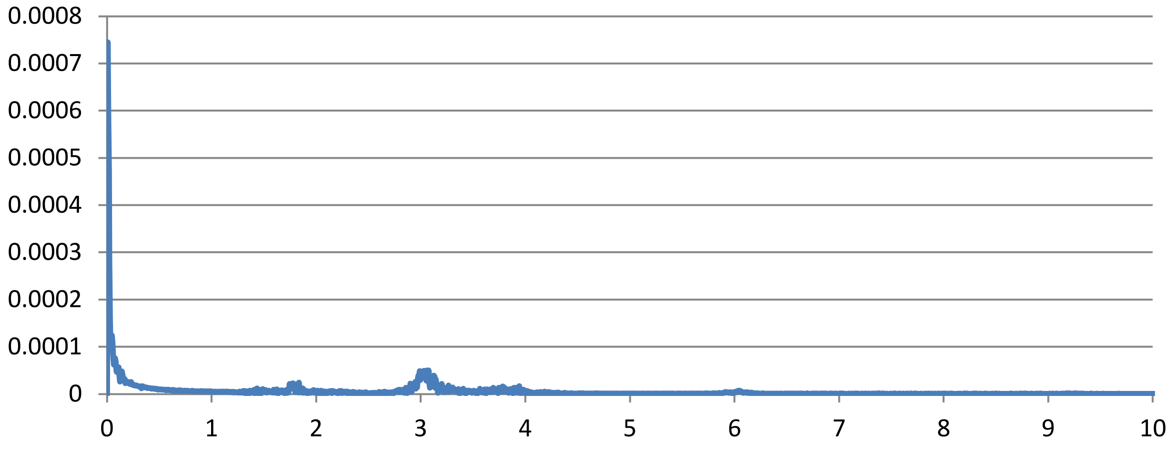


Lwr Junction

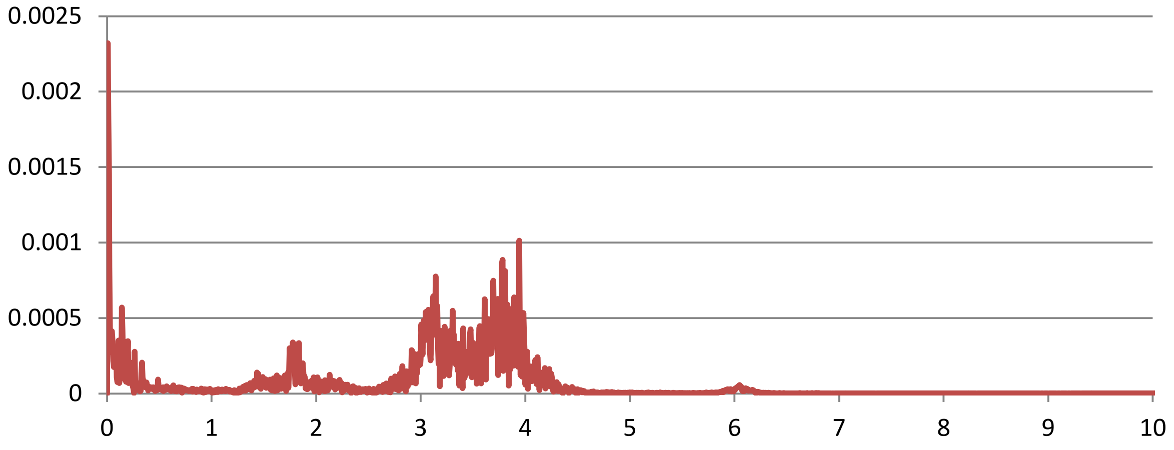


11.6.4 Bracer FFT Analysis

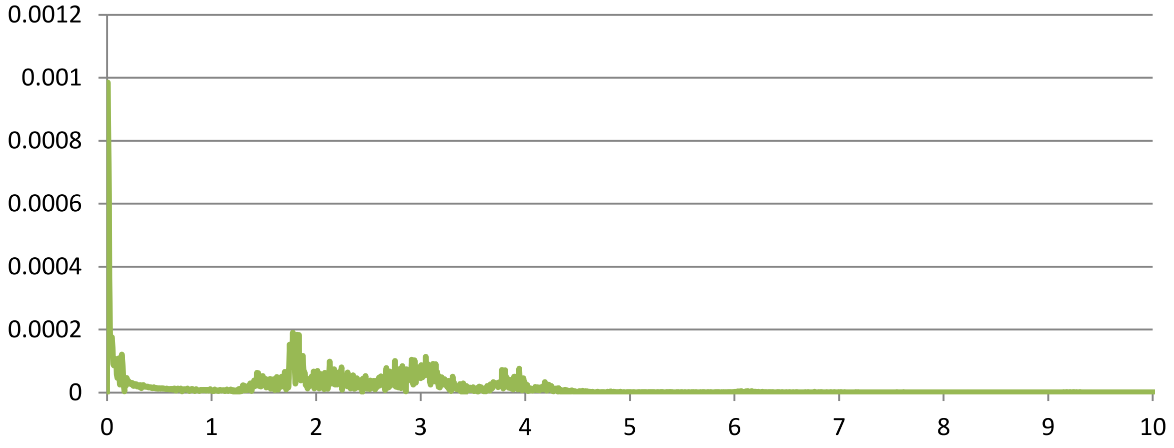
Upper Junction



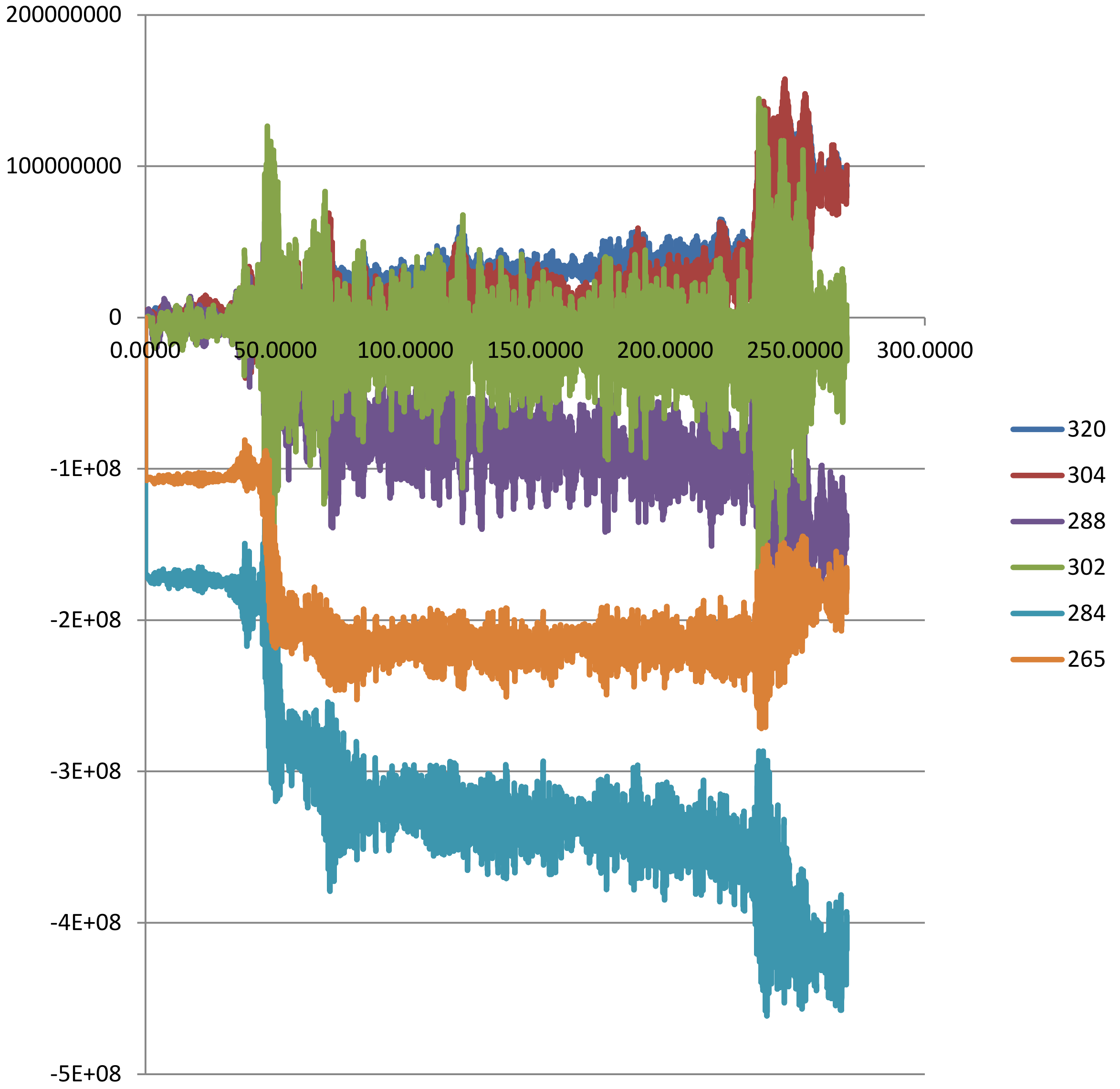
Mid Junction



Lwr Junction



11.6.5 Beam stresses



Max Beam Stresses

320	304	302	288	284	265	
1.40E+08	1.57E+08	1.45E+08	9.97E+07	2.84E+02	2.65E+02	*[Pa]*
139.588	157.448	144.533	99.711	0.000	0.000	*[MPa]*

Min Beam Stresses

320	304	302	288	284	265	
-4.81E+07	-9.85E+07	-1.48E+08	-1.51E+08	-3.79E+08	-2.52E+08	*[Pa]*
-48.093	-98.496	-148.395	-150.958	-379.117	-252.471	*[MPa]*

Delta sigma

320	304	302	288	284	265	
187.681	255.944	292.928	250.669	379.117	252.471	*[MPa]*

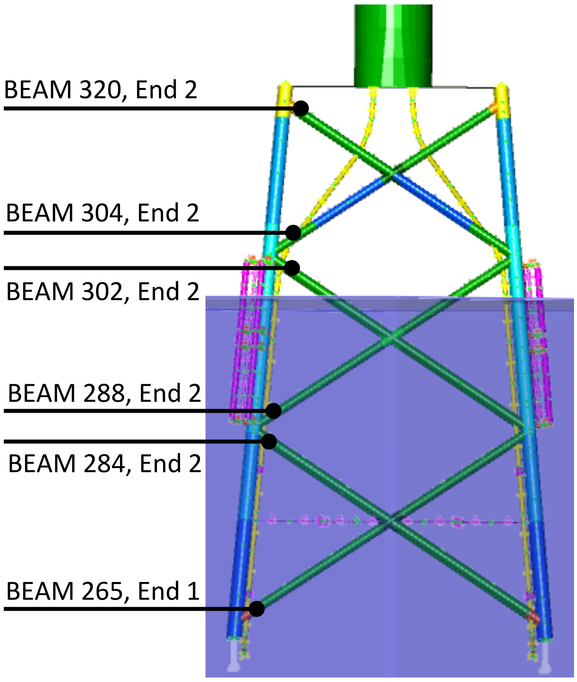
11.7 Sequentially Coupled Analysis- Event 1-DOF z const. -Usfos/Fedem

11.7.1 Description

- Fully coupled Usfos/Fedem analysis
- K, C, and M matrices replacing the interface node for Fedem analysis
    - DOF z is constrained in Fedem-Analysis
    - Tower and nacelle lumped mass in z direction for Usfos stress recovery analysis
- Linear springs along soil piles for soil properties
- Turbine setup in Fedem according to “Definition of a 5-MW Reference Wind Turbine for Offshore System Development”
    - J. Jonkman, S. Butterfield, W. Musial, and G. Scott
- Event 1

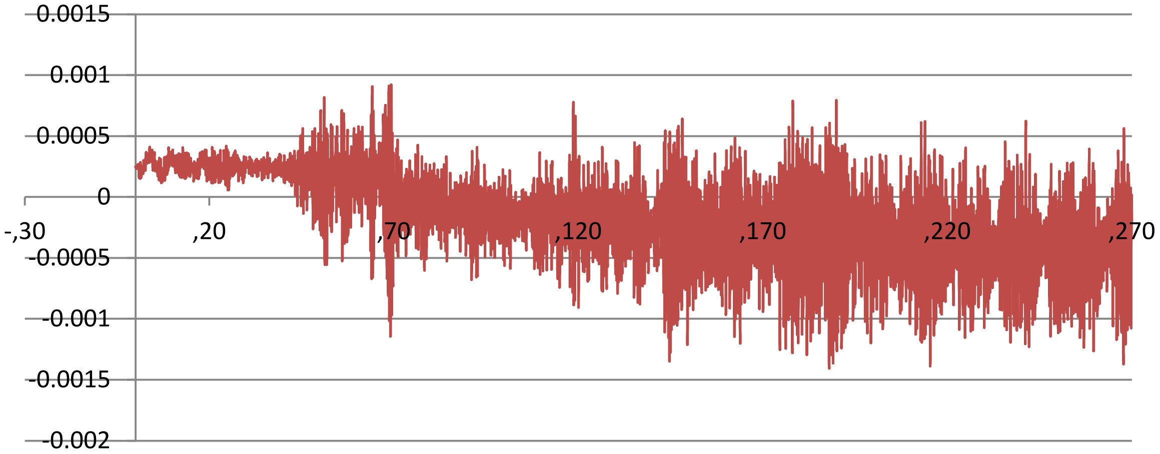
11.7.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	2.089E+01
Stress range	$\Delta\sigma$ [MPa]	3.602E+01
Damage		1.182E-06
Beam 304		
Max stress	σ_{max} [MPa]	3.334E+01
Stress range	$\Delta\sigma$ [MPa]	5.960E+01
Damage		3.301E-06
Beam 302		
Max stress	σ_{max} [MPa]	5.208E+01
Stress range	$\Delta\sigma$ [MPa]	1.186E+02
Damage		1.315E-04
Beam 288		
Max stress	σ_{max} [MPa]	4.653E+01
Stress range	$\Delta\sigma$ [MPa]	1.097E+02
Damage		7.081E-05
Beam 284		
Max stress	σ_{max} [MPa]	2.840E-04
Stress range	$\Delta\sigma$ [MPa]	2.089E+02
Damage		8.428E-06
Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	1.287E+02
Damage		1.631E-06

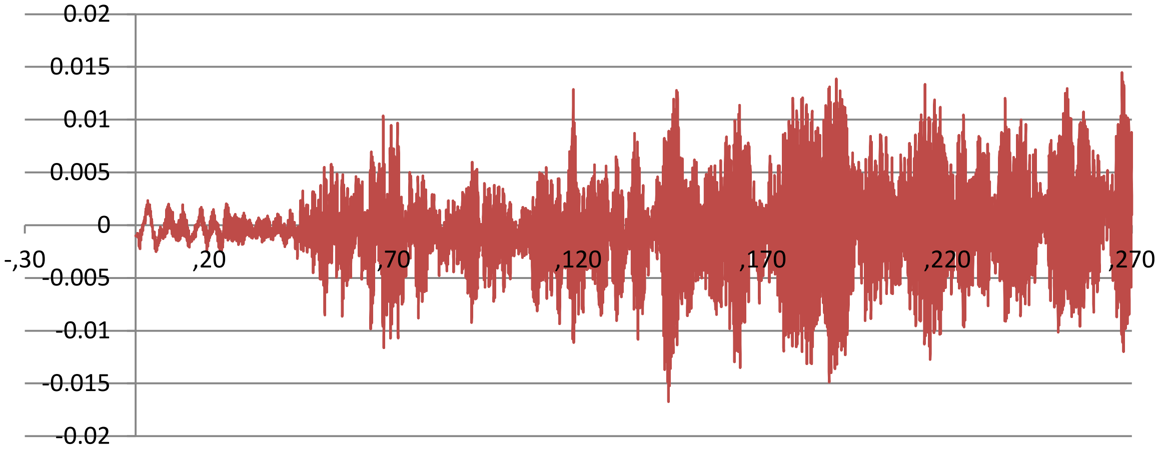


11.7.3 Bracer displacement

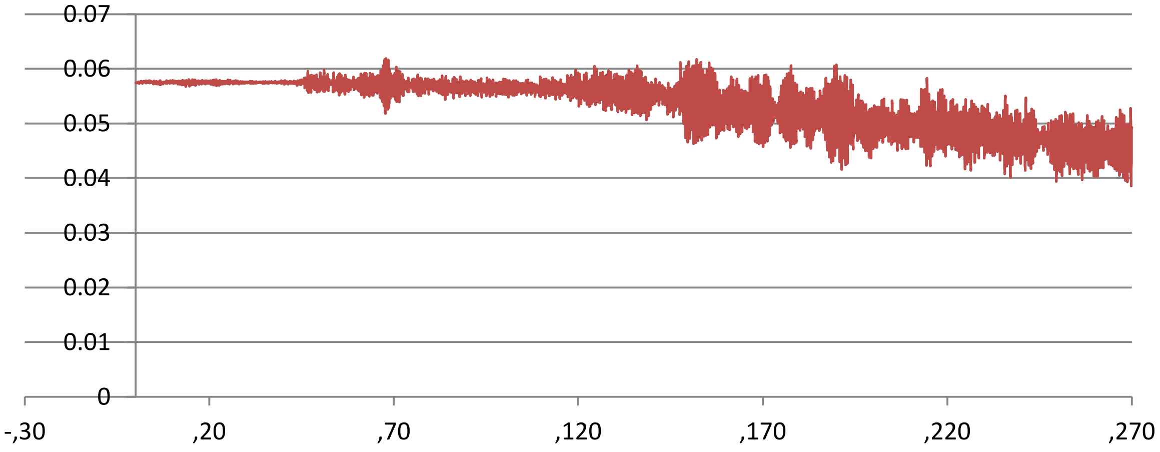
Upper Junction



Mid Junction

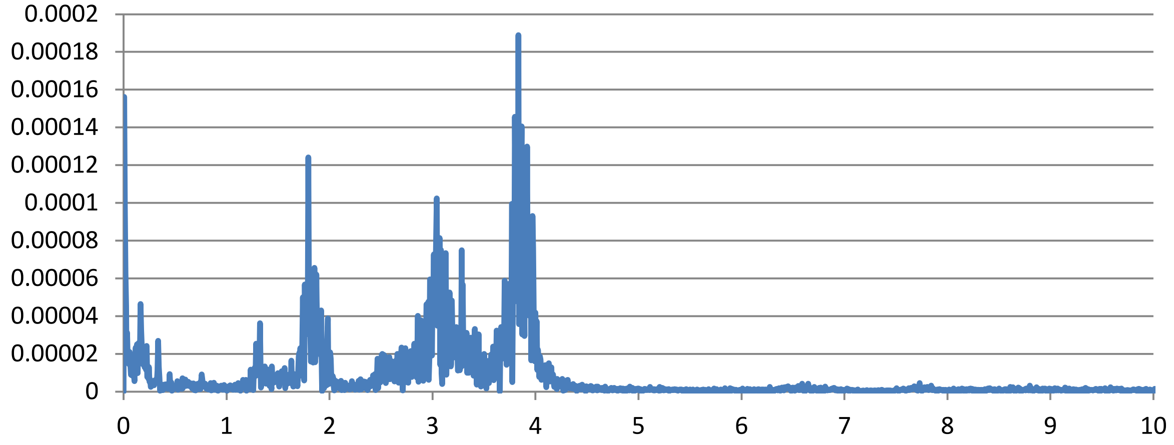


Lwr Junction

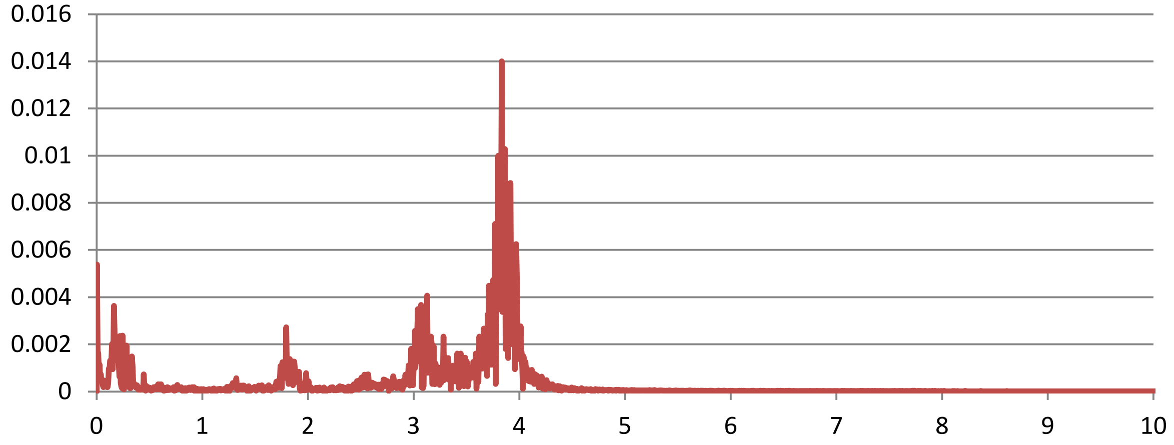


11.7.4 Bracer FFT Analysis

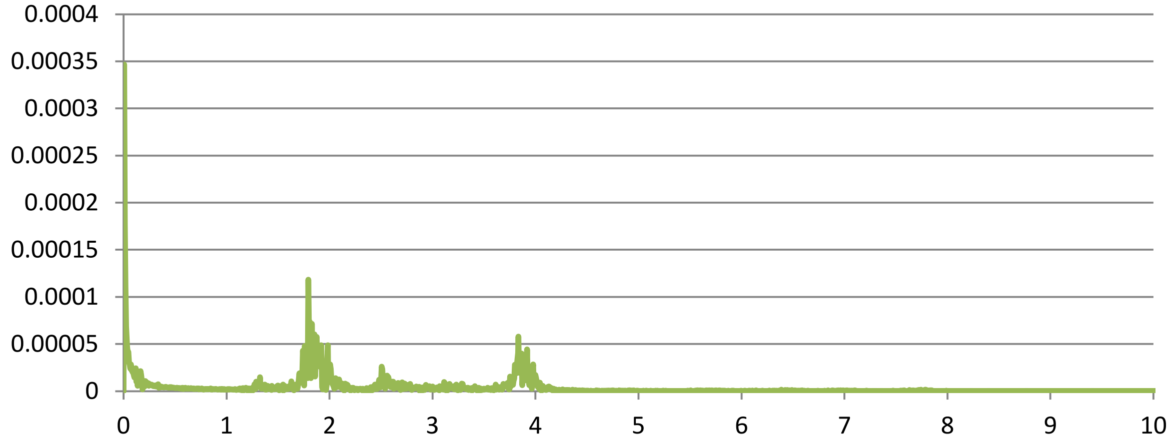
Upper Junction



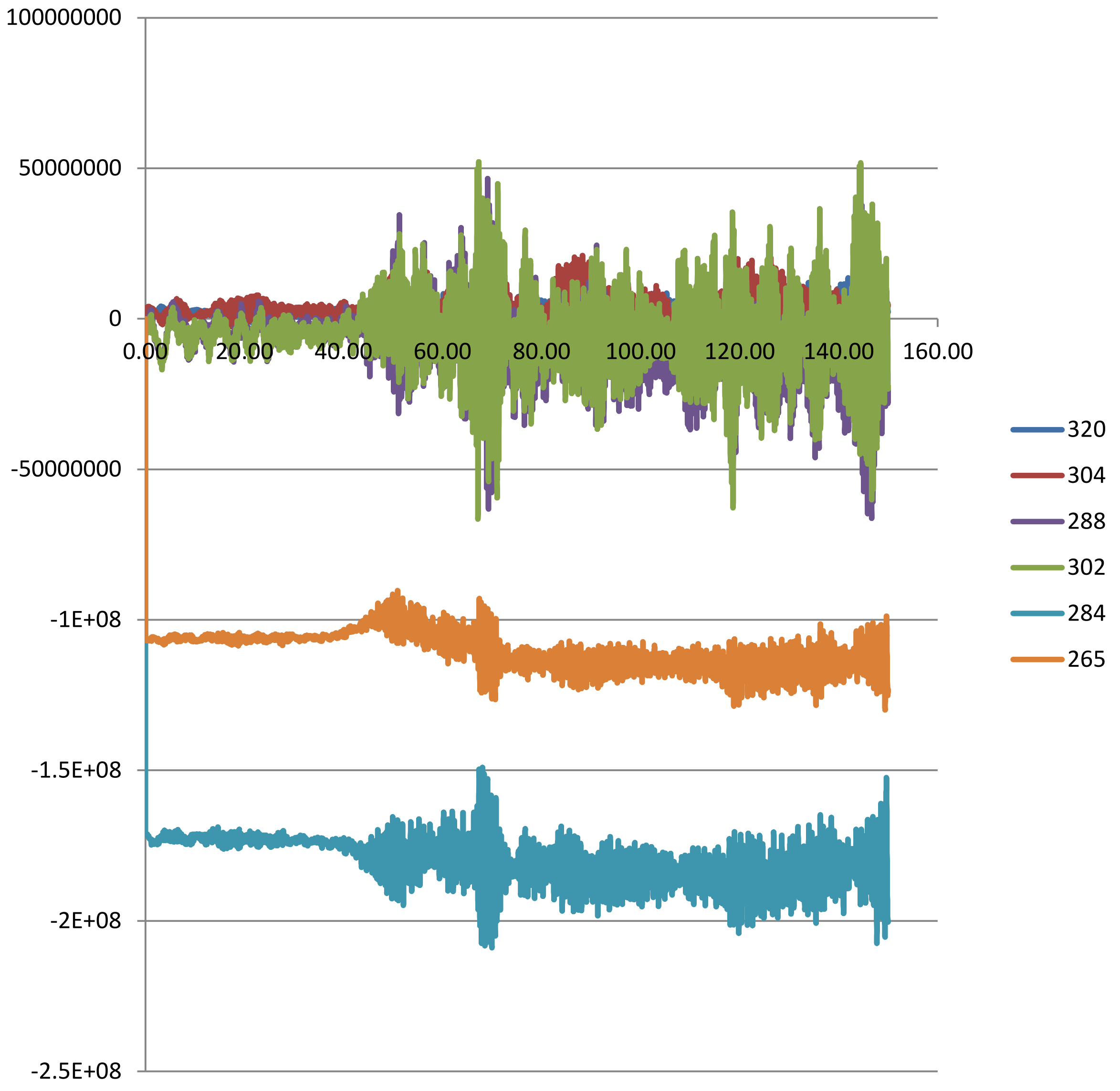
Mid Junction



Lwr Junction



11.7.5 Beam stresses



Max Beam Stresses

320	**304**	**302**	**288**	**284**	**265**	
2.09E+07	3.33E+07	5.21E+07	4.65E+07	2.84E+02	2.65E+02	*[Pa]*
20.890	33.343	52.079	46.529	0.000	0.000	*[MPa]*

Min Beam Stresses

320	**304**	**302**	**288**	**284**	**265**	
-1.51E+07	-2.63E+07	-6.65E+07	-6.32E+07	-2.09E+08	-1.29E+08	*[Pa]*
-15.133	-26.256	-66.525	-63.196	-208.916	-128.677	*[MPa]*

Delta sigma

320	**304**	**302**	**288**	**284**	**265**	
36.023	59.599	118.604	109.725	208.916	128.677	*[MPa]*

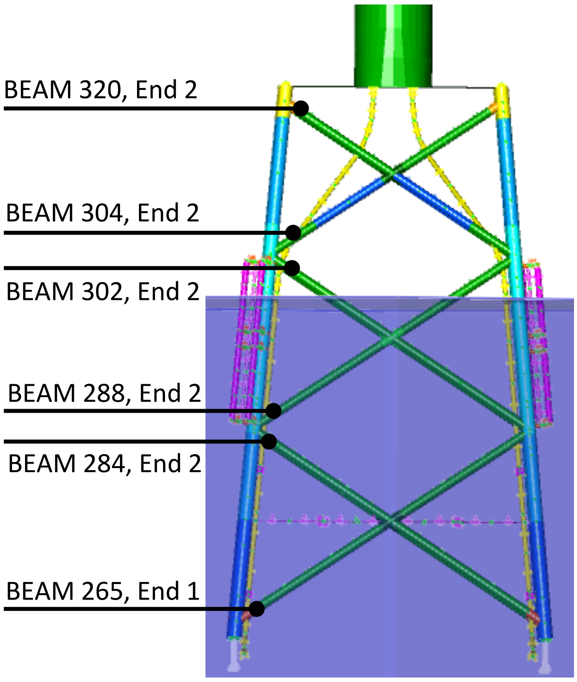
11.8 Sequentially Coupled Analysis- Event 2-DOF z const. -Usfos/Fedem

11.8.1 Description

- Fully coupled Usfos/Fedem analysis
- K, C, and M matrices replacing the interface node for Fedem analysis
    - DOF z is constrained in Fedem-Analysis
    - Tower and nacelle lumped mass in z direction for Usfos stress recovery analysis
- Linear springs along soil piles for soil properties
- Turbine setup in Fedem according to “Definition of a 5-MW Reference Wind Turbine for Offshore System Development”
    - J. Jonkman, S. Butterfield, W. Musial, and G. Scott
- Event 2

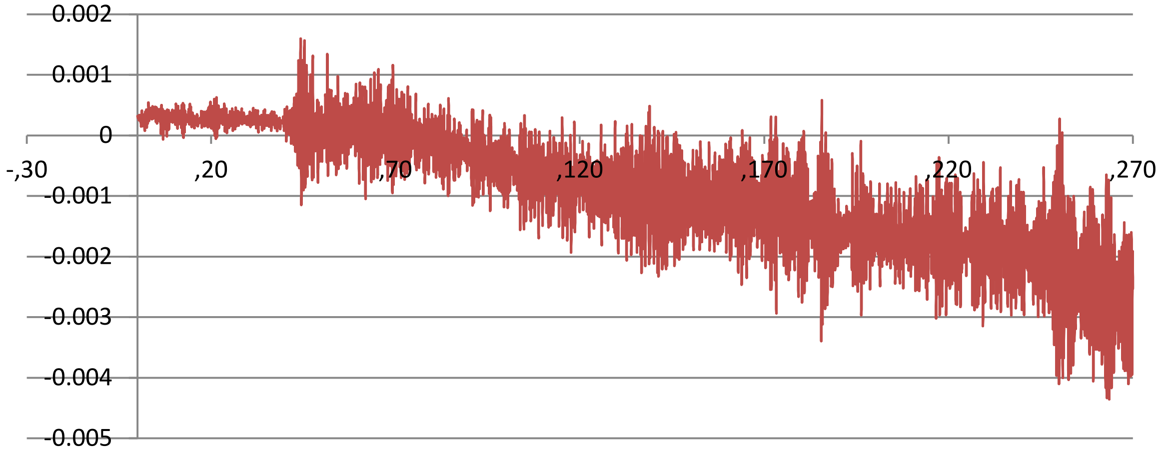
11.8.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	4.244E+01
Stress range	$\Delta\sigma$ [MPa]	5.706E+01
Damage		4.584E-06
Beam 304		
Max stress	σ_{max} [MPa]	5.379E+01
Stress range	$\Delta\sigma$ [MPa]	8.855E+01
Damage		1.218E-05
Beam 302		
Max stress	σ_{max} [MPa]	6.147E+01
Stress range	$\Delta\sigma$ [MPa]	1.382E+02
Damage		1.438E-04
Beam 288		
Max stress	σ_{max} [MPa]	4.296E+01
Stress range	$\Delta\sigma$ [MPa]	1.139E+02
Damage		9.602E-05
Beam 284		
Max stress	σ_{max} [MPa]	2.840E-04
Stress range	$\Delta\sigma$ [MPa]	2.048E+02
Damage		1.775E-05
Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	1.224E+02
Damage		4.176E-06

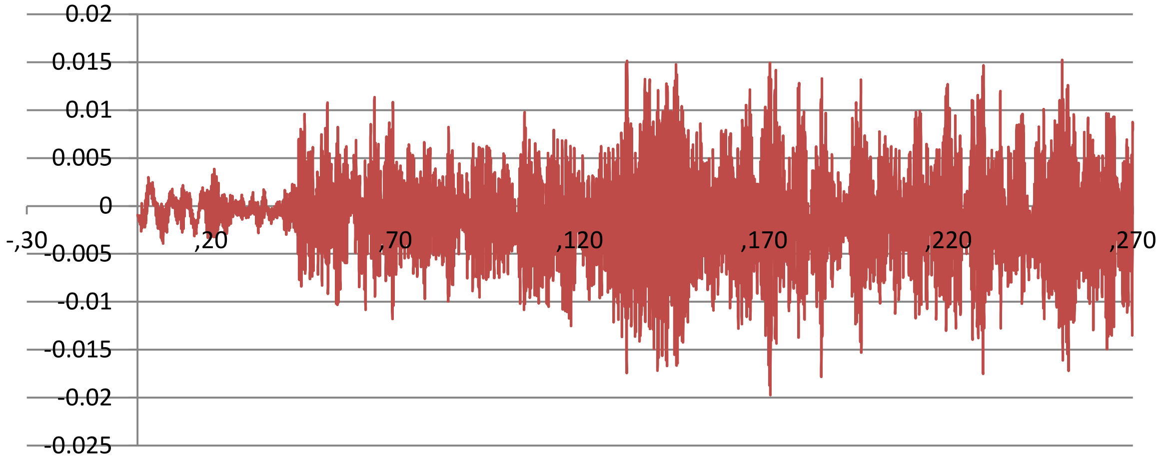


11.8.3 Bracer displacement

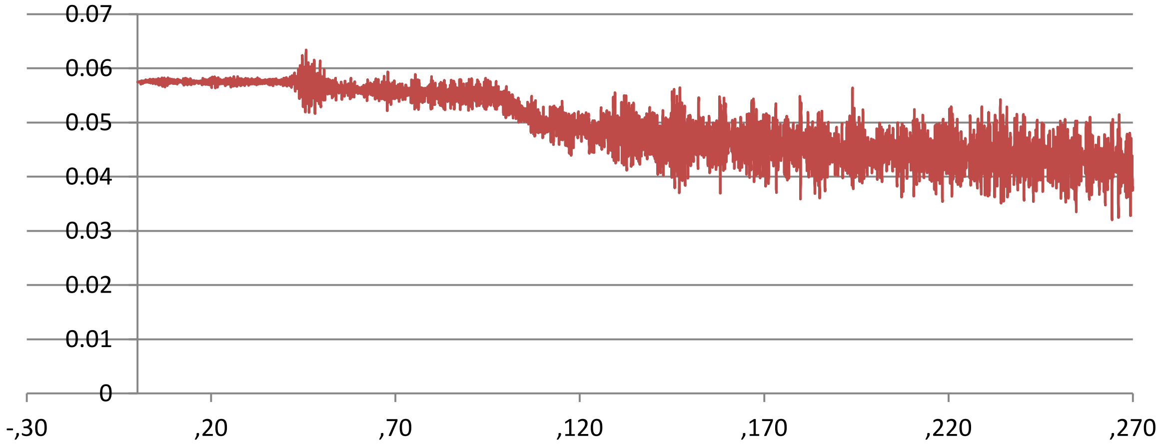
Upper Junction



Mid Junction

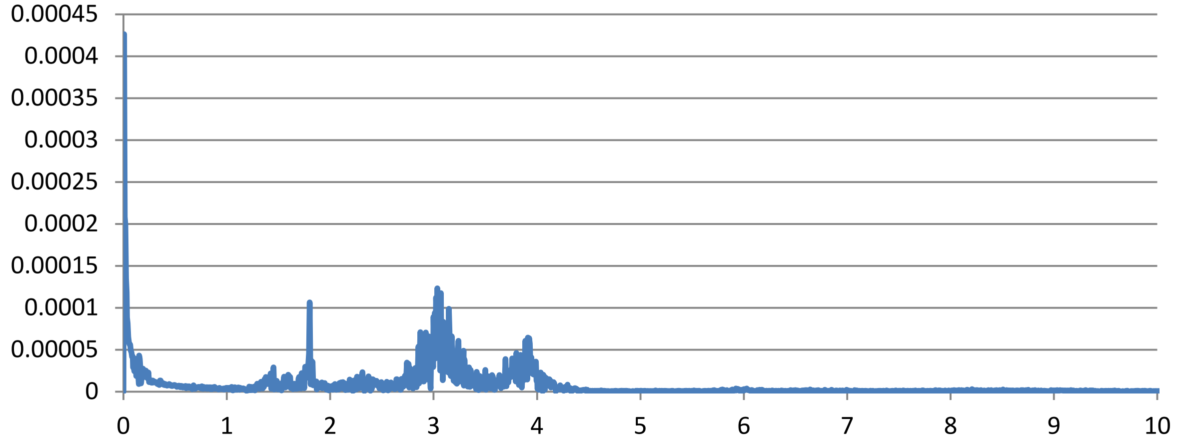


Lwr Junction

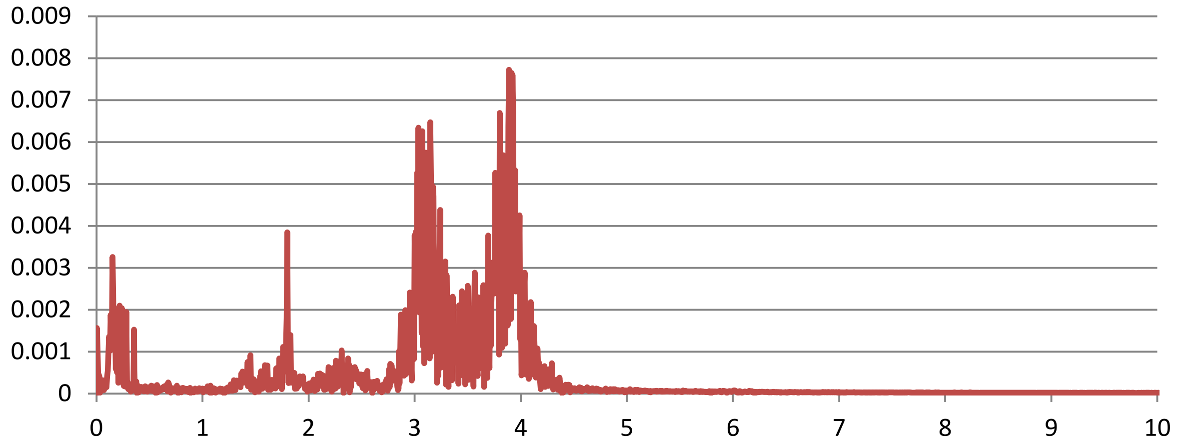


11.8.4 Bracer FFT Analysis

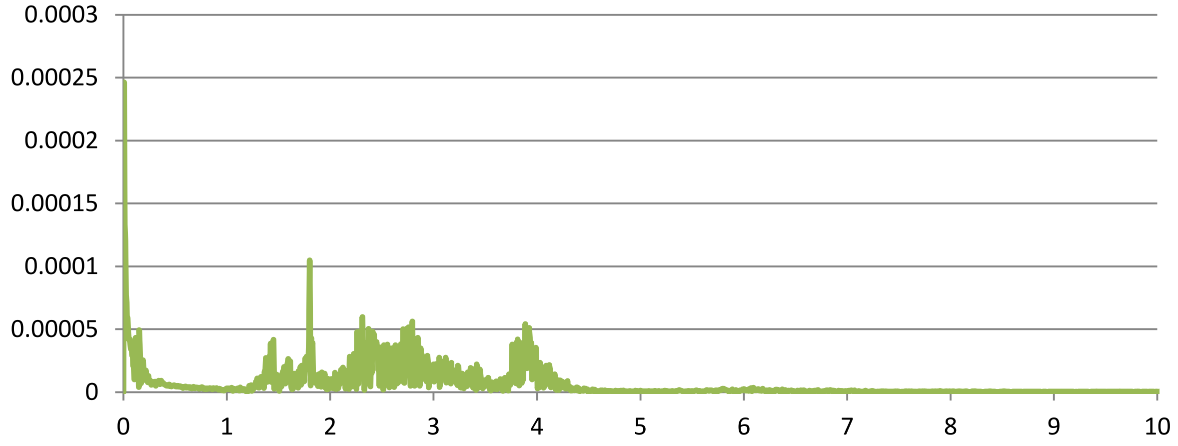
Upper Junction



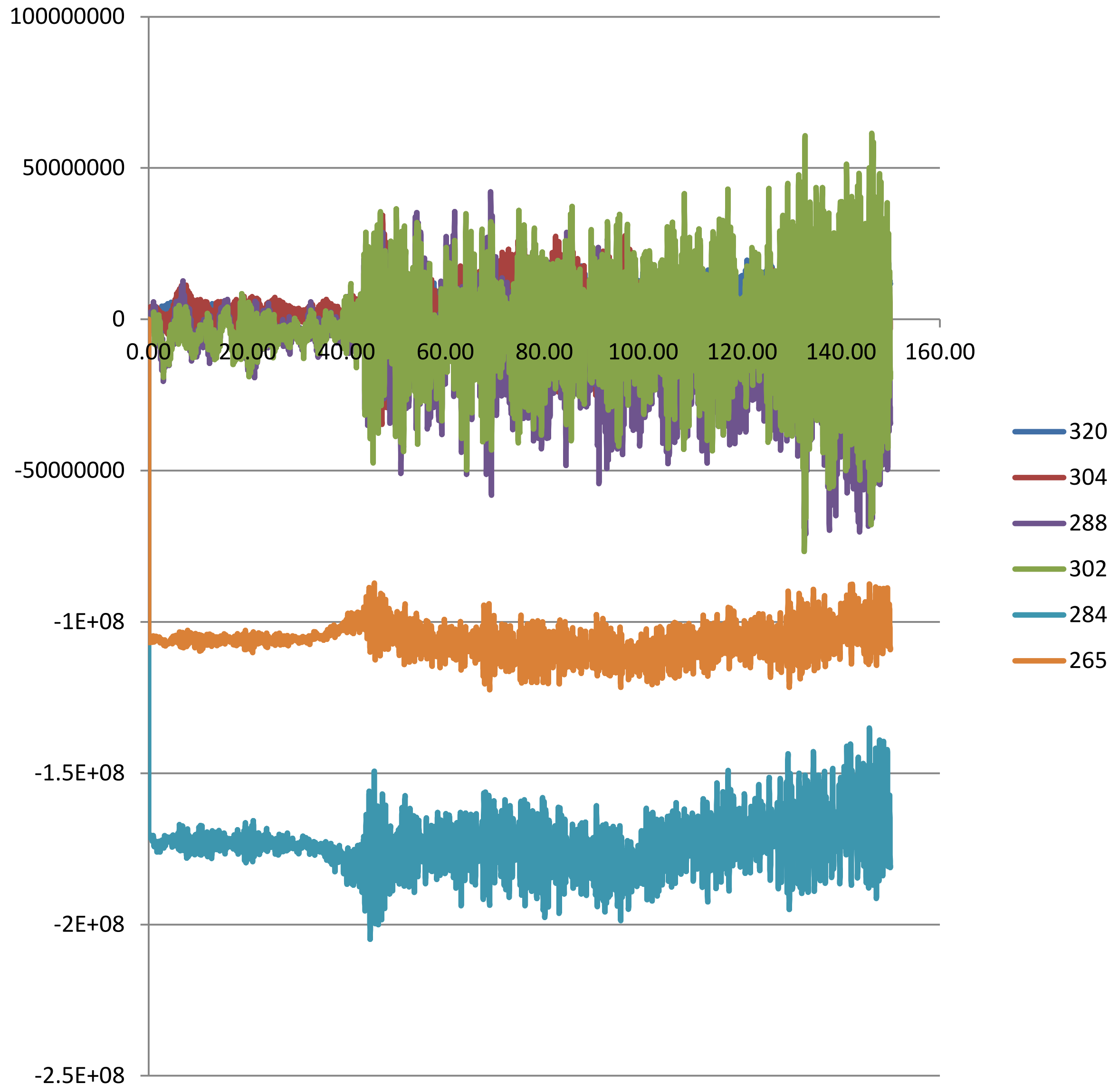
Mid Junction



Lwr Junction



11.8.5 Beam stresses



Max Beam Stresses

320	304	302	288	284	265	
4.24E+07	5.38E+07	6.15E+07	4.30E+07	2.84E+02	2.65E+02	*[Pa]*
42.437	53.790	61.471	42.955	0.000	0.000	*[MPa]*

Min Beam Stresses

320	304	302	288	284	265	
-1.46E+07	-3.48E+07	-7.67E+07	-7.10E+07	-2.05E+08	-1.22E+08	*[Pa]*
-14.624	-34.757	-76.701	-70.961	-204.816	-122.409	*[MPa]*

Delta sigma

320	304	302	288	284	265	
57.062	88.546	138.172	113.916	204.816	122.409	*[MPa]*

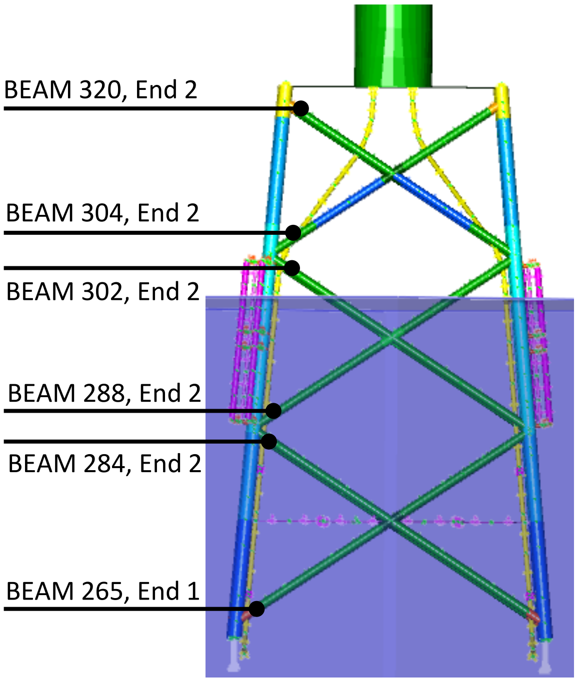
11.9 Sequentially Coupled Analysis- Event 3-DOF z const. -Usfos/Fedem

11.9.1 Description

- Fully coupled Usfos/Fedem analysis
- K, C, and M matrices replacing the interface node for Fedem analysis
  - DOF z is constrained in Fedem-Analysis
  - Tower and nacelle lumped mass in z direction for Usfos stress recovery analysis
- Linear springs along soil piles for soil properties
- Turbine setup in Fedem according to “Definition of a 5-MW Reference Wind Turbine for Offshore System Development”
  - J. Jonkman, S. Butterfield, W. Musial, and G. Scott
- Event 3

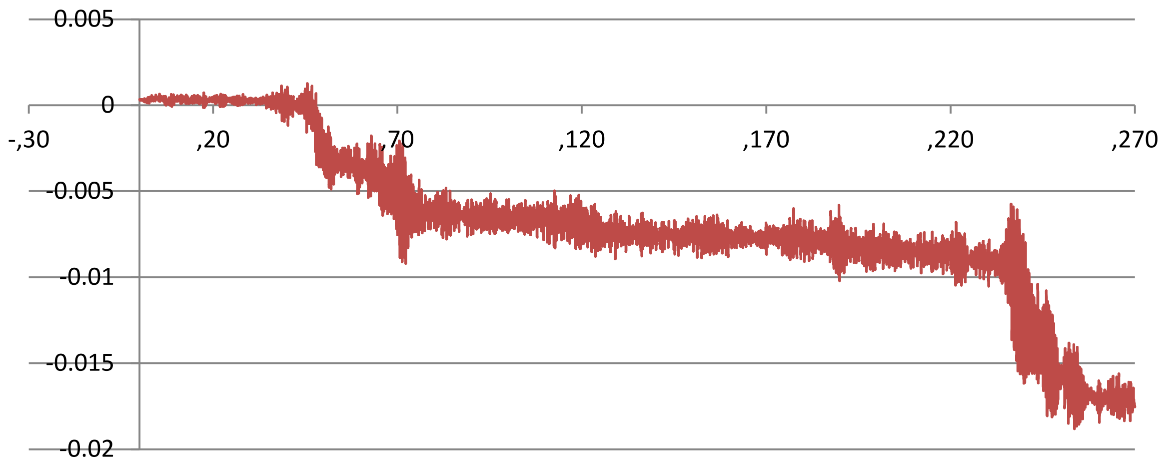
11.9.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	1.264E+02
Stress range	$\Delta\sigma$ [MPa]	1.751E+02
Damage		1.015E-04
Beam 304		
Max stress	σ_{max} [MPa]	1.402E+02
Stress range	$\Delta\sigma$ [MPa]	2.368E+02
Damage		2.180E-04
Beam 302		
Max stress	σ_{max} [MPa]	1.485E+02
Stress range	$\Delta\sigma$ [MPa]	2.945E+02
Damage		7.578E-04
Beam 288		
Max stress	σ_{max} [MPa]	1.017E+02
Stress range	$\Delta\sigma$ [MPa]	2.440E+02
Damage		1.384E-04
Beam 284		
Max stress	σ_{max} [MPa]	2.840E-04
Stress range	$\Delta\sigma$ [MPa]	3.796E+02
Damage		6.395E-05
Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	2.474E+02
Damage		3.088E-05

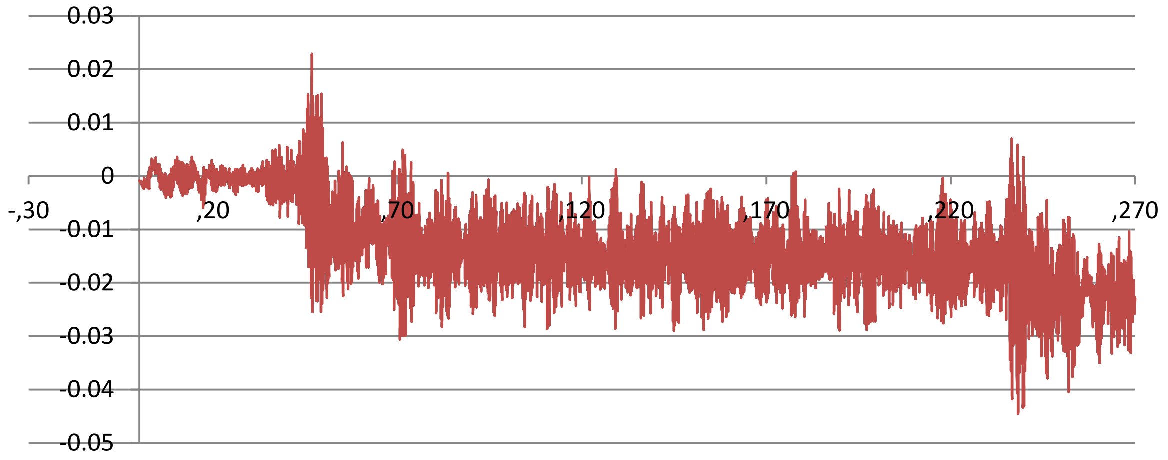


11.9.3 Bracer displacement

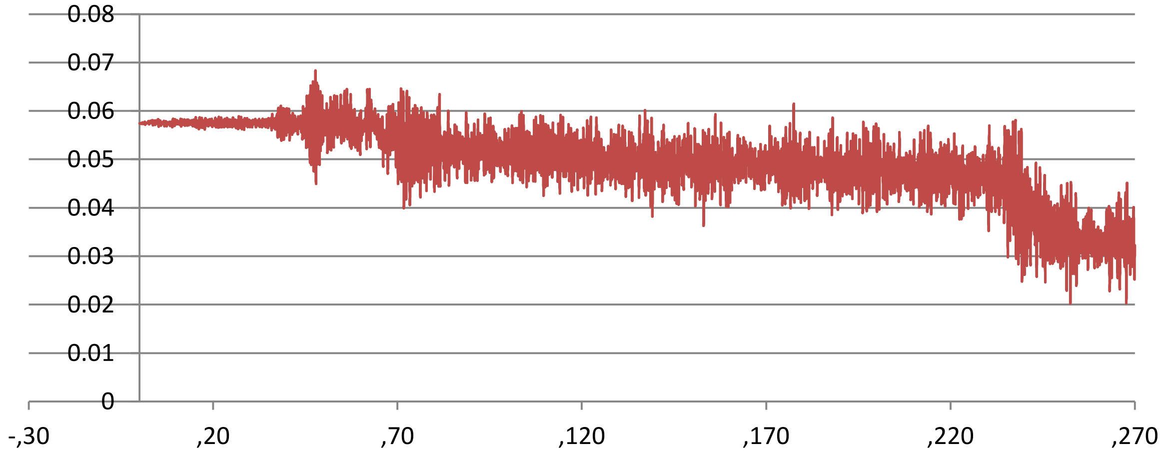
Upper Junction



Mid Junction

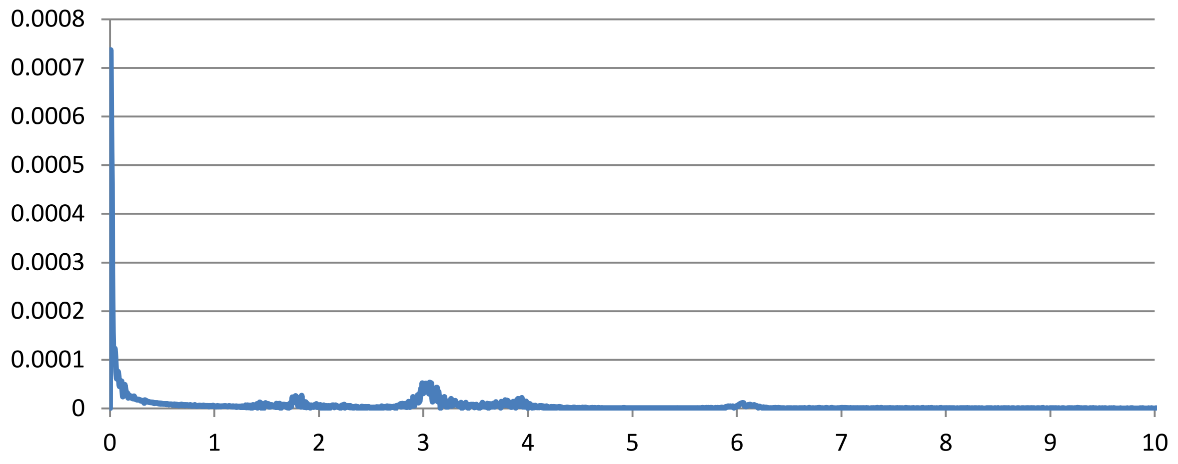


Lwr Junction

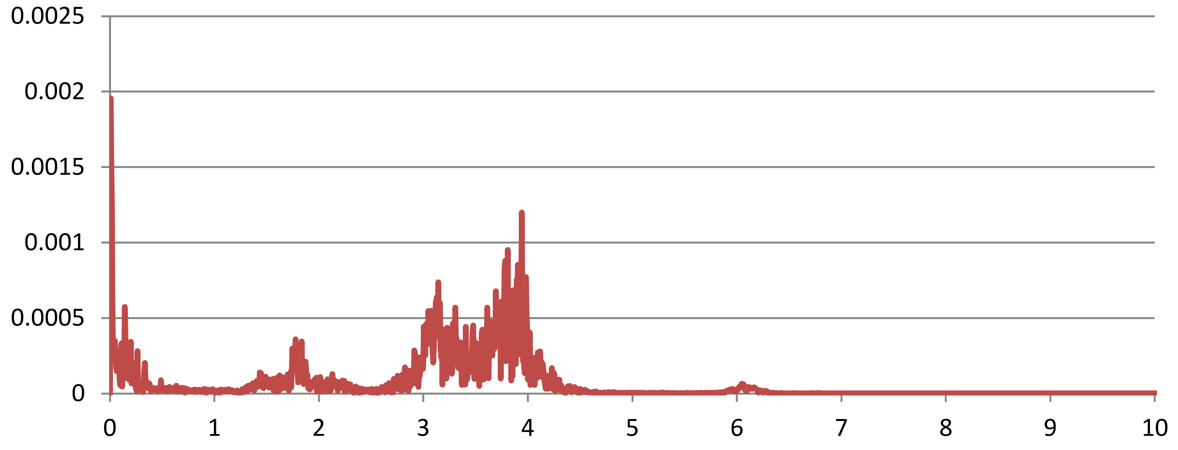


11.9.4 Bracer FFT Analysis

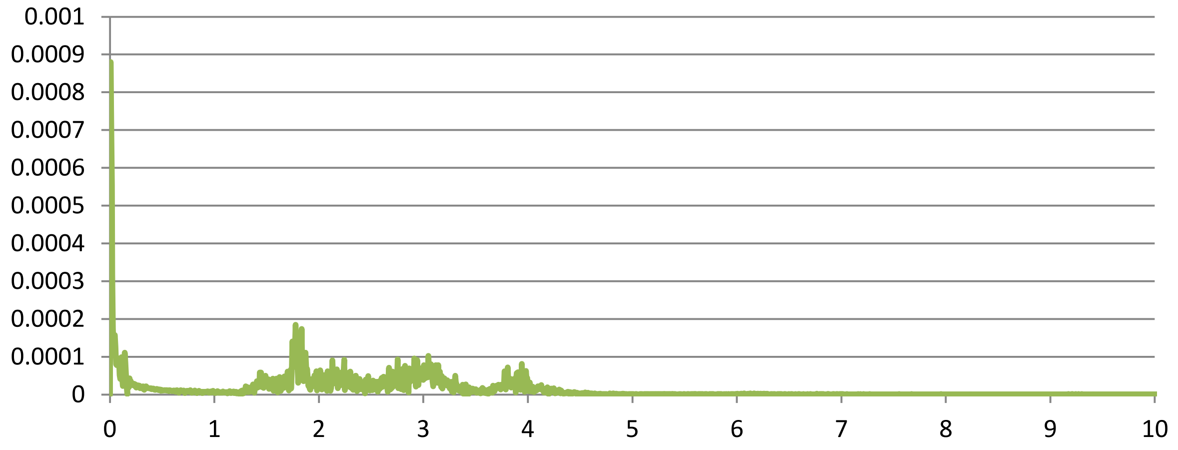
Upper Junction



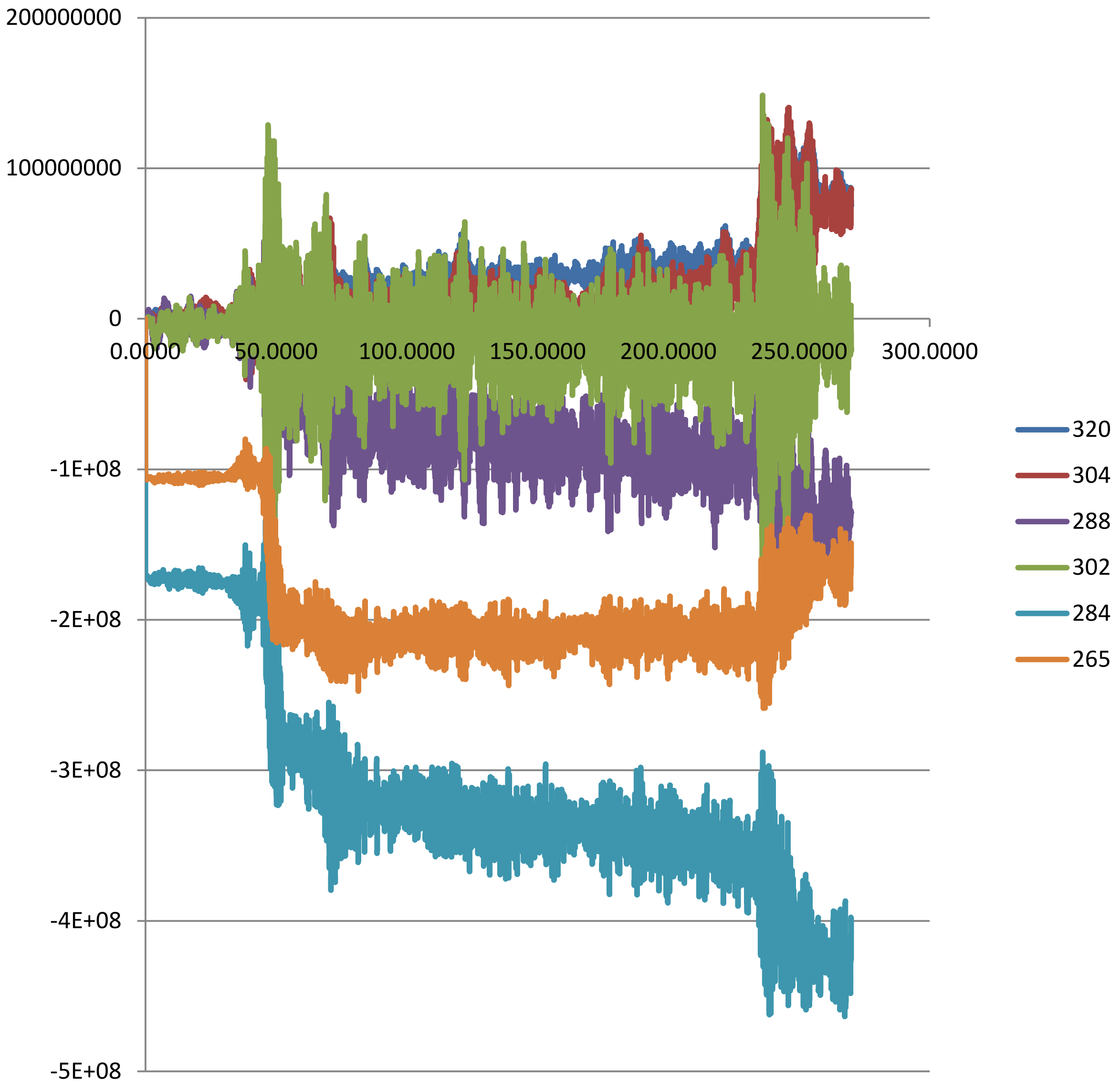
Mid Junction



Lwr Junction



11.9.5 Beam stresses



Max Beam Stresses

320	304	302	288	284	265	
1.26E+08	1.40E+08	1.48E+08	1.02E+08	2.84E+02	2.65E+02	*[Pa]*
126.445	140.240	148.475	101.746	0.000	0.000	*[MPa]*

Min Beam Stresses

320	304	302	288	284	265	
-4.86E+07	-9.66E+07	-1.46E+08	-1.42E+08	-3.80E+08	-2.47E+08	*[Pa]*
-48.625	-96.600	-145.977	-142.259	-379.564	-247.370	*[MPa]*

Delta sigma

320	304	302	288	284	265	
175.070	236.840	294.452	244.005	379.564	247.370	*[MPa]*

11.10 FEDEM stress recovery – FCA displacement input – Event 1

11.10.1 Description

- FEDEM stress recovery analysis with FEDEM Fully Coupled Analysis interface node displacement as input.
    - DOF z is constrained in FEDEM-Analysis
    - Tower and nacelle lumped mass in z direction for FEDEM stress recovery analysis
    - No windturbine, tower or nacelle
    - No Wind loads
- K, C, and M matrices replacing the soil piles properties
- Turbine setup in Fedem according to “Definition of a 5-MW Reference Wind Turbine for Offshore System Development”
    - J. Jonkman, S. Butterfield, W. Musial, and G. Scott
- Event 1

11.10.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	1.255E+01
Stress range	$\Delta\sigma$ [MPa]	1.922E+01
Damage		1.315E-08

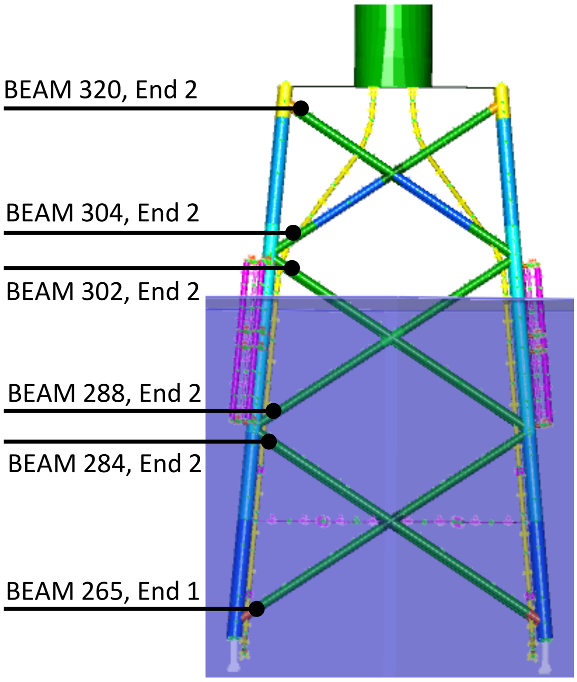
Beam 304		
Max stress	σ_{max} [MPa]	4.929E+00
Stress range	$\Delta\sigma$ [MPa]	3.039E+01
Damage		4.989E-07

Beam 302		
Max stress	σ_{max} [MPa]	1.856E+01
Stress range	$\Delta\sigma$ [MPa]	1.986E+01
Damage		5.958E-08

Beam 288		
Max stress	σ_{max} [MPa]	1.836E+01
Stress range	$\Delta\sigma$ [MPa]	3.805E+01
Damage		1.137E-06

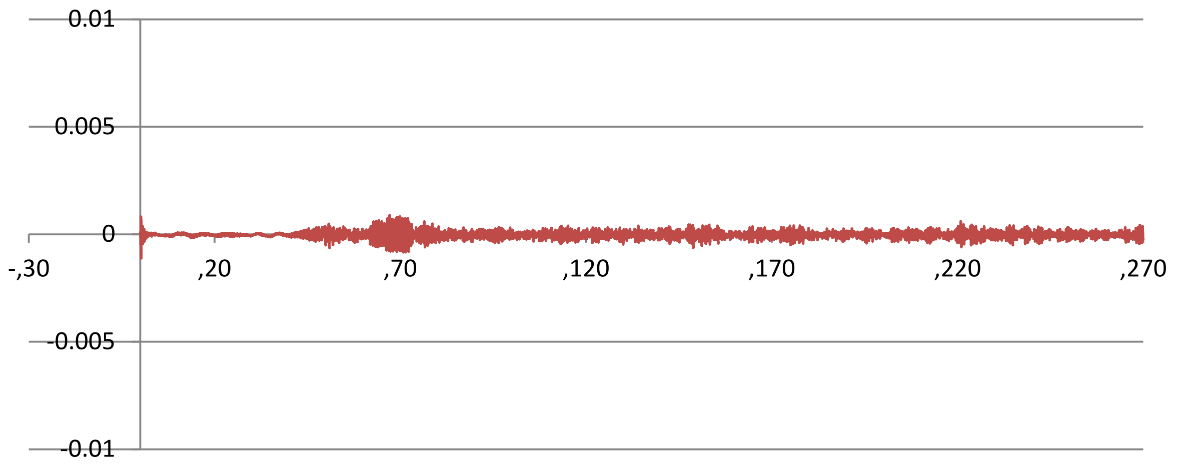
Beam 284		
Max stress	σ_{max} [MPa]	1.375E+02
Stress range	$\Delta\sigma$ [MPa]	6.635E+01
Damage		9.713E-06

Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	1.296E+02
Damage		5.781E-06

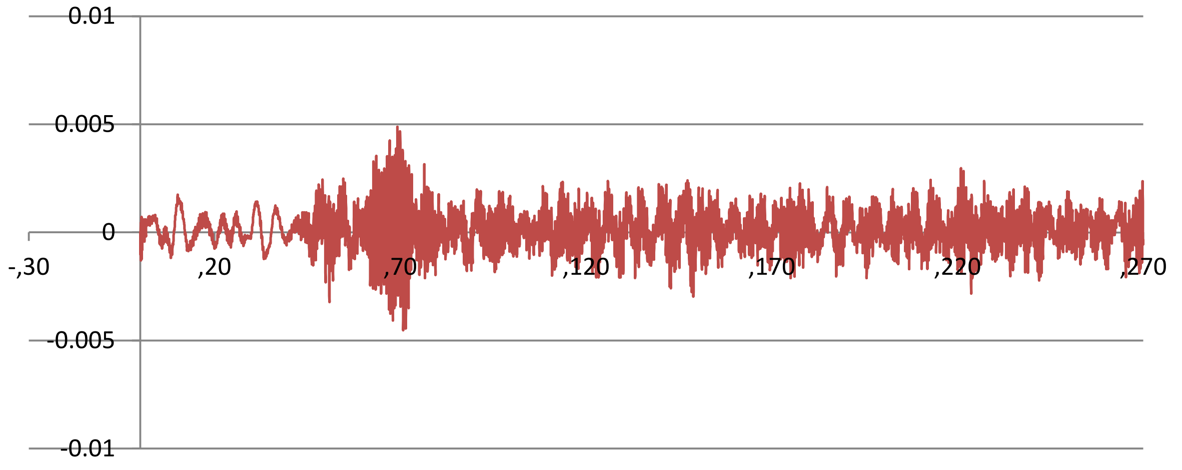


11.10.3 Bracer displacement

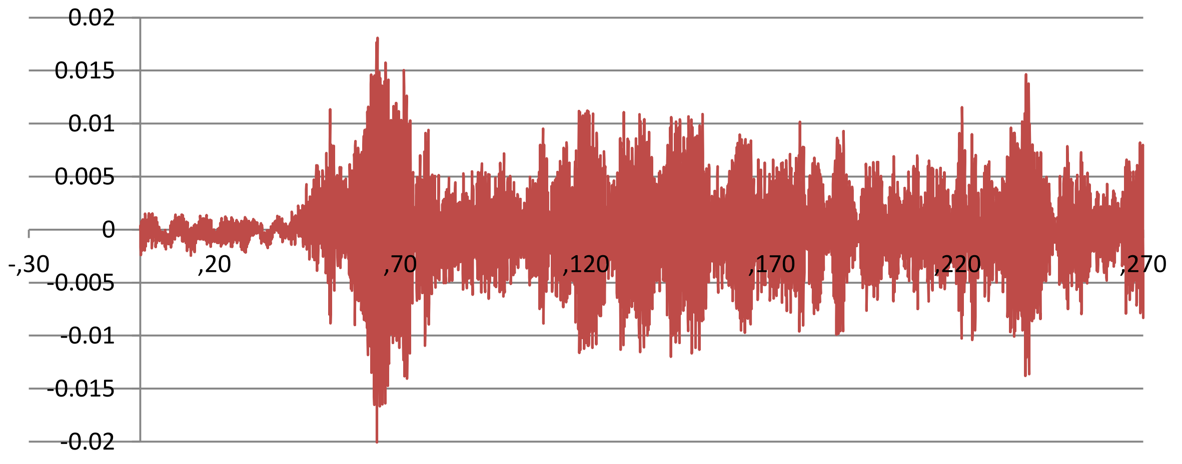
Upper Junction



Mid Junction

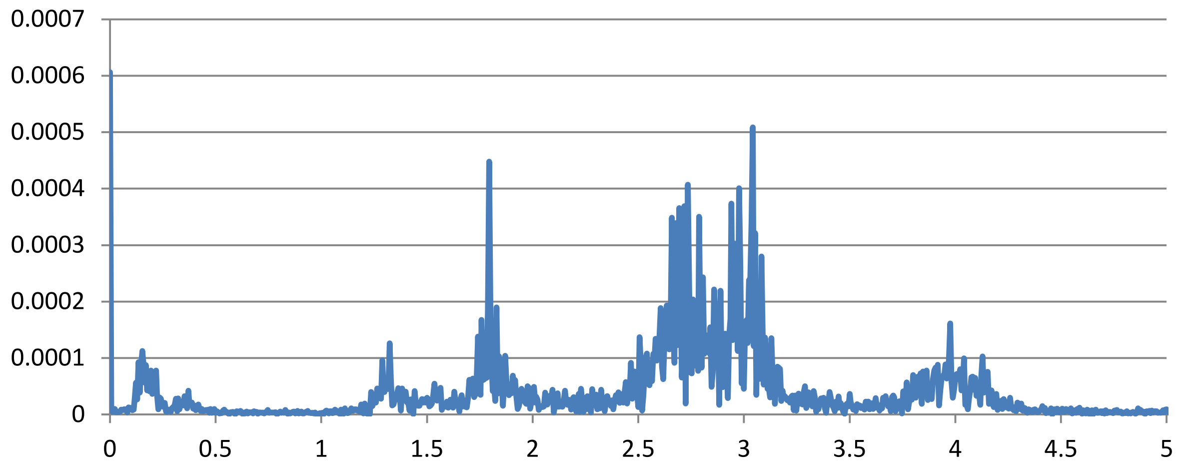


Lwr Junction

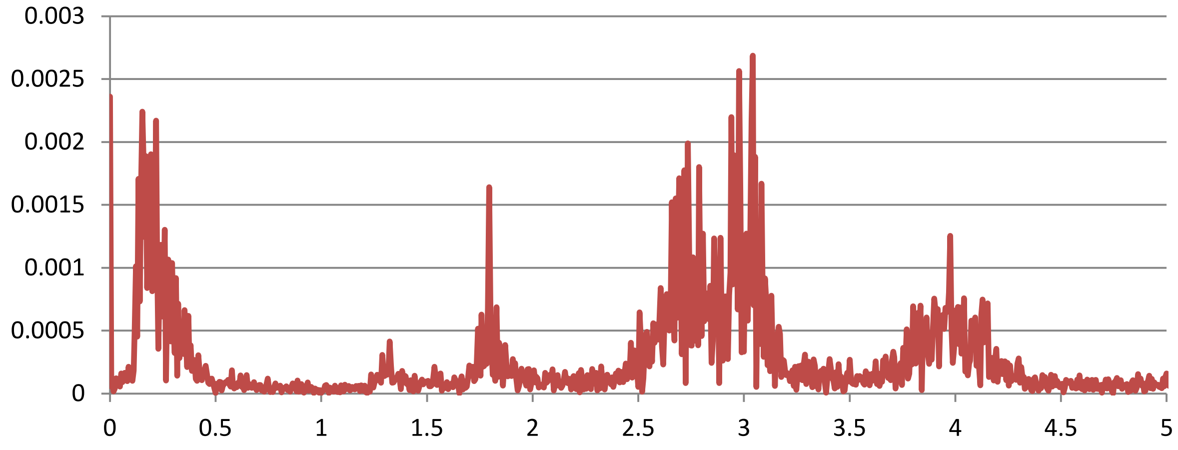


11.10.4 Bracer FFT Analysis

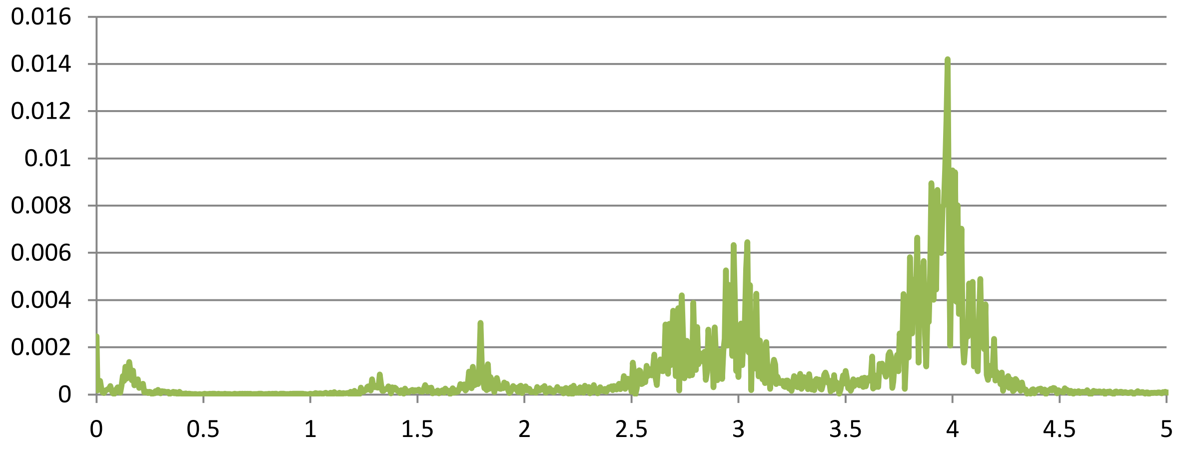
Upper Junction



Mid Junction



Lwr Junction



11.10.5 Beam stresses

Max Beam Stresses

320	304	302	288	284	265	
1.26E+07	4.93E+06	1.86E+07	1.84E+07	1.38E+08	2.65E+02	*[Pa]*
12.554	4.929	18.559	18.356	137.531	0.000	*[MPa]*

Min Beam Stresses

320	304	302	288	284	265	
-6.66E+06	-2.55E+07	-1.31E+06	-1.97E+07	7.12E+07	-1.30E+08	*[Pa]*
-6.664	-25.461	-1.306	-19.698	71.179	-129.557	*[MPa]*

Delta sigma

320	304	302	288	284	265	
19.219	30.390	19.865	38.054	66.351	129.558	*[MPa]*

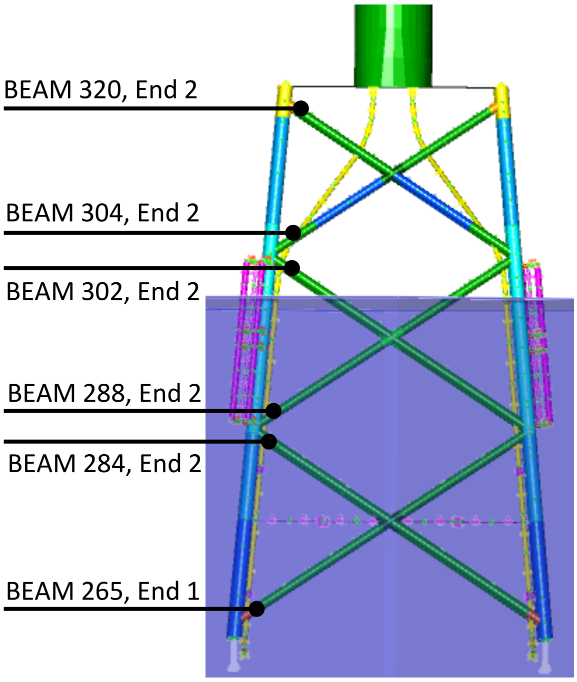
11.11 Usfos stress recovery – FCA displacement input – Event 1

11.11.1 Description

- Usfos stress recovery analysis with fedem Fully Coupled Analysis interface node displacement as input.
    - DOF z is constrained in Fedem-Analysis
    - Tower and nacelle lumped mass in z direction for Usfos stress recovery analysis
- Linear springs along soil piles for soil properties
- Turbine setup in Fedem according to “Definition of a 5-MW Reference Wind Turbine for Offshore System Development”
    - J. Jonkman, S. Butterfield, W. Musial, and G. Scott
- Event 1

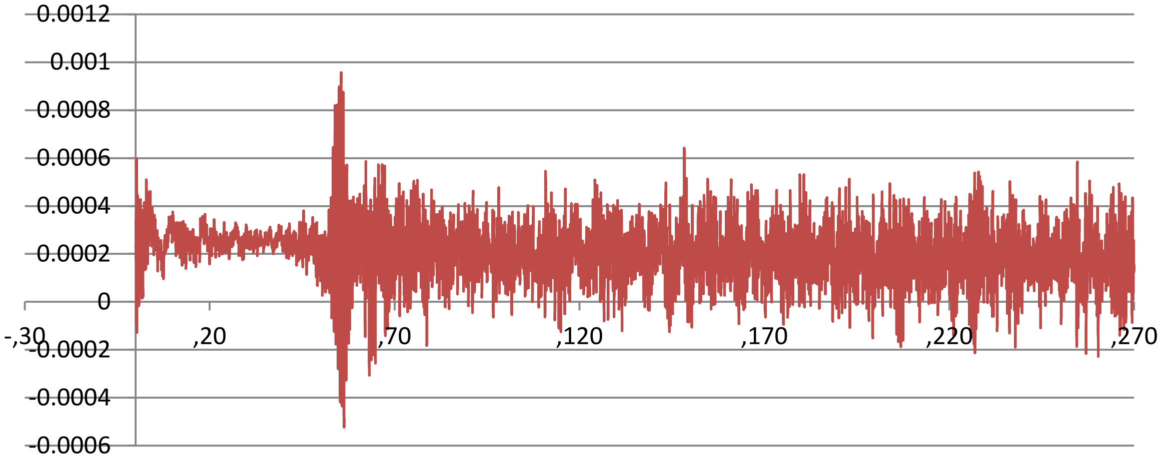
11.11.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	5.861E+00
Stress range	$\Delta\sigma$ [MPa]	1.073E+01
Damage		1.136E-08
Beam 304		
Max stress	σ_{max} [MPa]	1.006E+01
Stress range	$\Delta\sigma$ [MPa]	1.838E+01
Damage		5.352E-08
Beam 302		
Max stress	σ_{max} [MPa]	2.369E+01
Stress range	$\Delta\sigma$ [MPa]	5.363E+01
Damage		2.125E-06
Beam 288		
Max stress	σ_{max} [MPa]	2.568E+01
Stress range	$\Delta\sigma$ [MPa]	4.979E+01
Damage		2.455E-06
Beam 284		
Max stress	σ_{max} [MPa]	2.840E-04
Stress range	$\Delta\sigma$ [MPa]	1.902E+02
Damage		9.469E-08
Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	1.099E+02
Damage		2.218E-08

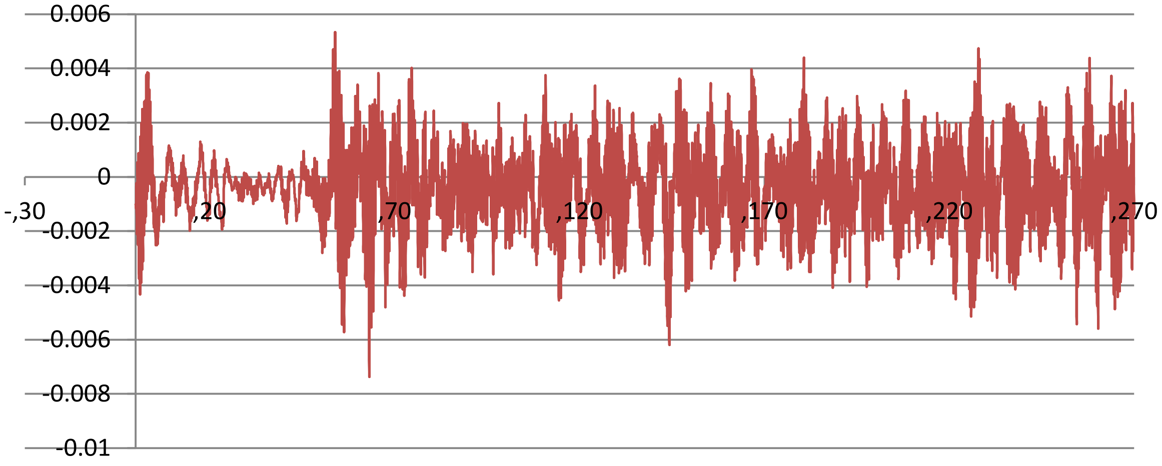


11.11.3 Bracer displacement

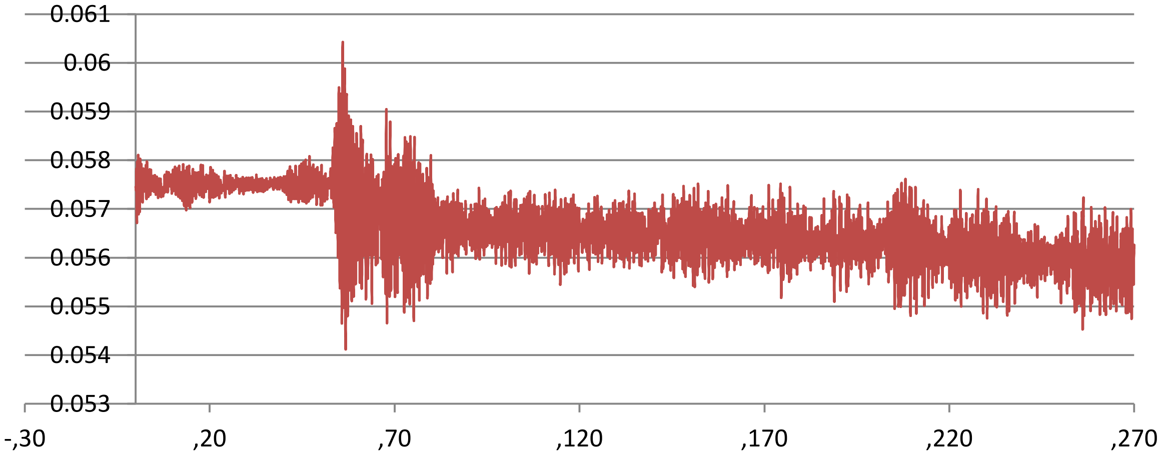
Upper Junction



Mid Junction

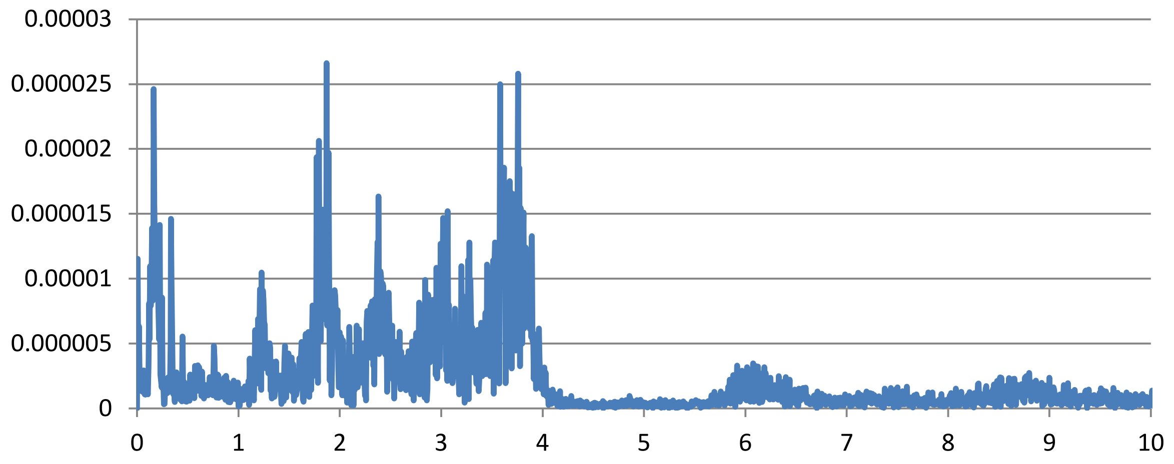


Lwr Junction

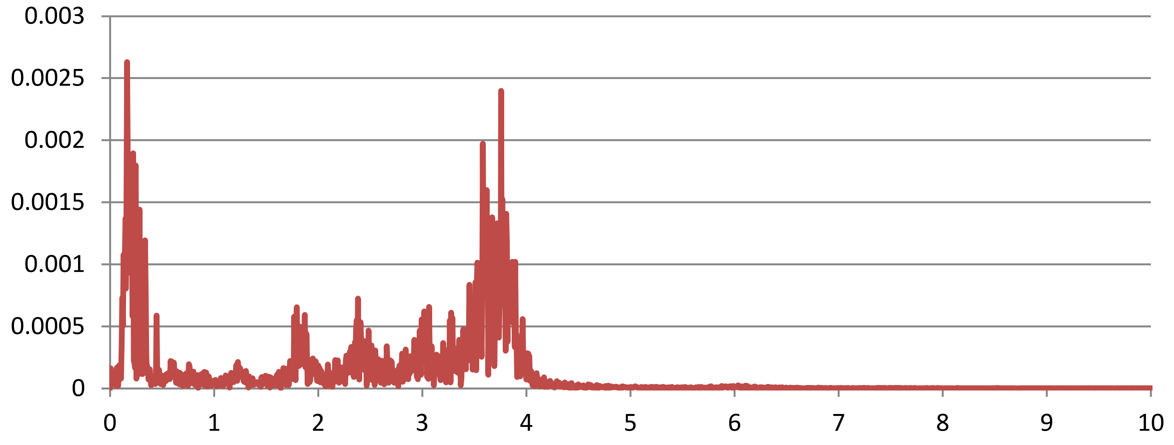


11.11.4 Bracer FFT Analysis

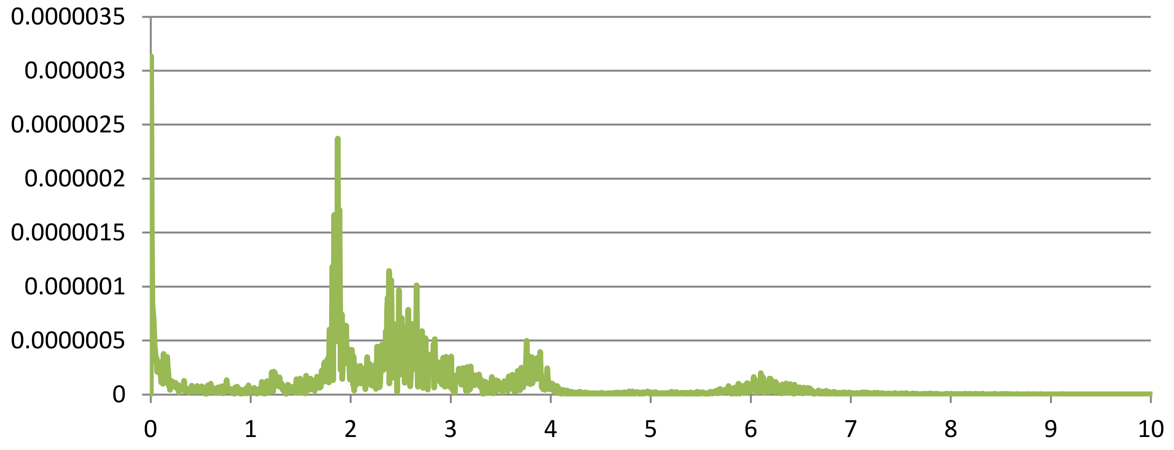
Upper Junction



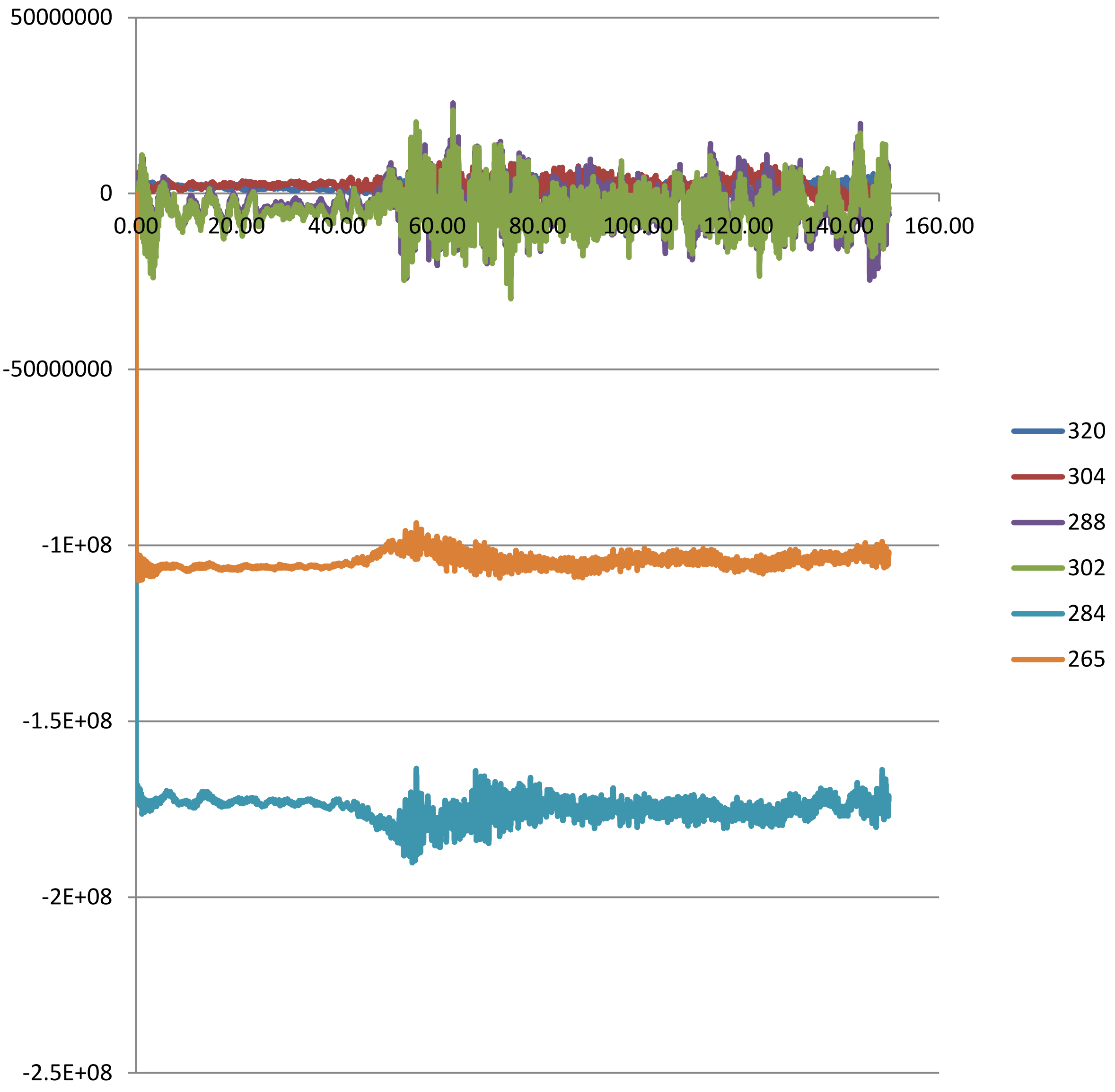
Mid Junction



Lwr Junction



11.11.5 Beam stresses



Max Beam Stresses

320	**304**	**302**	**288**	**284**	**265**	
5.86E+06	1.01E+07	2.37E+07	2.57E+07	2.84E+02	2.65E+02	*[Pa]*
5.861	10.064	23.694	25.680	0.000	0.000	*[MPa]*

Min Beam Stresses

320	**304**	**302**	**288**	**284**	**265**	
-4.87E+06	-8.32E+06	-2.99E+07	-2.41E+07	-1.90E+08	-1.10E+08	*[Pa]*
-4.870	-8.318	-29.938	-24.105	-190.241	-109.943	*[MPa]*

Delta sigma

320	**304**	**302**	**288**	**284**	**265**	
10.731	18.382	53.632	49.785	190.241	109.943	*[MPa]*

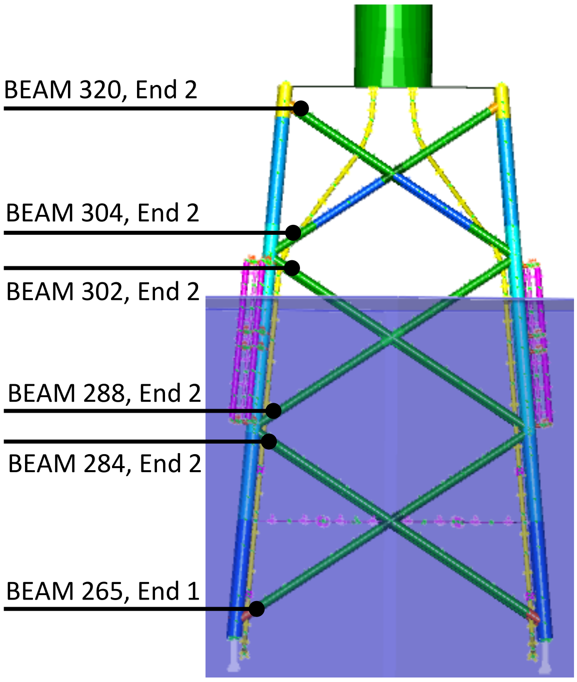
11.12 Fedem Jacket model wave load influence – Event 1

11.12.1 Description

- Fedem jacket model, without tower, stress recovery analysis
- Waves as the only external loads.
- K, C, and M matrices replacing the soil piles properties
- Event 1

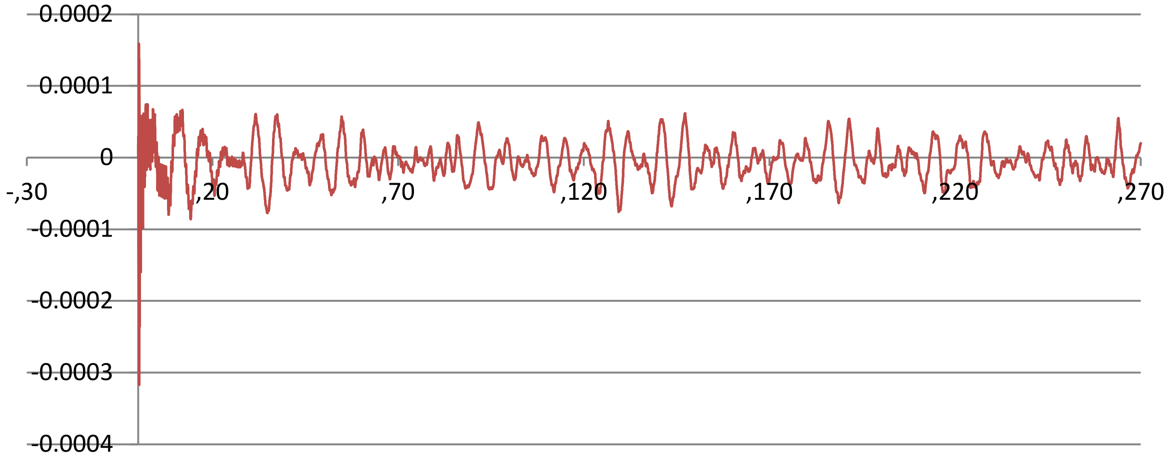
11.12.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	3.200E-04
Stress range	$\Delta\sigma$ [MPa]	2.229E+00
Damage		2.450E-12
Beam 304		
Max stress	σ_{max} [MPa]	2.699E-01
Stress range	$\Delta\sigma$ [MPa]	1.463E+00
Damage		9.851E-12
Beam 302		
Max stress	σ_{max} [MPa]	4.002E+00
Stress range	$\Delta\sigma$ [MPa]	5.471E+00
Damage		7.837E-10
Beam 288		
Max stress	σ_{max} [MPa]	4.107E+00
Stress range	$\Delta\sigma$ [MPa]	4.138E+00
Damage		4.382E-10
Beam 284		
Max stress	σ_{max} [MPa]	1.075E+02
Stress range	$\Delta\sigma$ [MPa]	6.086E+00
Damage		6.363E-10
Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	9.500E+01
Damage		1.233E-10

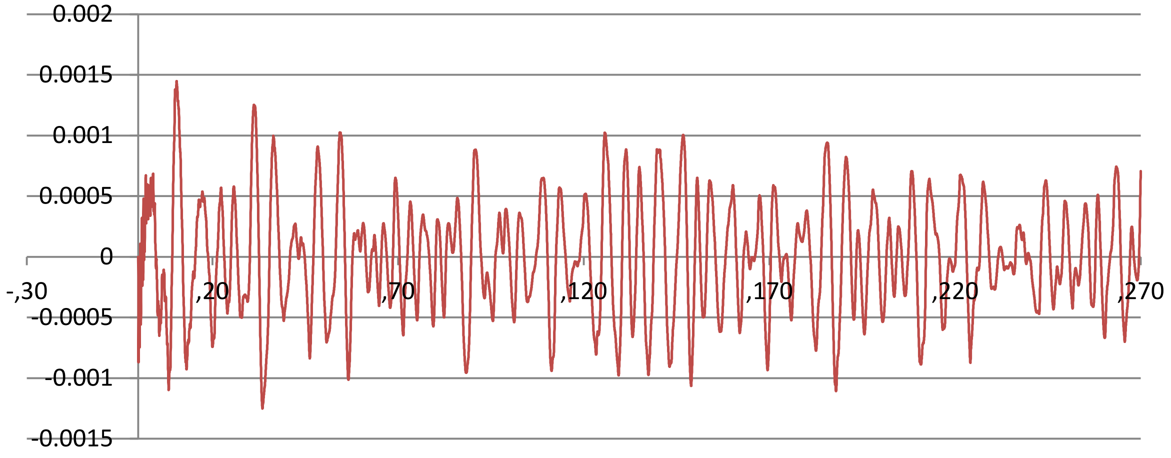


11.12.3 Bracer displacement

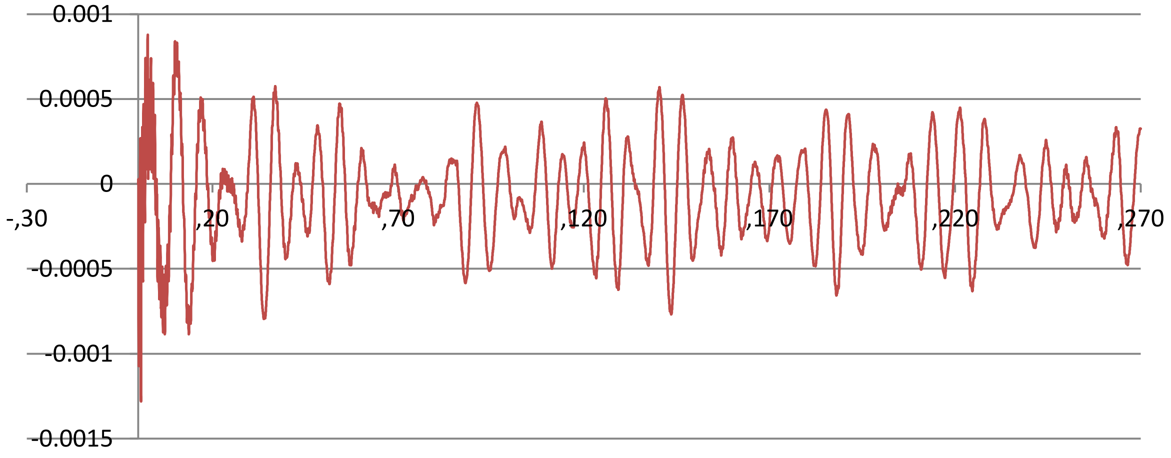
Upper Junction



Mid Junction

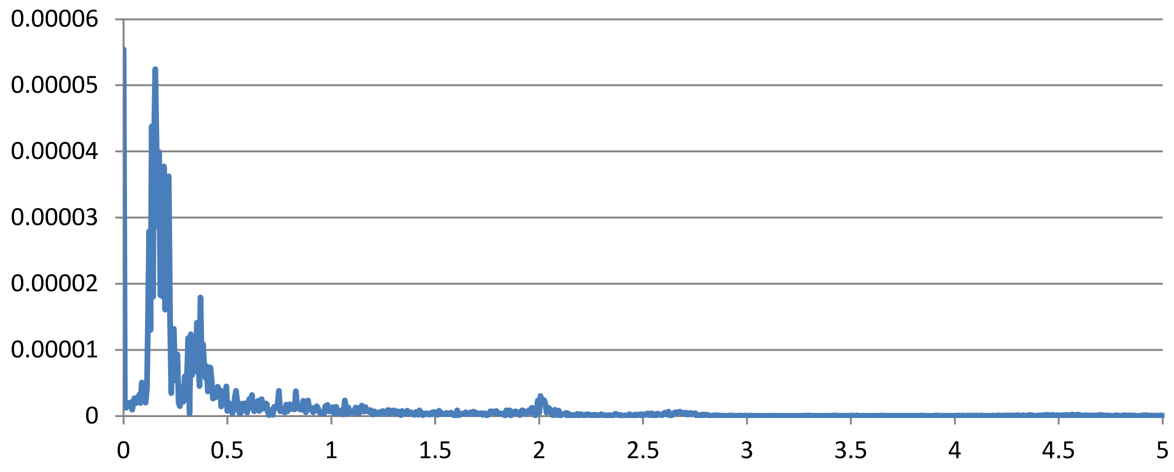


Lwr Junction

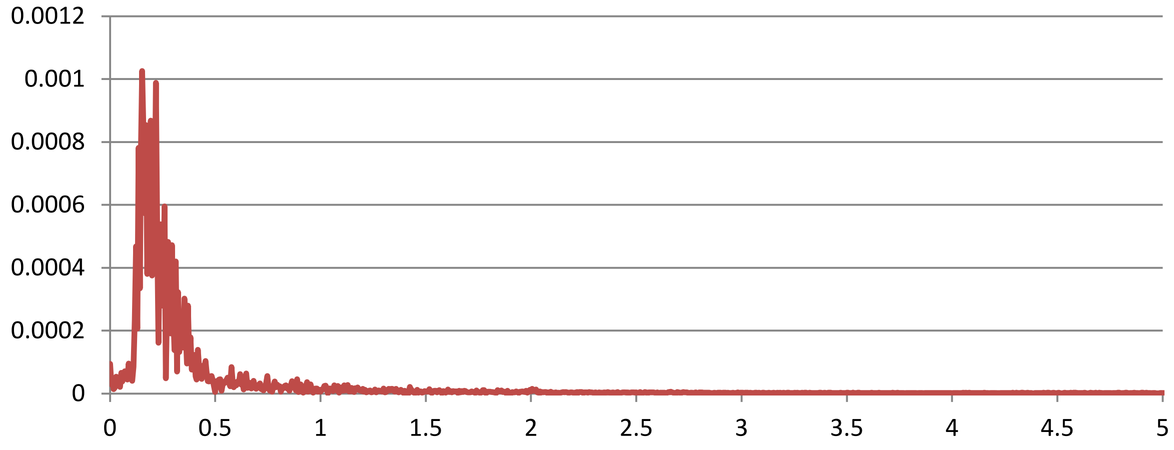


11.12.4 Bracer FFT Analysis

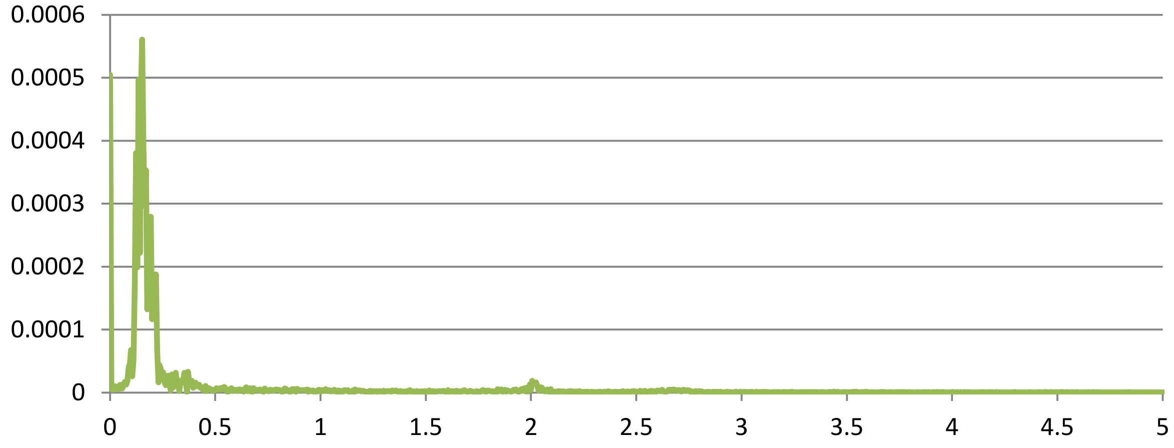
Upper Junction



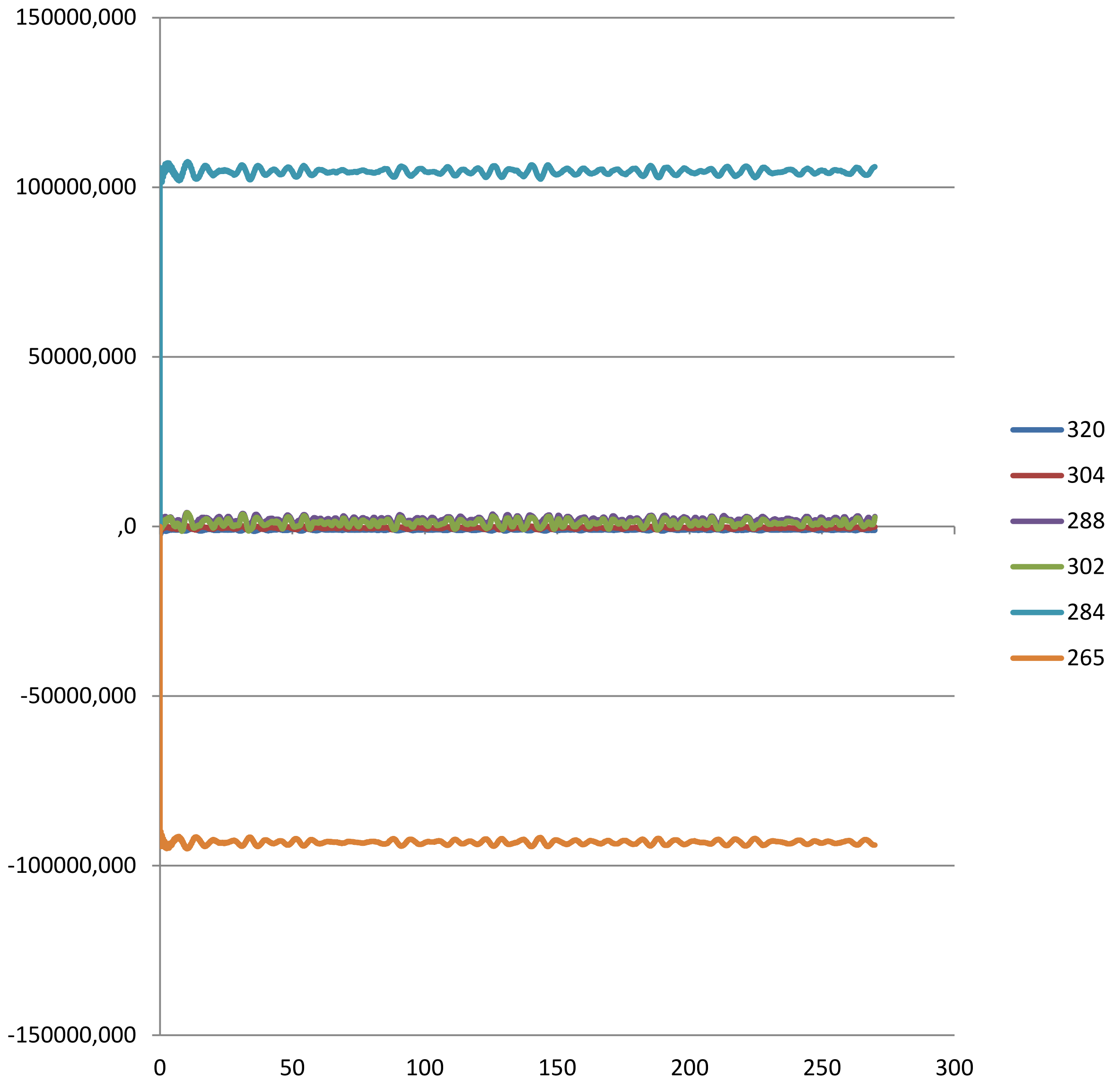
Mid Junction



Lwr Junction



11.12.5 Beam stresses



Max Beam Stresses

320	304	302	288	284	265	
3.20E+02	2.70E+05	4.00E+06	4.11E+06	1.08E+08	2.65E+02	*[Pa]*
0.000	0.270	4.002	4.107	107.532	0.000	*[MPa]*

Min Beam Stresses

320	304	302	288	284	265	
-2.23E+06	-1.19E+06	-1.47E+06	-3.06E+04	1.01E+08	-9.50E+07	*[Pa]*
-2.229	-1.193	-1.469	-0.031	101.446	-95.005	*[MPa]*

Delta sigma

320	304	302	288	284	265	
2.229	1.463	5.471	4.138	6.086	95.005	*[MPa]*

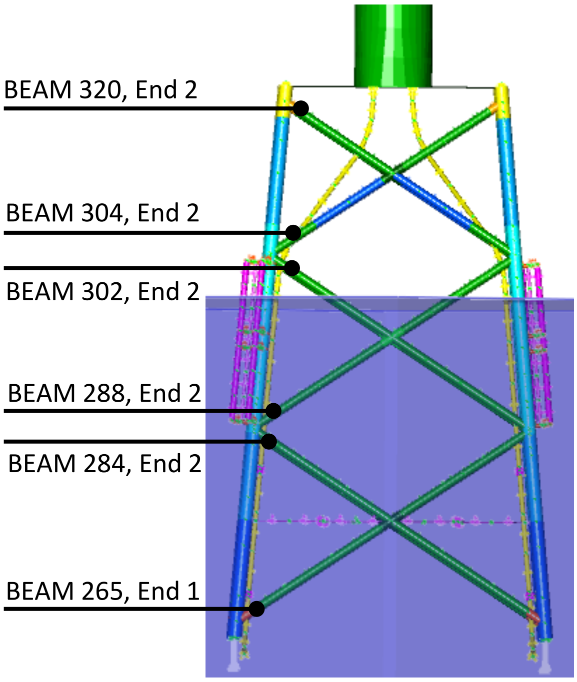
11.13 Usfos Jacket model wave load influence – Event 1

11.13.1 Description

- Usfos jacket model, without tower, stress recovery analysis
- Waves as the only external loads.
- Linear springs along soil piles for soil properties
- Event 1

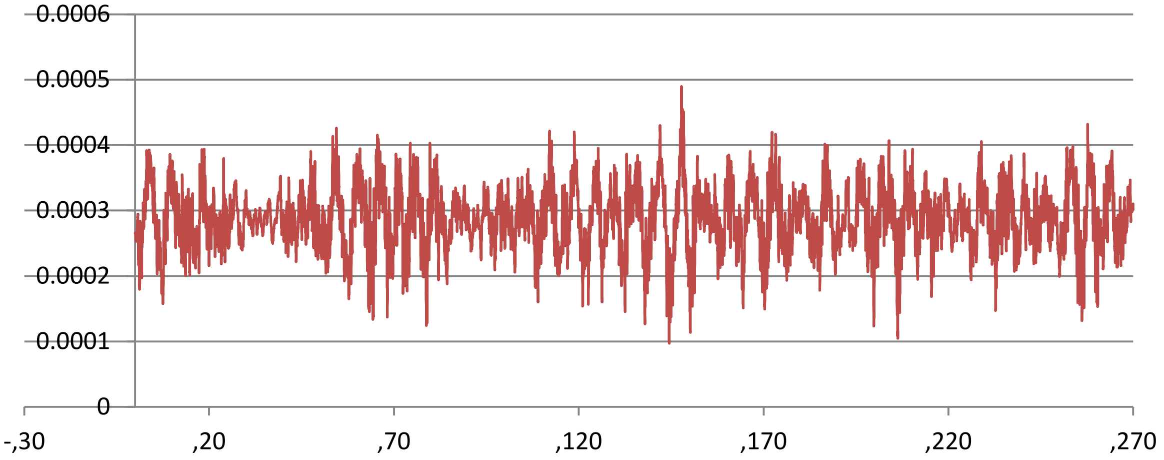
11.13.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	1.290E+00
Stress range	$\Delta\sigma$ [MPa]	1.241E+00
Damage		2.589E-11
Beam 304		
Max stress	σ_{max} [MPa]	5.454E-01
Stress range	$\Delta\sigma$ [MPa]	2.992E+00
Damage		2.925E-10
Beam 302		
Max stress	σ_{max} [MPa]	1.227E+01
Stress range	$\Delta\sigma$ [MPa]	2.475E+01
Damage		1.088E-07
Beam 288		
Max stress	σ_{max} [MPa]	1.206E+01
Stress range	$\Delta\sigma$ [MPa]	2.304E+01
Damage		7.895E-08
Beam 284		
Max stress	σ_{max} [MPa]	2.840E-04
Stress range	$\Delta\sigma$ [MPa]	1.780E+02
Damage		1.858E-09
Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	1.031E+02
Damage		1.257E-10

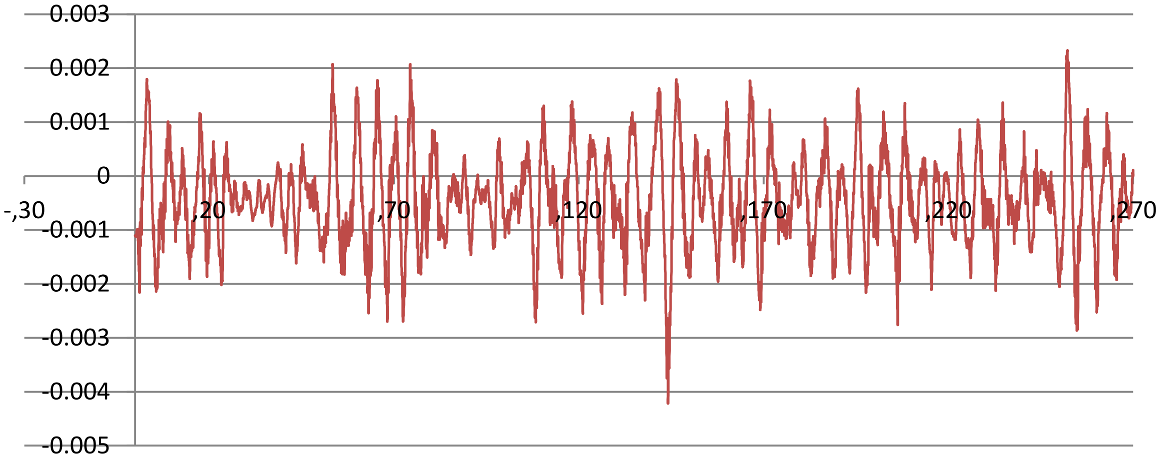


11.13.3 Bracer displacement

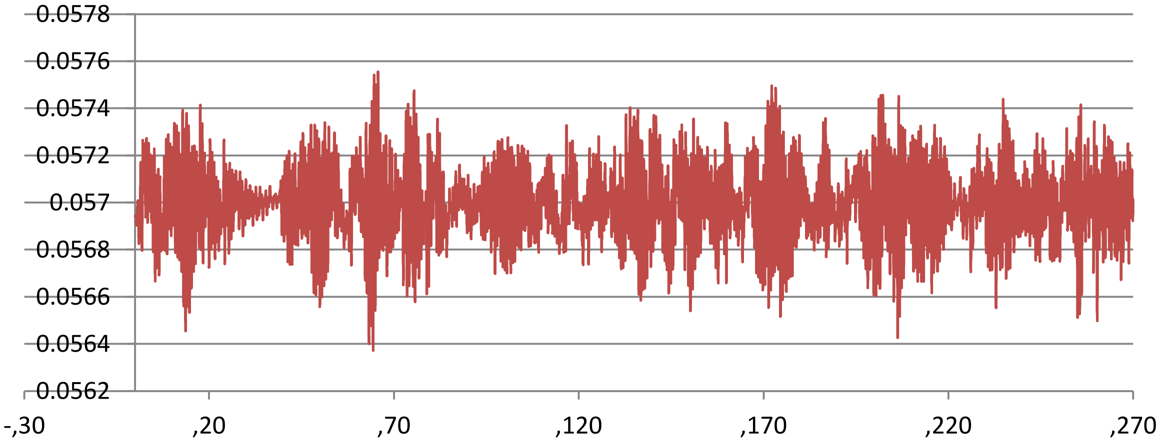
Upper Junction



Mid Junction

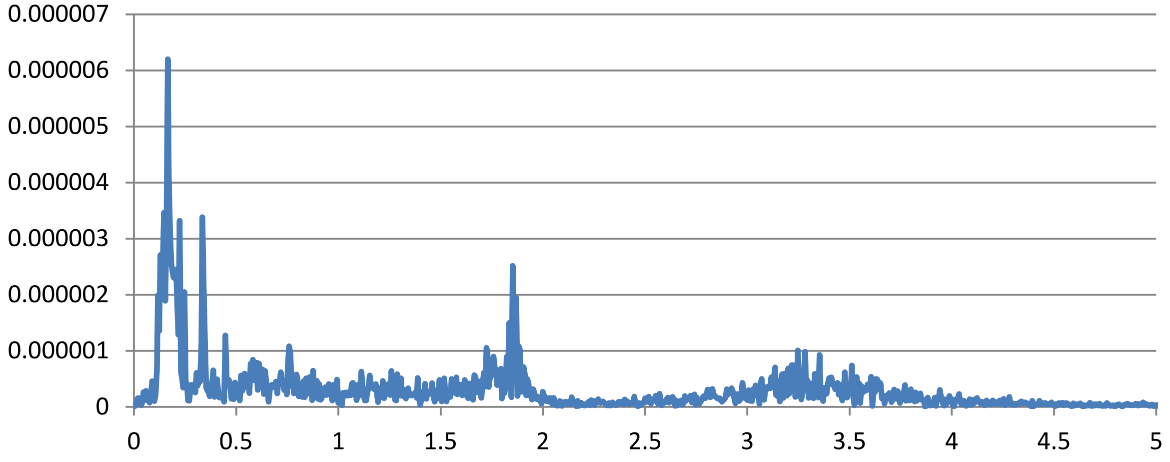


Lwr Junction

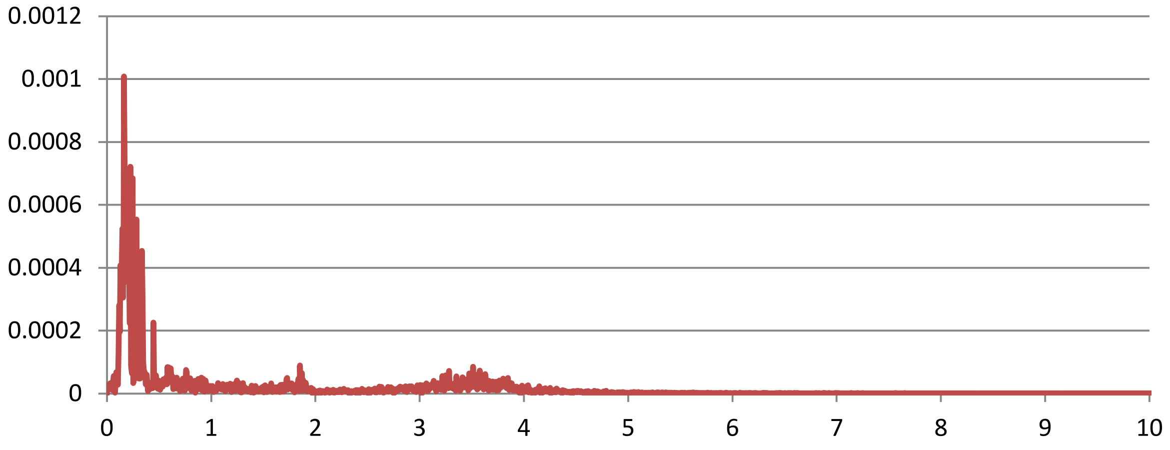


11.13.4 Bracer FFT Analysis

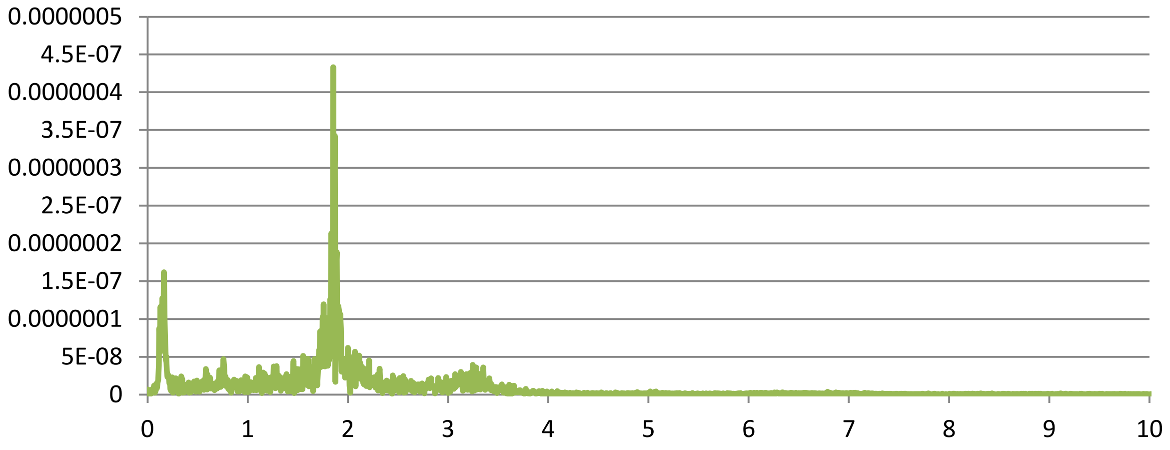
Upper Junction



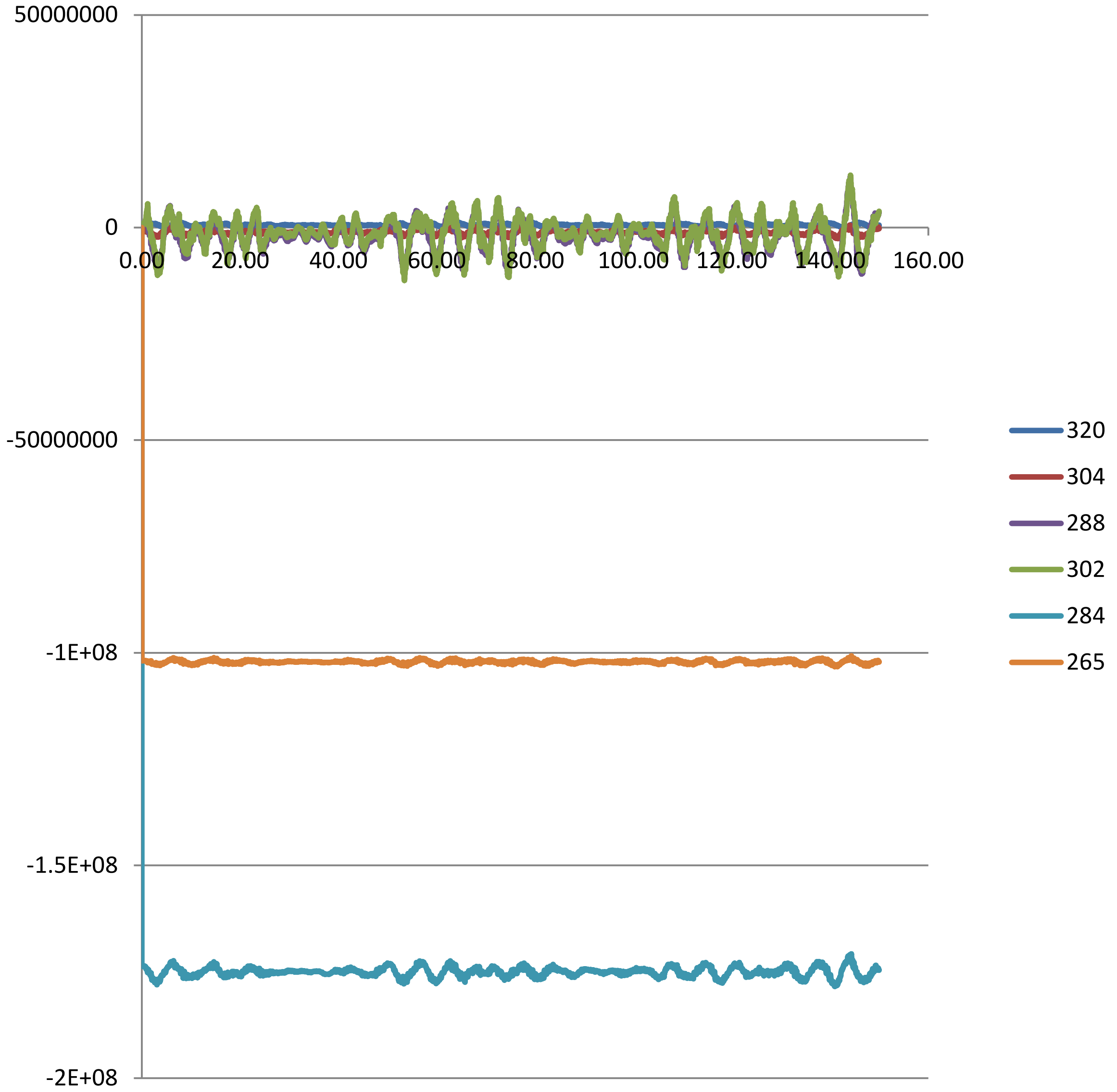
Mid Junction



Lwr Junction



11.13.5 Beam stresses



Max Beam Stresses

320	304	302	288	284	265	
1.29E+06	5.45E+05	1.23E+07	1.21E+07	2.84E+02	2.65E+02	*[Pa]*
1.290	0.545	12.274	12.063	0.000	0.000	*[MPa]*

Min Beam Stresses

320	304	302	288	284	265	
4.84E+04	-2.45E+06	-1.25E+07	-1.10E+07	-1.78E+08	-1.03E+08	*[Pa]*
0.048	-2.447	-12.474	-10.975	-178.043	-103.095	*[MPa]*

Delta sigma

320	304	302	288	284	265	
1.241	2.992	24.748	23.038	178.043	103.095	*[MPa]*

11.14 Sequentially Coupled Analysis- Event 1-C01-Sensitivity

11.14.1 Description

- Sensitivity analysis for damping [C] matrix. -20% of original value
- Fully coupled Usfos/Fedem analysis
- K, C, and M matrices replacing the interface node for Fedem analysis
    - DOF z is constrained in Fedem-Analysis
    - Tower and nacelle lumped mass in z direction for Usfos stress recovery analysis
- Linear springs along soil piles for soil properties
- Turbine setup in Fedem according to “Definition of a 5-MW Reference Wind Turbine for Offshore System Development”
    - J. Jonkman, S. Butterfield, W. Musial, and G. Scott
- Event 1

11.14.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	2.086E+01
Stress range	$\Delta\sigma$ [MPa]	3.603E+01
Damage		1.181E-06

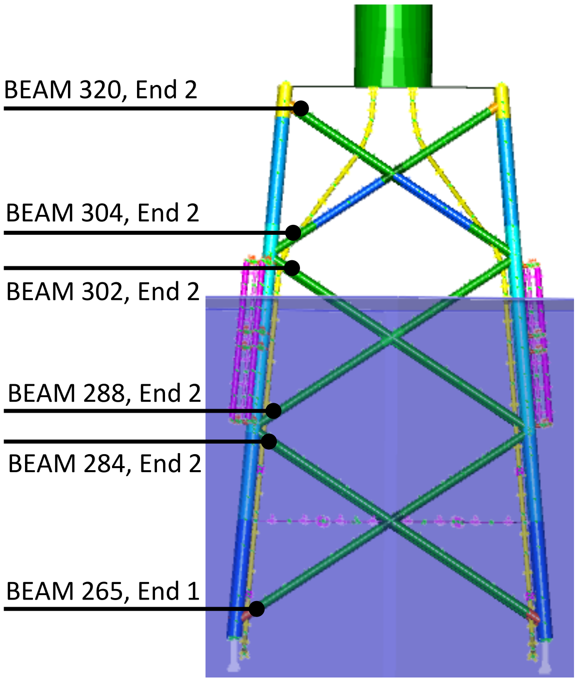
Beam 304		
Max stress	σ_{max} [MPa]	3.330E+01
Stress range	$\Delta\sigma$ [MPa]	5.953E+01
Damage		3.297E-06

Beam 302		
Max stress	σ_{max} [MPa]	5.194E+01
Stress range	$\Delta\sigma$ [MPa]	1.186E+02
Damage		1.314E-04

Beam 288		
Max stress	σ_{max} [MPa]	4.648E+01
Stress range	$\Delta\sigma$ [MPa]	1.096E+02
Damage		7.081E-05

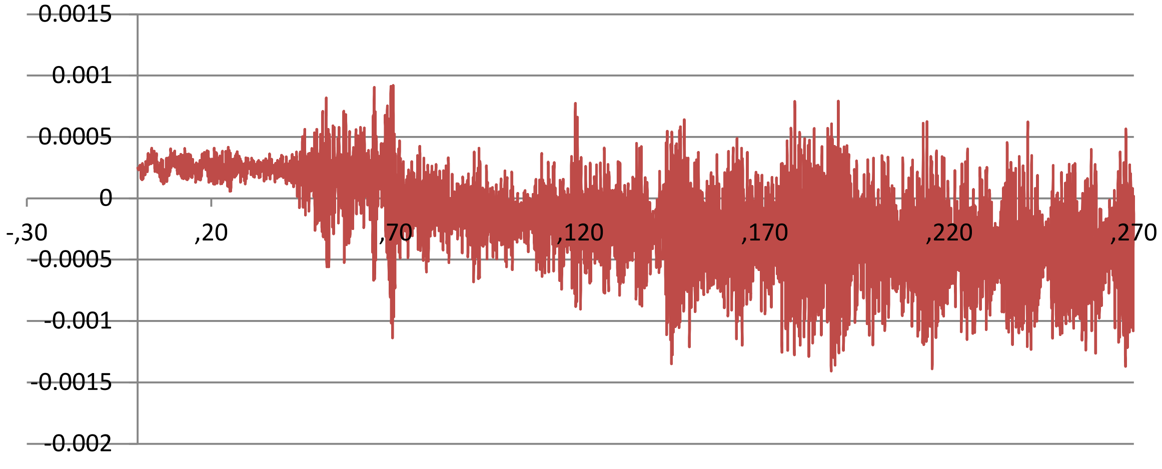
Beam 284		
Max stress	σ_{max} [MPa]	2.840E-04
Stress range	$\Delta\sigma$ [MPa]	2.089E+02
Damage		8.445E-06

Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	1.286E+02
Damage		1.635E-06

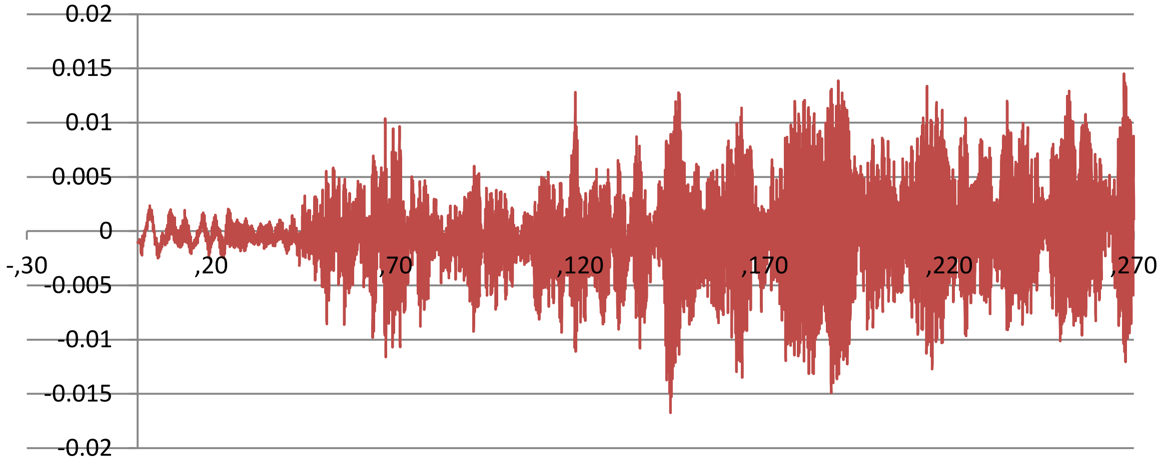


11.14.3 Bracer displacement

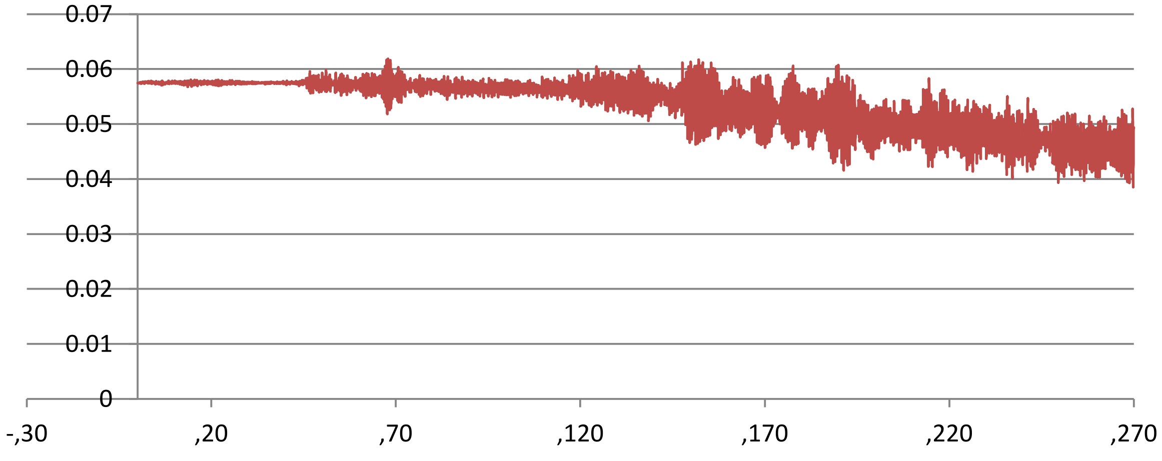
Upper Junction



Mid Junction

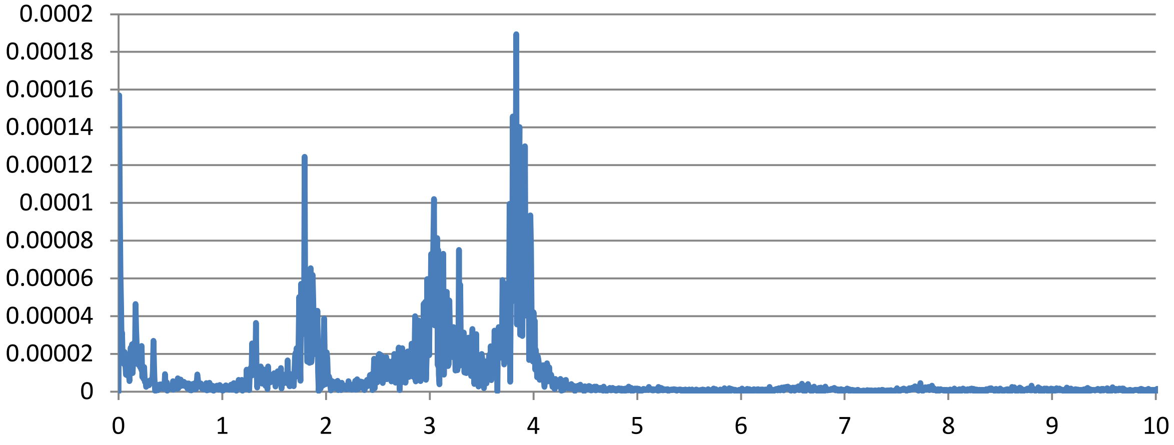


Lwr Junction

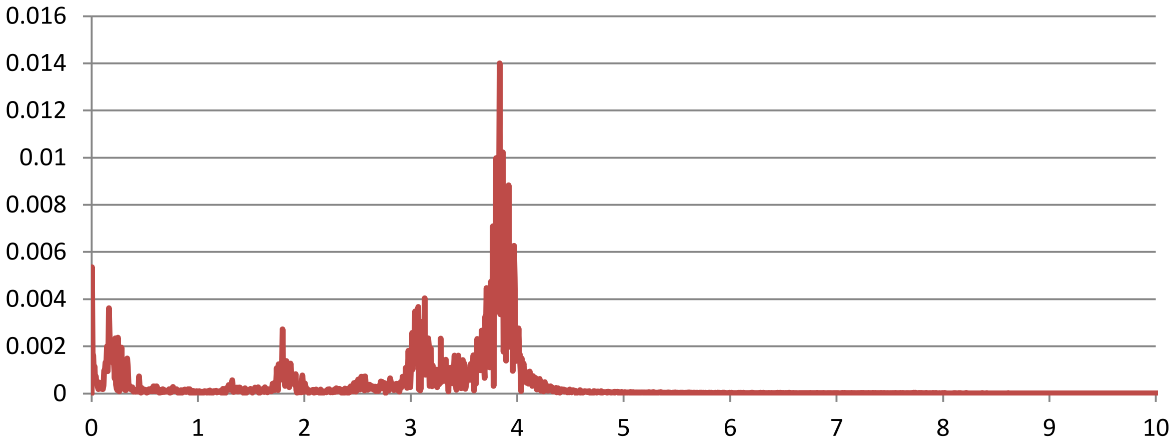


11.14.4 Bracer FFT Analysis

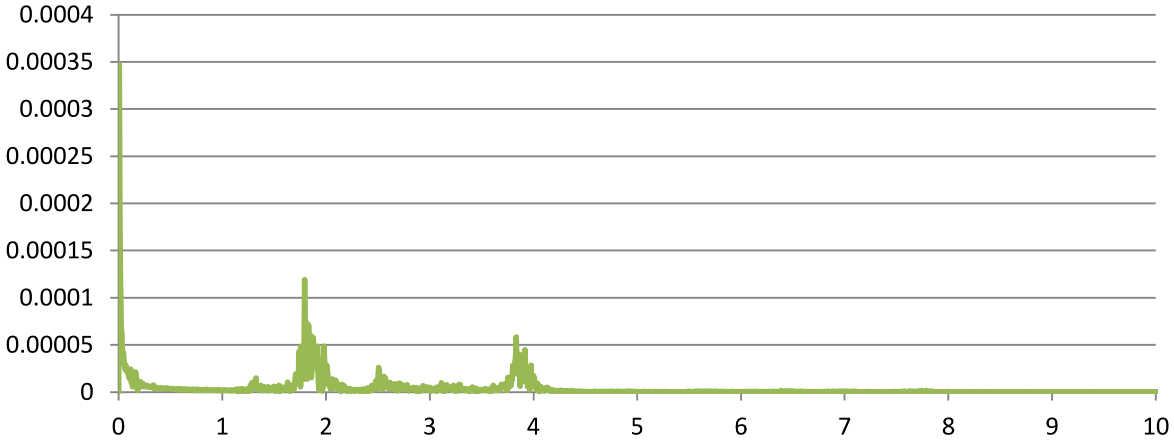
Upper Junction



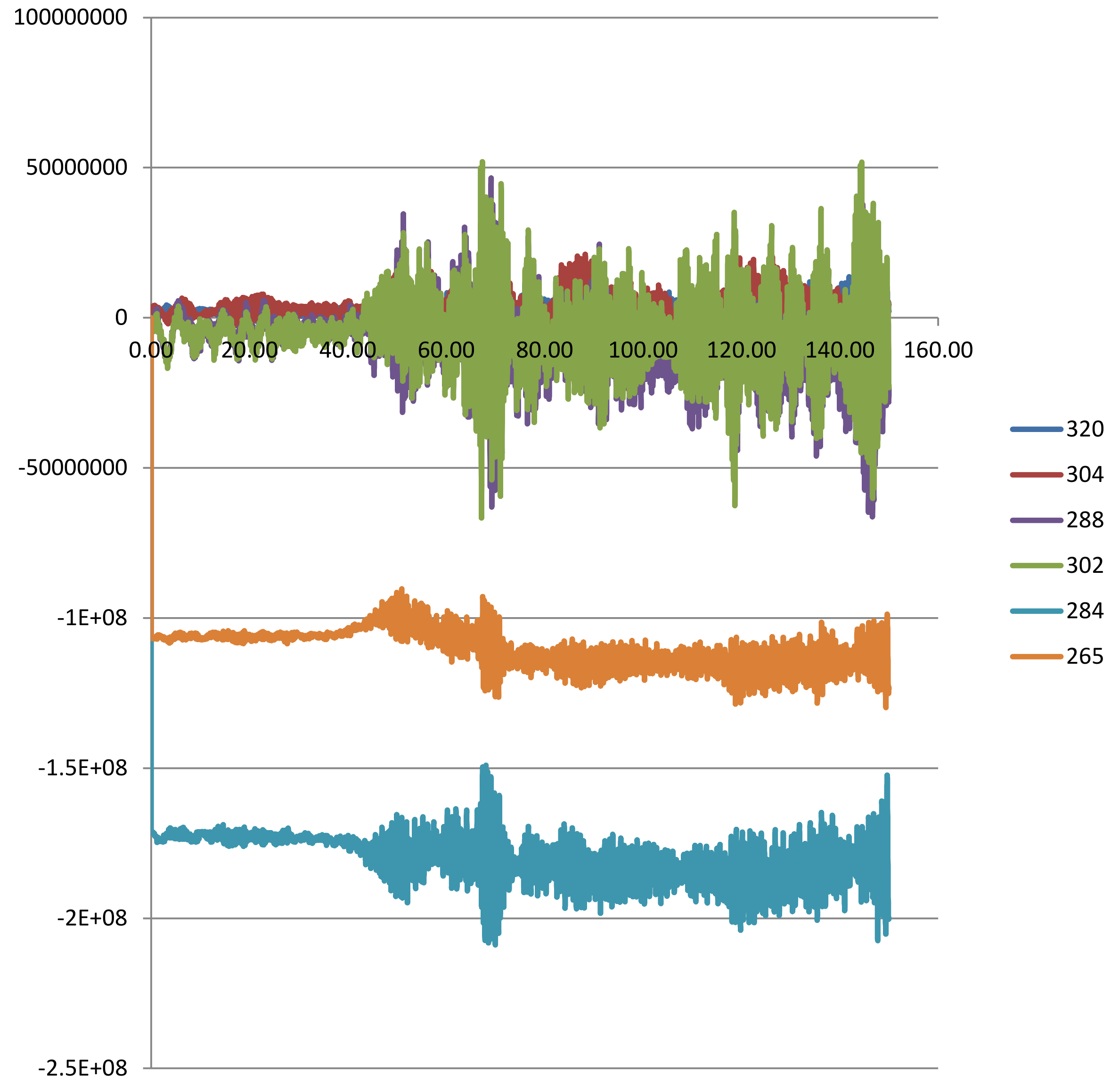
Mid Junction



Lwr Junction



11.14.5 Beam stresses



Max Beam Stresses

320	304	302	288	284	265	
2.09E+07	3.33E+07	5.19E+07	4.65E+07	2.84E+02	2.65E+02	*[Pa]*
20.862	33.305	51.940	46.481	0.000	0.000	*[MPa]*

Min Beam Stresses

320	304	302	288	284	265	
-1.52E+07	-2.62E+07	-6.66E+07	-6.31E+07	-2.09E+08	-1.29E+08	*[Pa]*
-15.164	-26.224	-66.614	-63.096	-208.864	-128.618	*[MPa]*

Delta sigma

320	304	302	288	284	265	
36.026	59.529	118.554	109.577	208.864	128.618	*[MPa]*

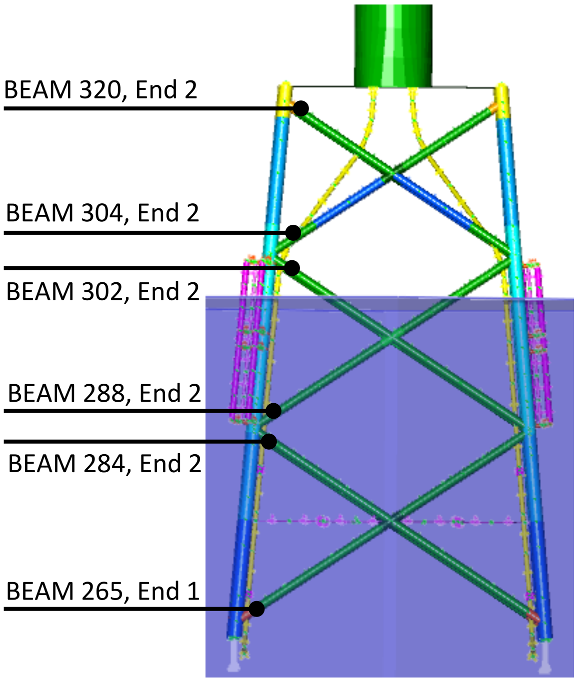
11.15 Sequentially Coupled Analysis- Event 1-C02-Sensitivity

11.15.1 Description

- Sensitivity analysis for damping [C] matrix. -10% of original value
- Fully coupled Usfos/Fedem analysis
- K, C, and M matrices replacing the interface node for Fedem analysis
    - DOF z is constrained in Fedem-Analysis
    - Tower and nacelle lumped mass in z direction for Usfos stress recovery analysis
- Linear springs along soil piles for soil properties
- Turbine setup in Fedem according to “Definition of a 5-MW Reference Wind Turbine for Offshore System Development”
    - J. Jonkman, S. Butterfield, W. Musial, and G. Scott
- Event 1

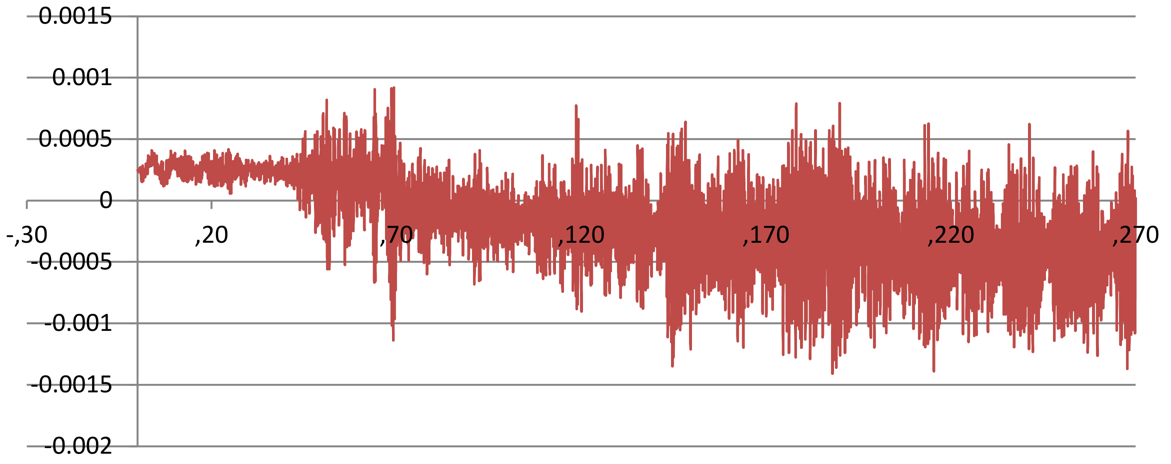
11.15.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	2.086E+01
Stress range	$\Delta\sigma$ [MPa]	3.603E+01
Damage		1.181E-06
Beam 304		
Max stress	σ_{max} [MPa]	3.330E+01
Stress range	$\Delta\sigma$ [MPa]	5.953E+01
Damage		3.297E-06
Beam 302		
Max stress	σ_{max} [MPa]	5.194E+01
Stress range	$\Delta\sigma$ [MPa]	1.186E+02
Damage		1.314E-04
Beam 288		
Max stress	σ_{max} [MPa]	4.648E+01
Stress range	$\Delta\sigma$ [MPa]	1.096E+02
Damage		7.081E-05
Beam 284		
Max stress	σ_{max} [MPa]	2.840E-04
Stress range	$\Delta\sigma$ [MPa]	2.089E+02
Damage		8.445E-06
Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	1.286E+02
Damage		1.635E-06

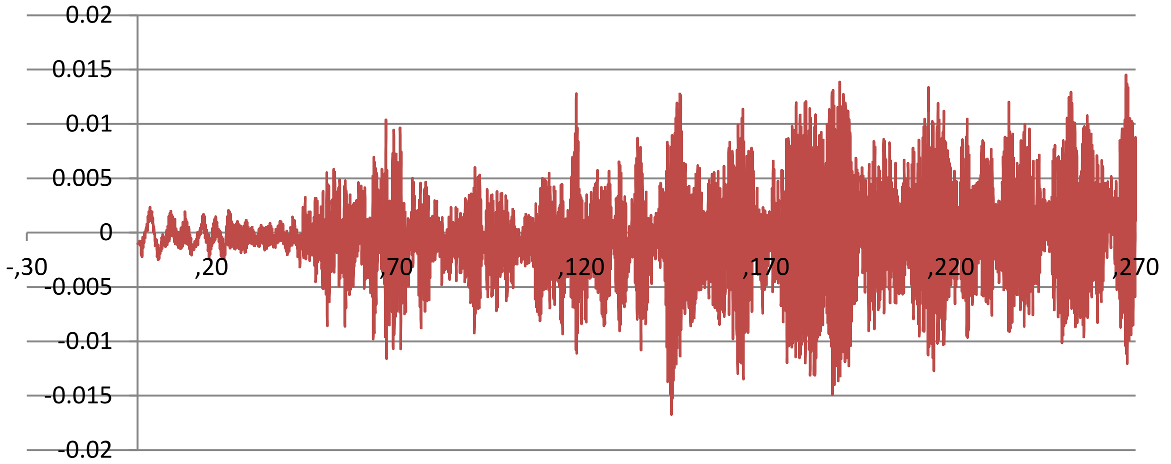


11.15.3 Bracer displacement

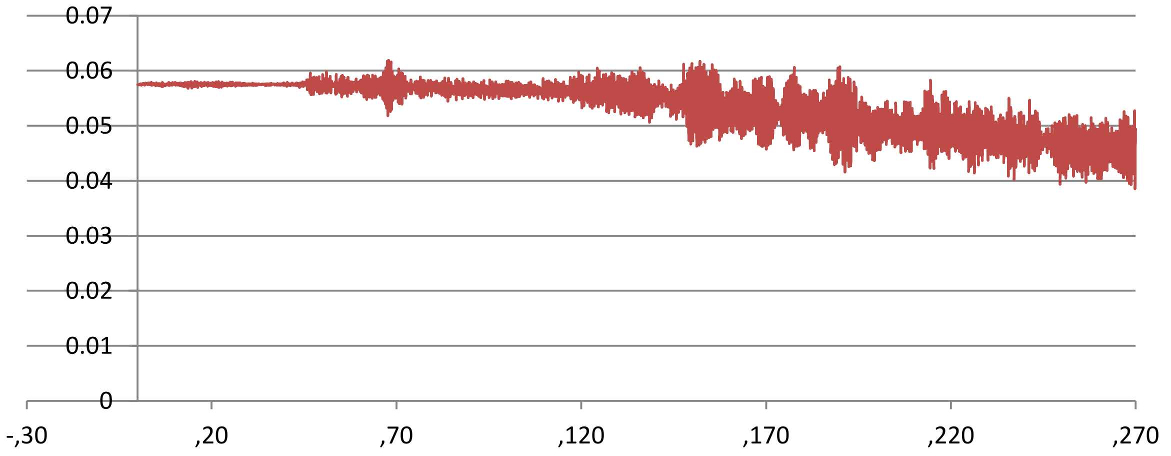
Upper Junction



Mid Junction

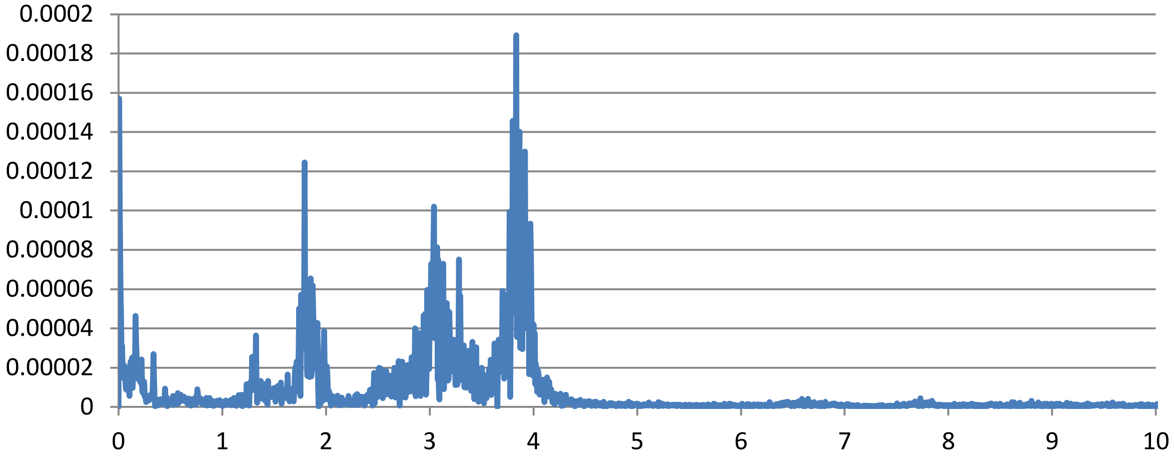


Lwr Junction

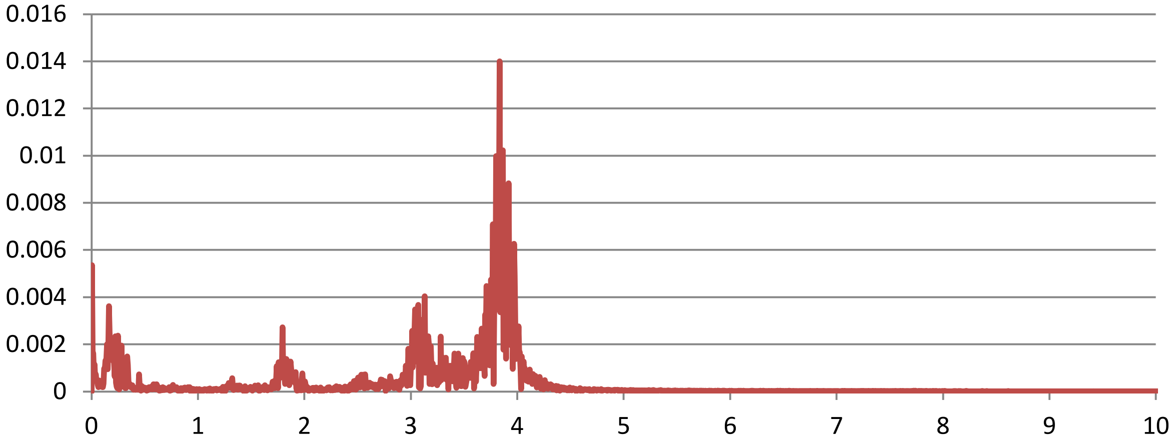


11.15.4 Bracer FFT Analysis

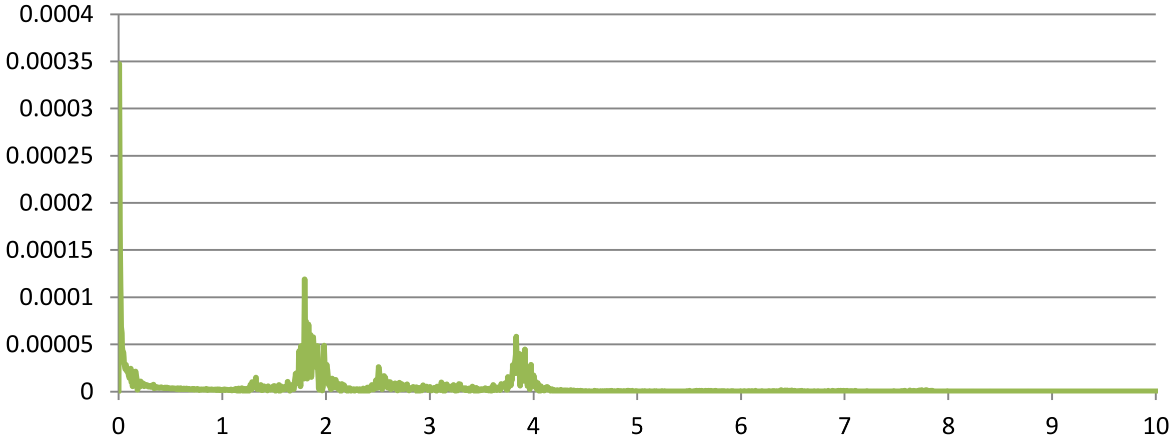
Upper Junction



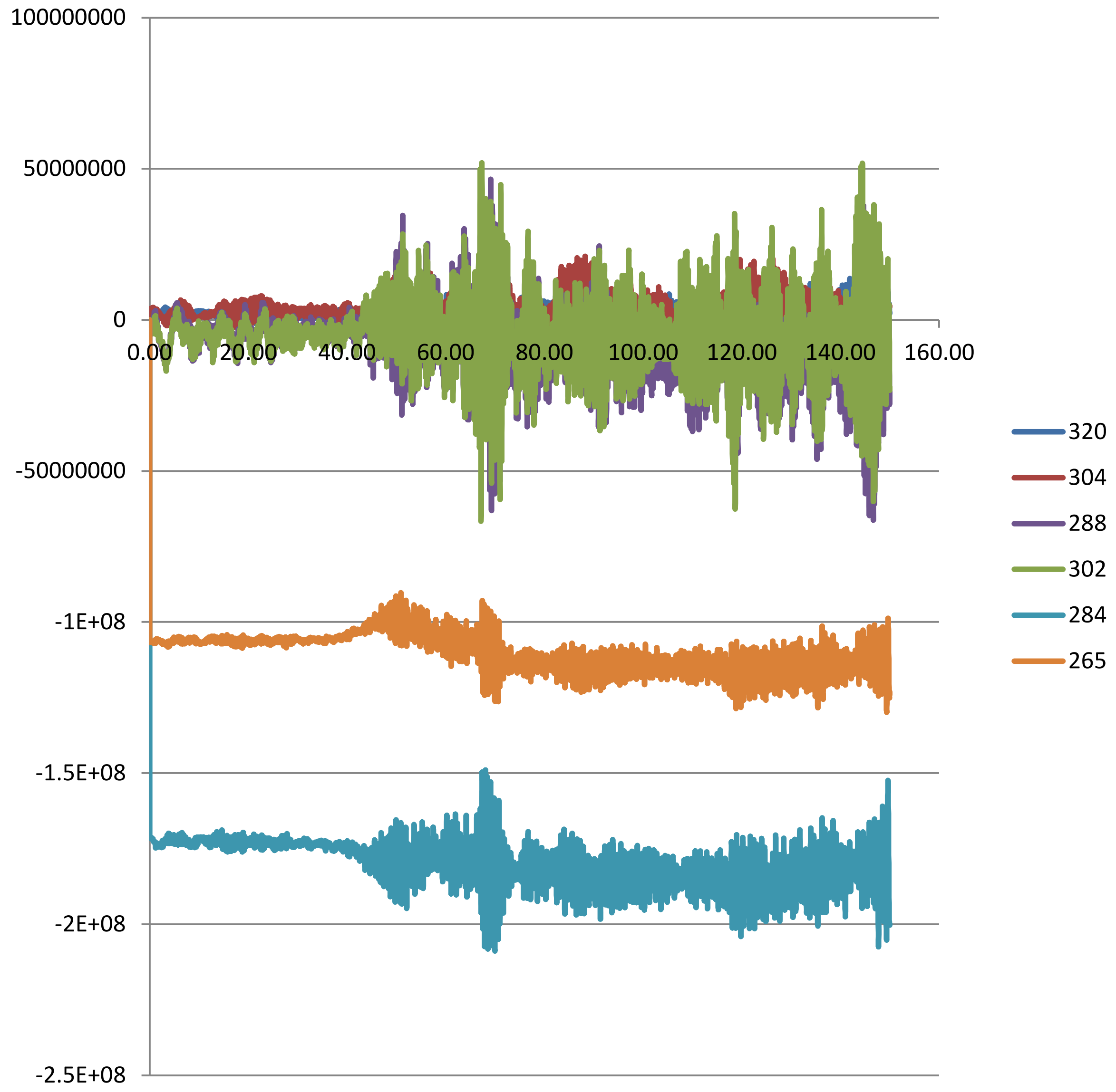
Mid Junction



Lwr Junction



11.15.5 Beam stresses



Max Beam Stresses

320	**304**	**302**	**288**	**284**	**265**	
2.09E+07	3.33E+07	5.19E+07	4.65E+07	2.84E+02	2.65E+02	*[Pa]*
20.862	33.305	51.940	46.481	0.000	0.000	*[MPa]*

Min Beam Stresses

320	**304**	**302**	**288**	**284**	**265**	
-1.52E+07	-2.62E+07	-6.66E+07	-6.31E+07	-2.09E+08	-1.29E+08	*[Pa]*
-15.164	-26.224	-66.614	-63.096	-208.864	-128.618	*[MPa]*

Delta sigma

320	**304**	**302**	**288**	**284**	**265**	
36.026	59.529	118.554	109.577	208.864	128.618	*[MPa]*

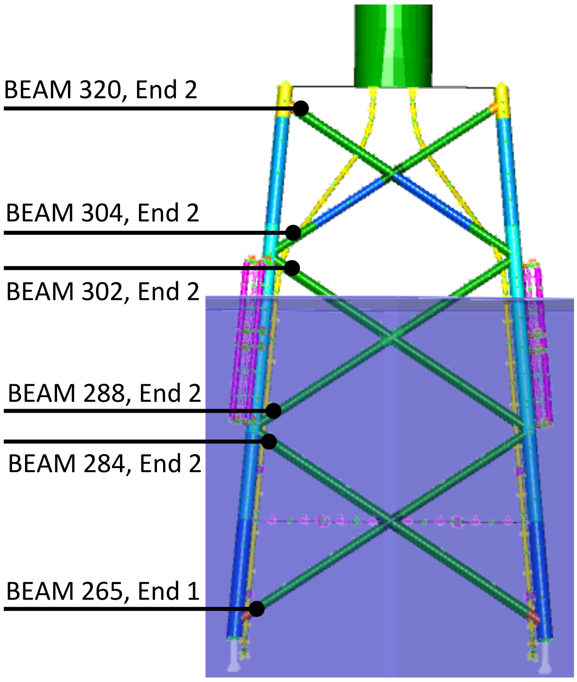
11.16 Sequentially Coupled Analysis- Event 1-C03-Sensitivity

11.16.1 Description

- Sensitivity analysis for damping [C] matrix. +s10% of original value
- Fully coupled Usfos/Fedem analysis
- K, C, and M matrices replacing the interface node for Fedem analysis
    - DOF z is constrained in Fedem-Analysis
    - Tower and nacelle lumped mass in z direction for Usfos stress recovery analysis
- Linear springs along soil piles for soil properties
- Turbine setup in Fedem according to “Definition of a 5-MW Reference Wind Turbine for Offshore System Development”
    - J. Jonkman, S. Butterfield, W. Musial, and G. Scott
- Event 1

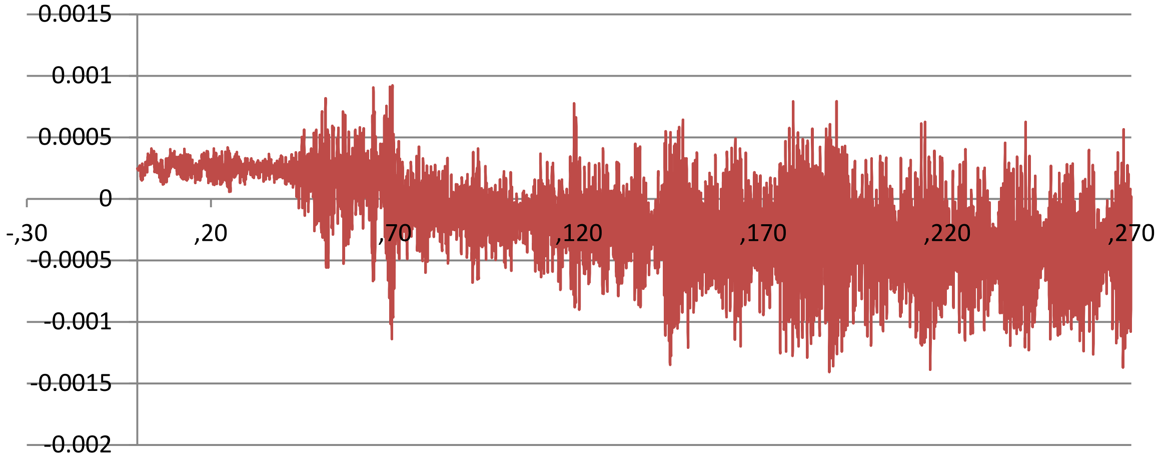
11.16.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	2.086E+01
Stress range	$\Delta\sigma$ [MPa]	3.603E+01
Damage		1.181E-06
Beam 304		
Max stress	σ_{max} [MPa]	3.330E+01
Stress range	$\Delta\sigma$ [MPa]	5.953E+01
Damage		3.297E-06
Beam 302		
Max stress	σ_{max} [MPa]	5.194E+01
Stress range	$\Delta\sigma$ [MPa]	1.186E+02
Damage		1.314E-04
Beam 288		
Max stress	σ_{max} [MPa]	4.648E+01
Stress range	$\Delta\sigma$ [MPa]	1.096E+02
Damage		7.081E-05
Beam 284		
Max stress	σ_{max} [MPa]	2.840E-04
Stress range	$\Delta\sigma$ [MPa]	2.089E+02
Damage		8.445E-06
Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	1.286E+02
Damage		1.635E-06

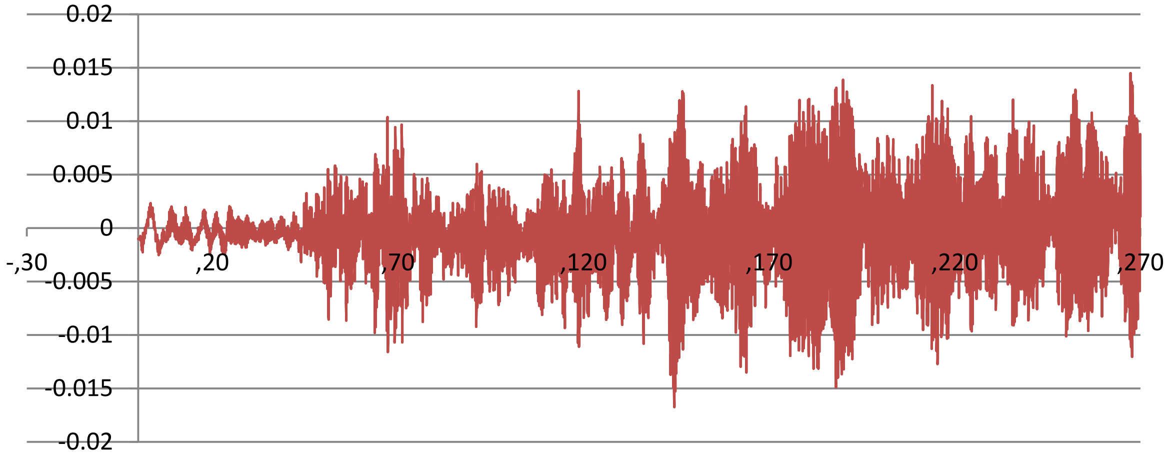


11.16.3 Bracer displacement

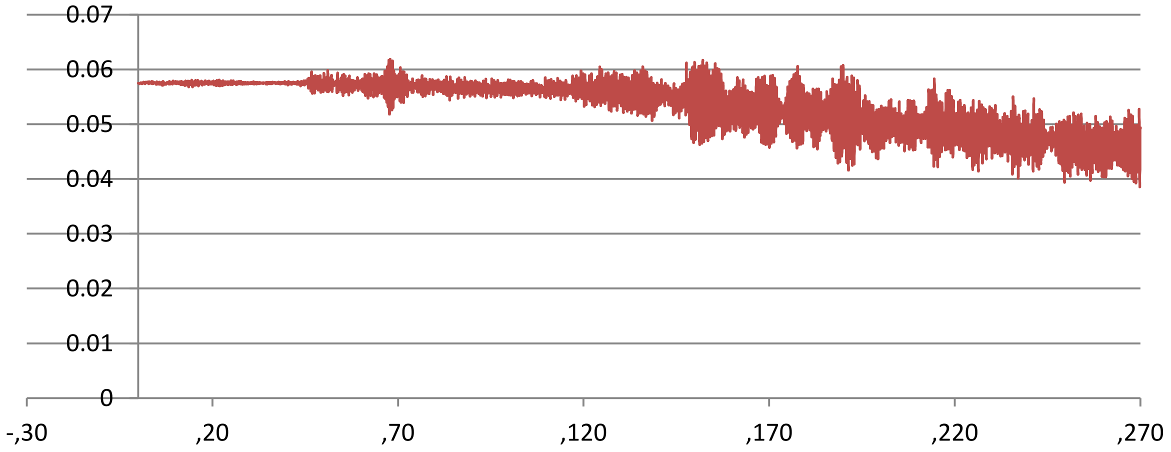
Upper Junction



Mid Junction

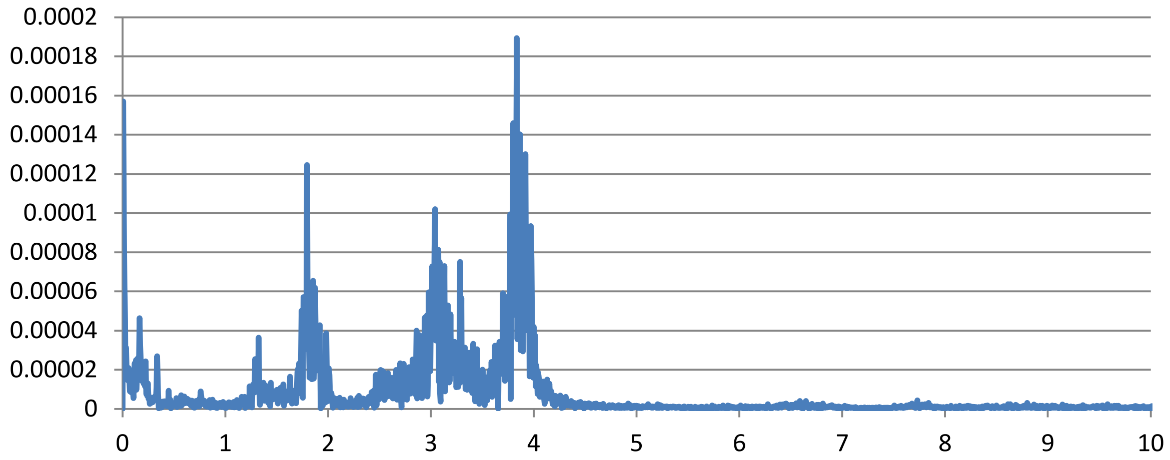


Lwr Junction

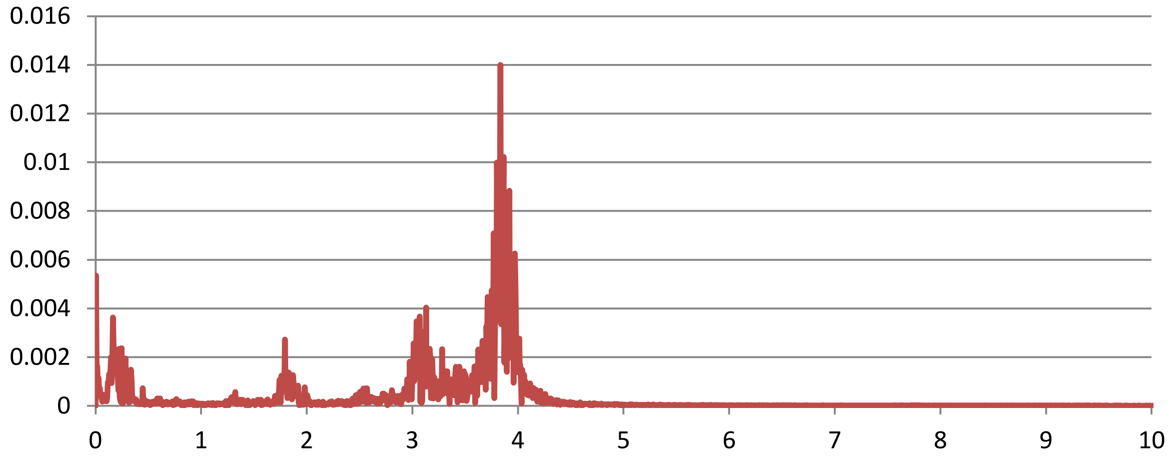


11.16.4 Bracer FFT Analysis

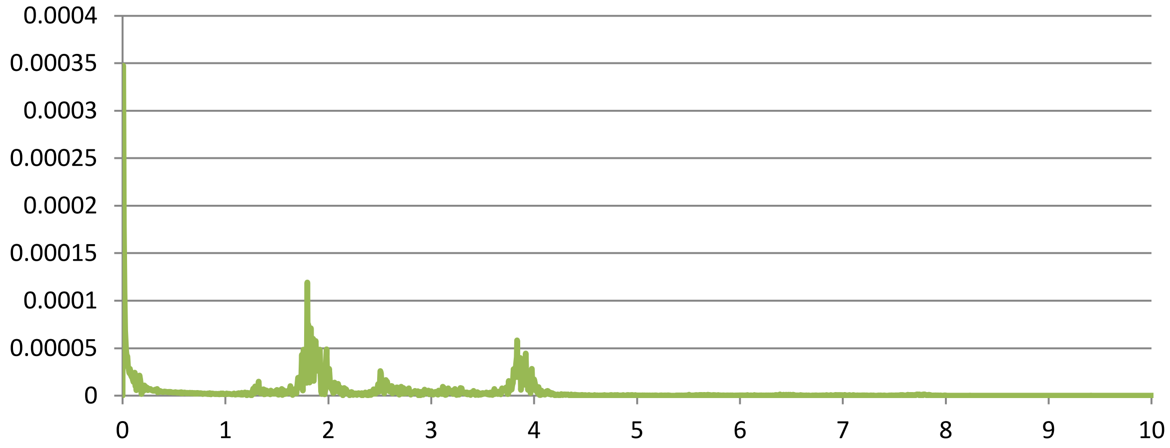
Upper Junction



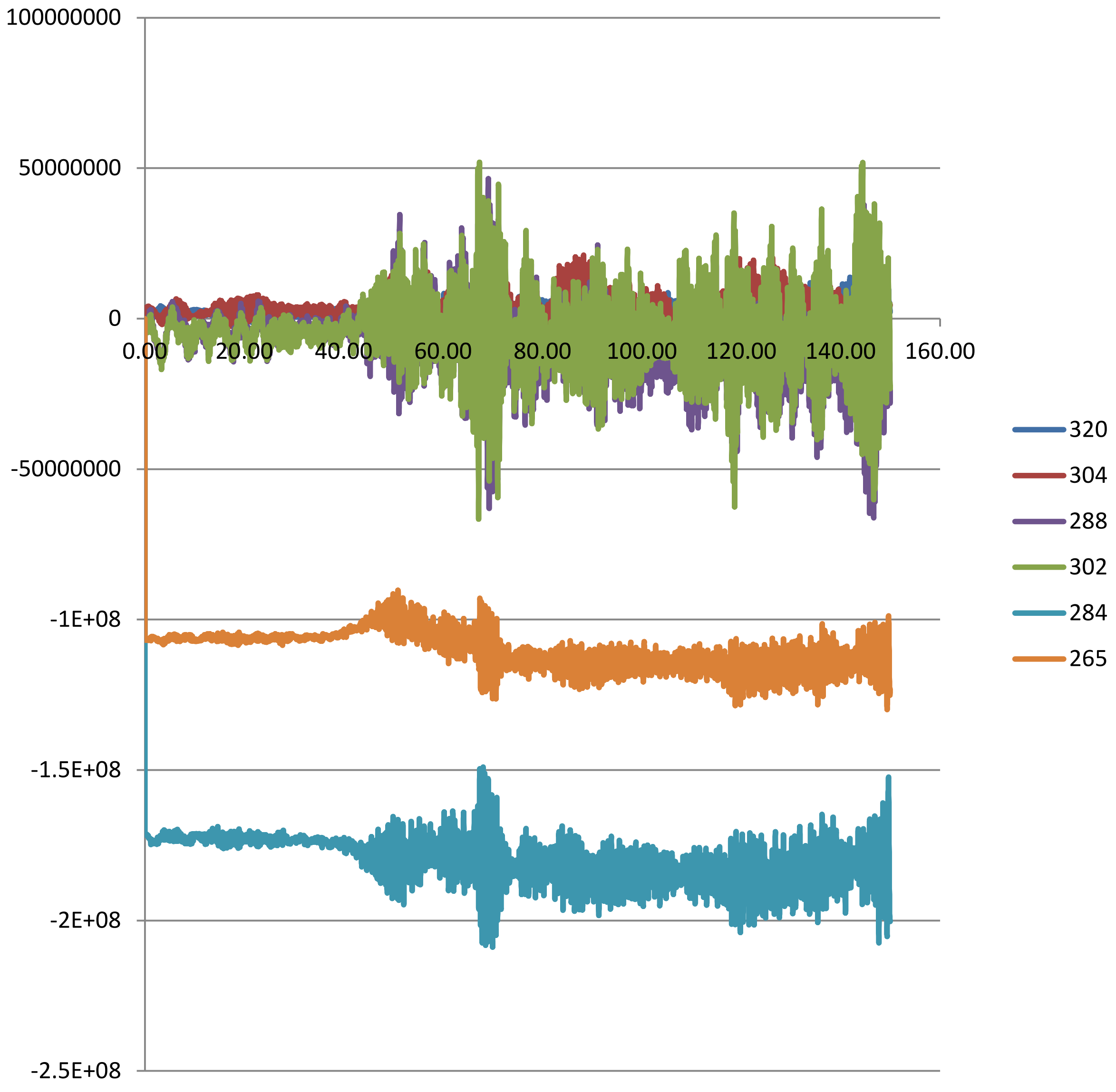
Mid Junction



Lwr Junction



11.16.5 Beam stresses



Max Beam Stresses

320	304	302	288	284	265	
2.09E+07	3.33E+07	5.19E+07	4.65E+07	2.84E+02	2.65E+02	*[Pa]*
20.862	33.305	51.940	46.481	0.000	0.000	*[MPa]*

Min Beam Stresses

320	304	302	288	284	265	
-1.52E+07	-2.62E+07	-6.66E+07	-6.31E+07	-2.09E+08	-1.29E+08	*[Pa]*
-15.164	-26.224	-66.614	-63.096	-208.864	-128.618	*[MPa]*

Delta sigma

320	304	302	288	284	265	
36.026	59.529	118.554	109.577	208.864	128.618	*[MPa]*

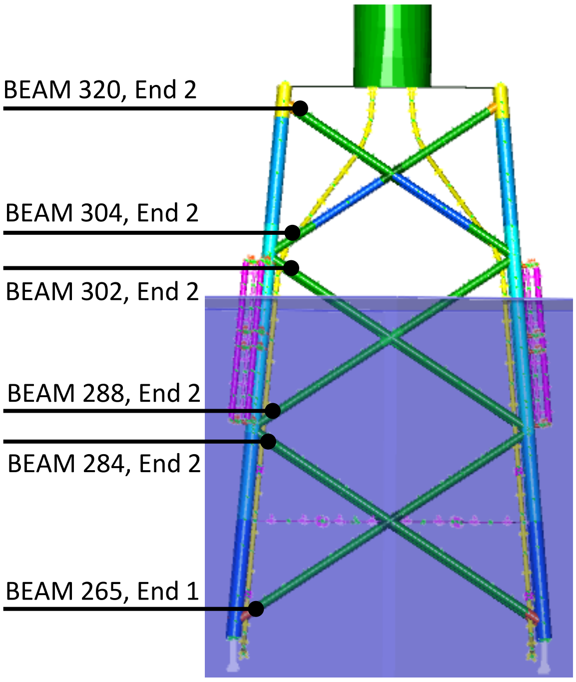
11.17 Sequentially Coupled Analysis- Event 1-C04-Sensitivity

11.17.1 Description

- Sensitivity analysis for damping [C] matrix. +20% of original value
- Fully coupled Usfos/Fedem analysis
- K, C, and M matrices replacing the interface node for Fedem analysis
  - DOF z is constrained in Fedem-Analysis
  - Tower and nacelle lumped mass in z direction for Usfos stress recovery analysis
- Linear springs along soil piles for soil properties
- Turbine setup in Fedem according to “Definition of a 5-MW Reference Wind Turbine for Offshore System Development”
  - J. Jonkman, S. Butterfield, W. Musial, and G. Scott
- Event 1

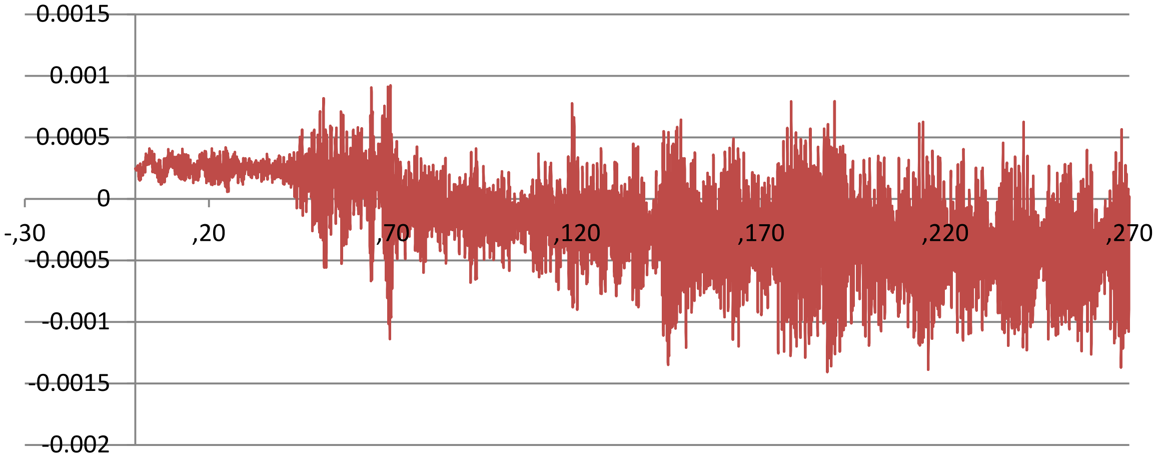
11.17.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	2.086E+01
Stress range	$\Delta\sigma$ [MPa]	3.603E+01
Damage		2.366E-06
Beam 304		
Max stress	σ_{max} [MPa]	3.330E+01
Stress range	$\Delta\sigma$ [MPa]	5.953E+01
Damage		6.632E-06
Beam 302		
Max stress	σ_{max} [MPa]	5.194E+01
Stress range	$\Delta\sigma$ [MPa]	1.186E+02
Damage		2.633E-04
Beam 288		
Max stress	σ_{max} [MPa]	4.648E+01
Stress range	$\Delta\sigma$ [MPa]	1.096E+02
Damage		1.417E-04
Beam 284		
Max stress	σ_{max} [MPa]	2.840E-04
Stress range	$\Delta\sigma$ [MPa]	2.089E+02
Damage		1.704E-05
Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	1.286E+02
Damage		3.303E-06

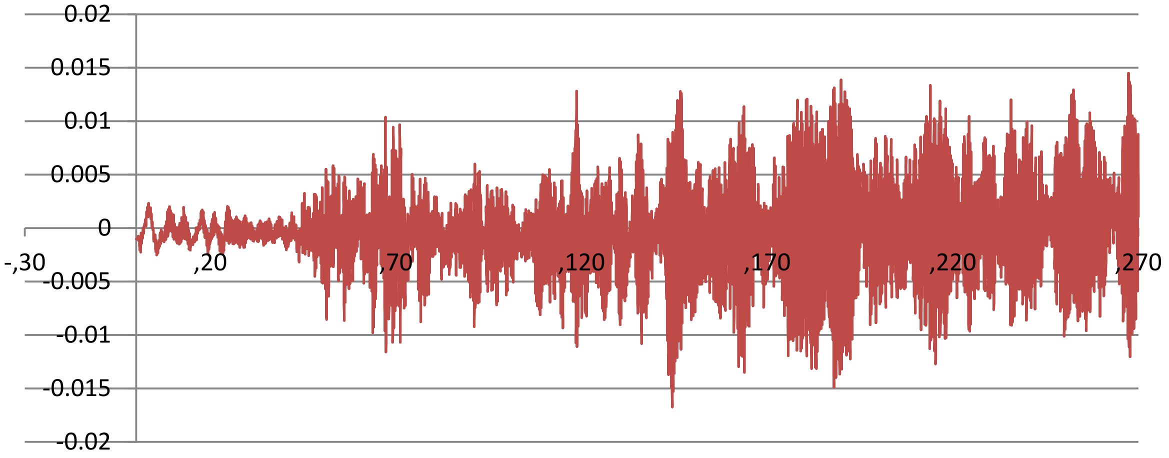


11.17.3 Bracer displacement

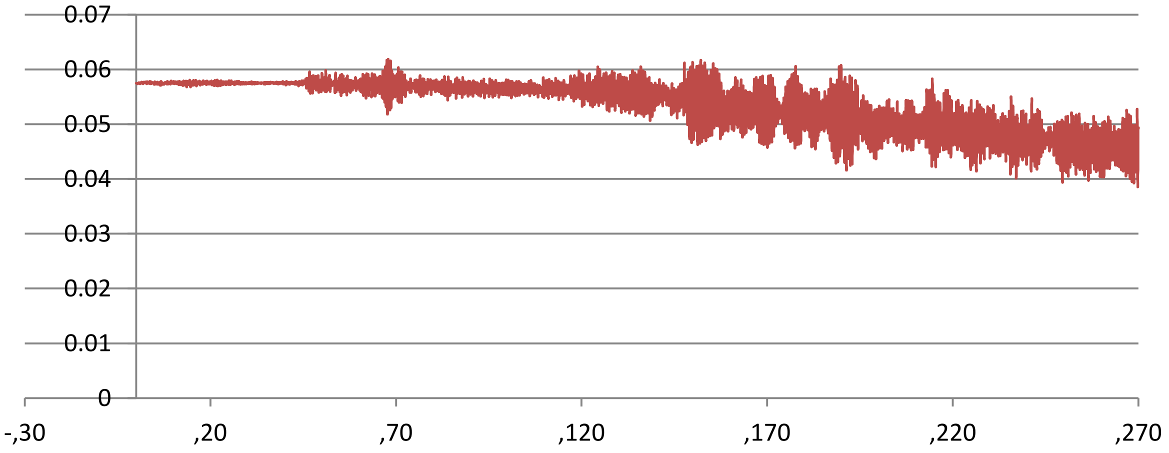
Upper Junction



Mid Junction

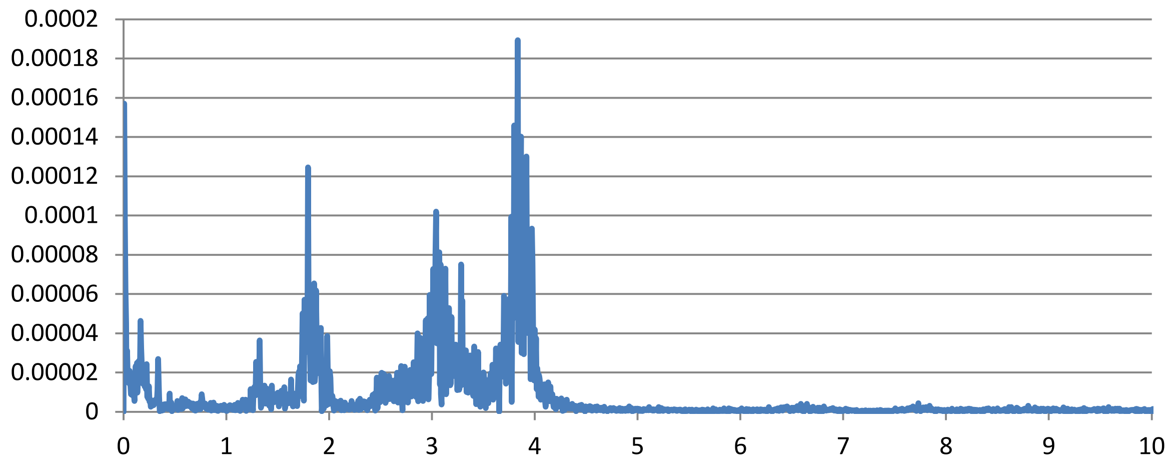


Lwr Junction

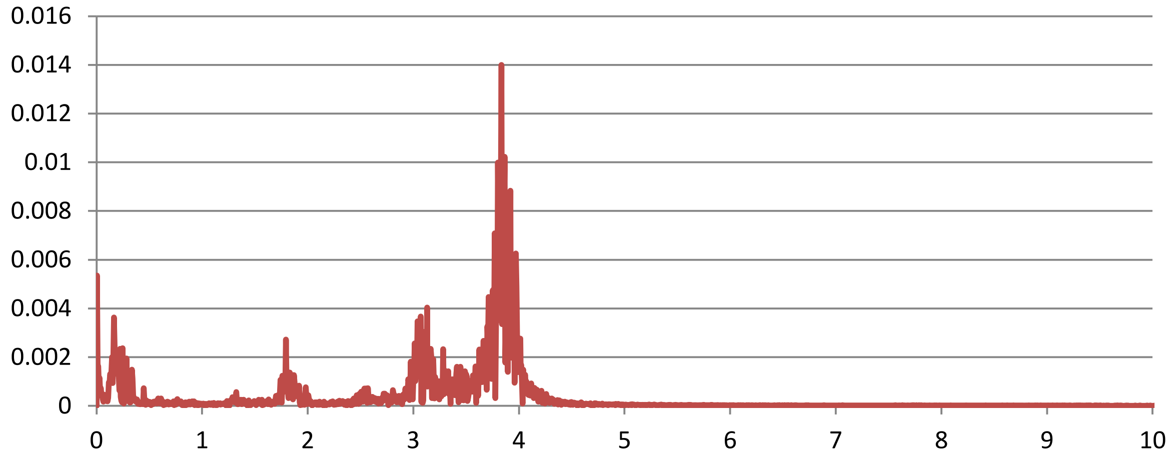


11.17.4 Bracer FFT Analysis

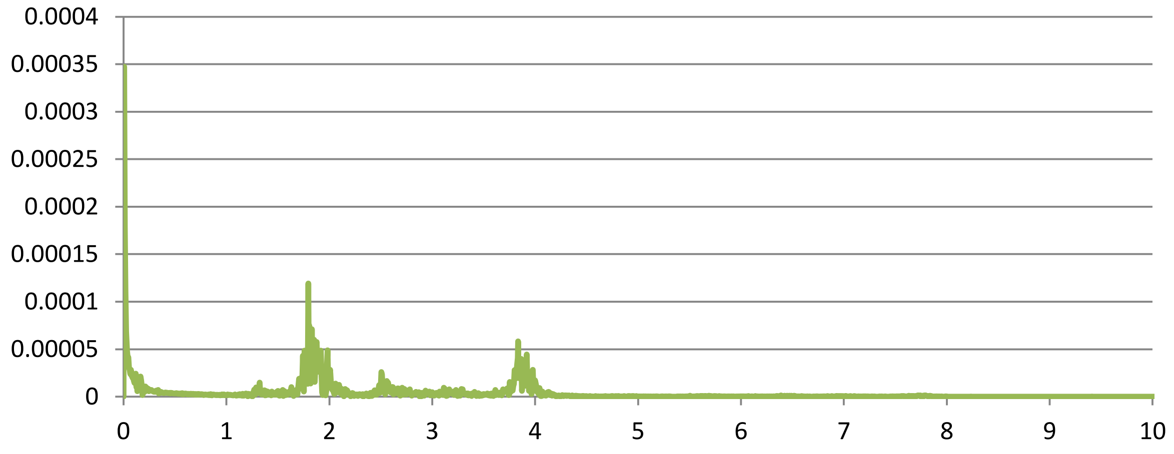
Upper Junction



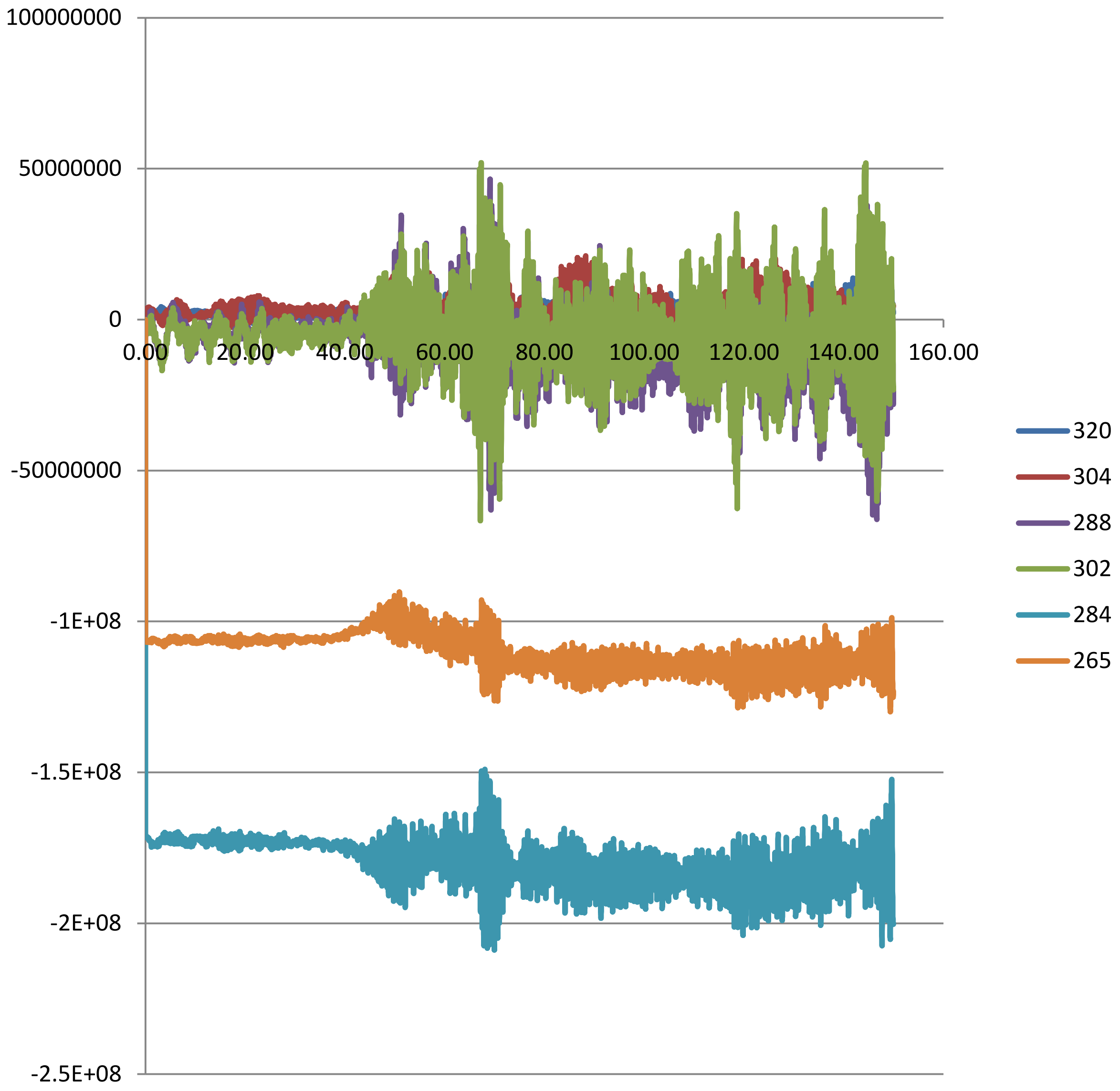
Mid Junction



Lwr Junction



11.17.5 Beam stresses



Max Beam Stresses

320	304	302	288	284	265	
2.09E+07	3.33E+07	5.19E+07	4.65E+07	2.84E+02	2.65E+02	*[Pa]*
20.862	33.305	51.940	46.481	0.000	0.000	*[MPa]*

Min Beam Stresses

320	304	302	288	284	265	
-1.52E+07	-2.62E+07	-6.66E+07	-6.31E+07	-2.09E+08	-1.29E+08	*[Pa]*
-15.164	-26.224	-66.614	-63.096	-208.864	-128.618	*[MPa]*

Delta sigma

320	304	302	288	284	265	
36.026	59.529	118.554	109.577	208.864	128.618	*[MPa]*

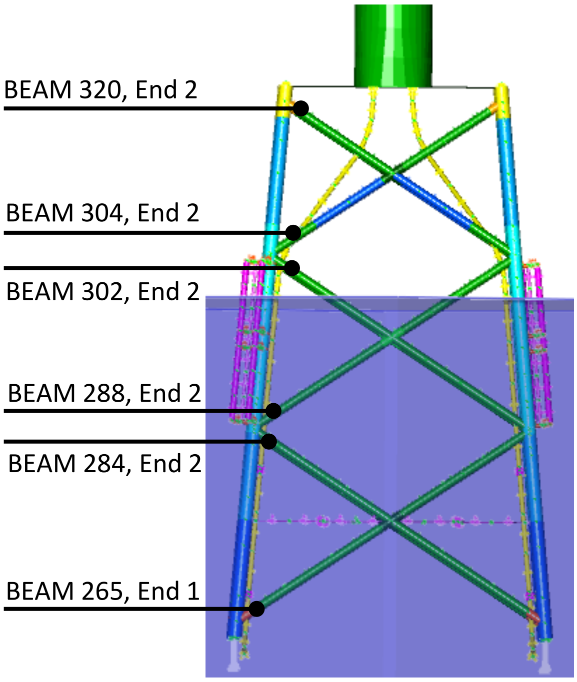
11.18 Sequentially Coupled Analysis- Event 1-M01-Sensitivity

11.18.1 Description

- Sensitivity analysis for mass [M] matrix. -20% of original value
- Fully coupled Usfos/Fedem analysis
- K, C, and M matrices replacing the interface node for Fedem analysis
  - DOF z is constrained in Fedem-Analysis
  - Tower and nacelle lumped mass in z direction for Usfos stress recovery analysis
- Linear springs along soil piles for soil properties
- Turbine setup in Fedem according to “Definition of a 5-MW Reference Wind Turbine for Offshore System Development”
  - J. Jonkman, S. Butterfield, W. Musial, and G. Scott
- Event 1

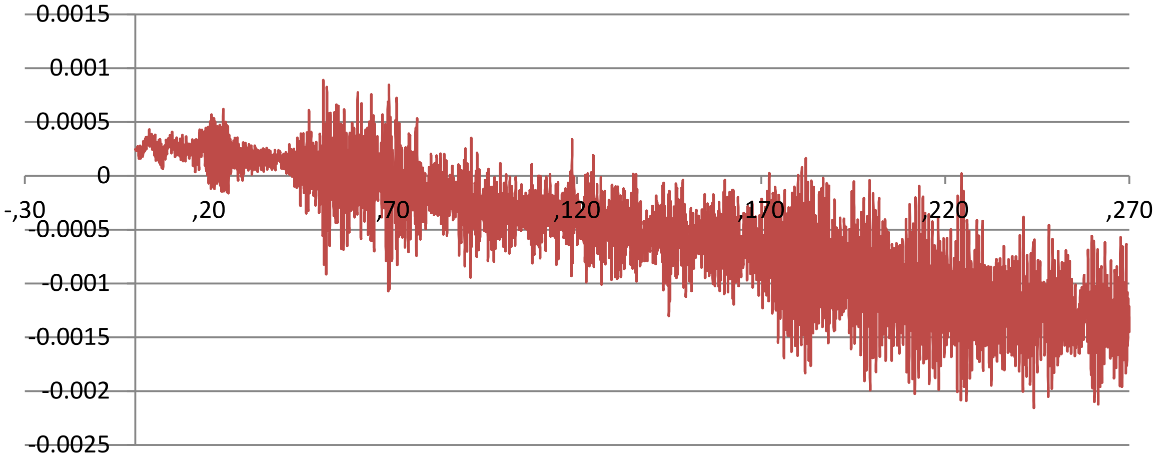
11.18.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	2.486E+01
Stress range	$\Delta\sigma$ [MPa]	3.701E+01
Damage		1.226E-06
Beam 304		
Max stress	σ_{max} [MPa]	3.671E+01
Stress range	$\Delta\sigma$ [MPa]	6.702E+01
Damage		8.278E-06
Beam 302		
Max stress	σ_{max} [MPa]	5.012E+01
Stress range	$\Delta\sigma$ [MPa]	1.068E+02
Damage		6.040E-05
Beam 288		
Max stress	σ_{max} [MPa]	3.110E+01
Stress range	$\Delta\sigma$ [MPa]	7.993E+01
Damage		2.519E-05
Beam 284		
Max stress	σ_{max} [MPa]	2.840E-04
Stress range	$\Delta\sigma$ [MPa]	1.956E+02
Damage		2.029E-05
Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	1.240E+02
Damage		5.293E-06

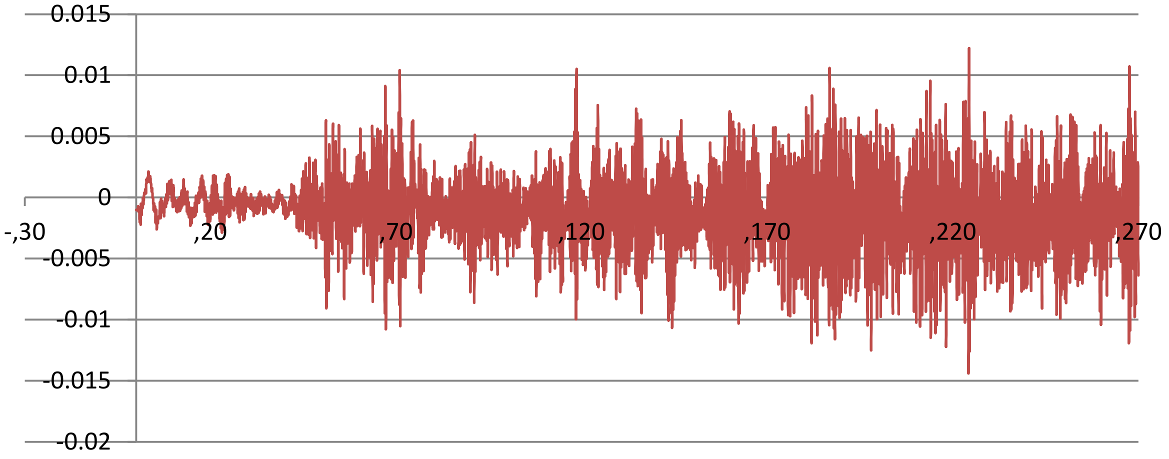


11.18.3 Bracer displacement

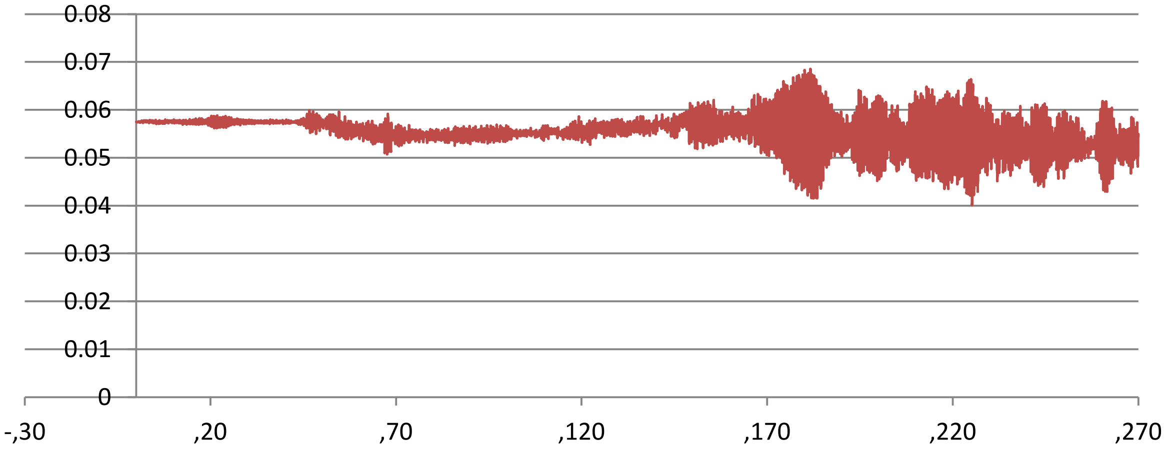
Upper Junction



Mid Junction

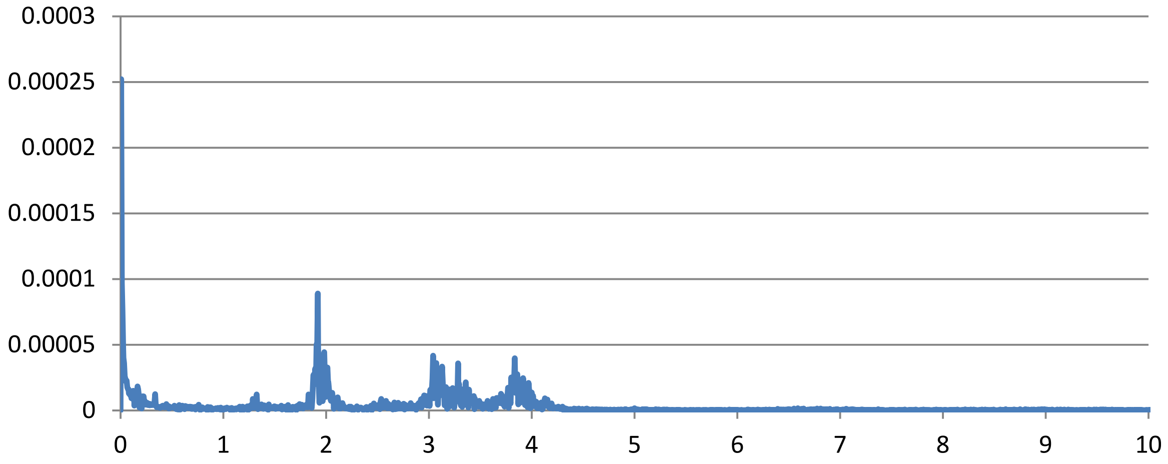


Lwr Junction

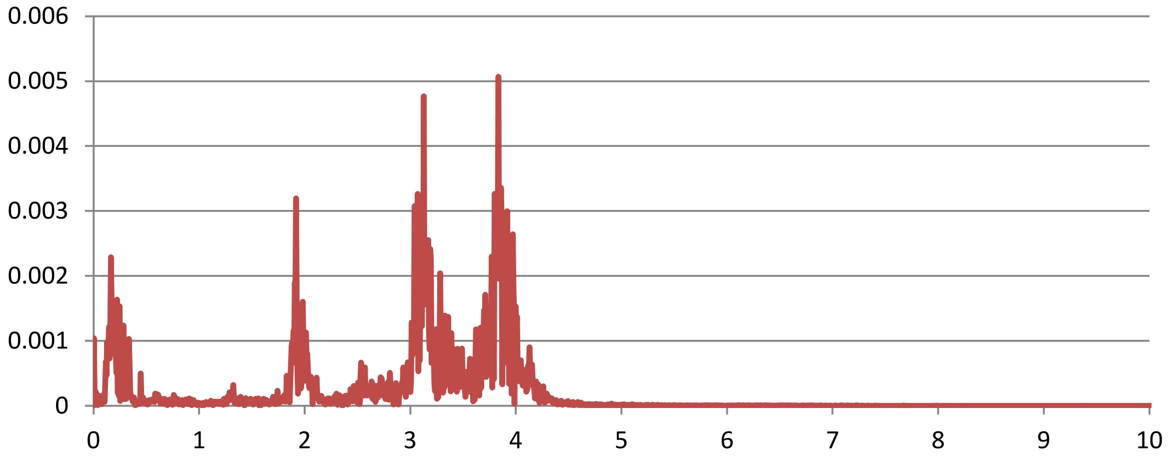


11.18.4 Bracer FFT Analysis

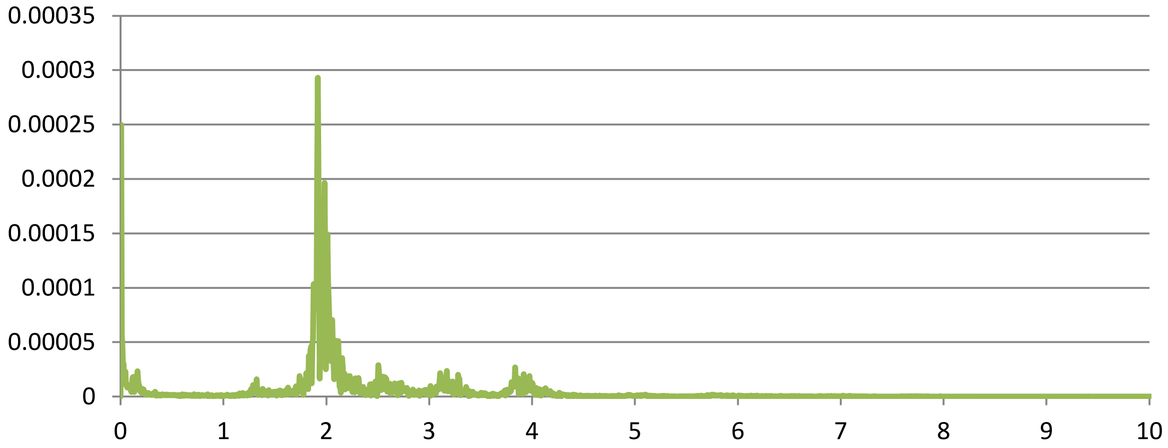
Upper Junction



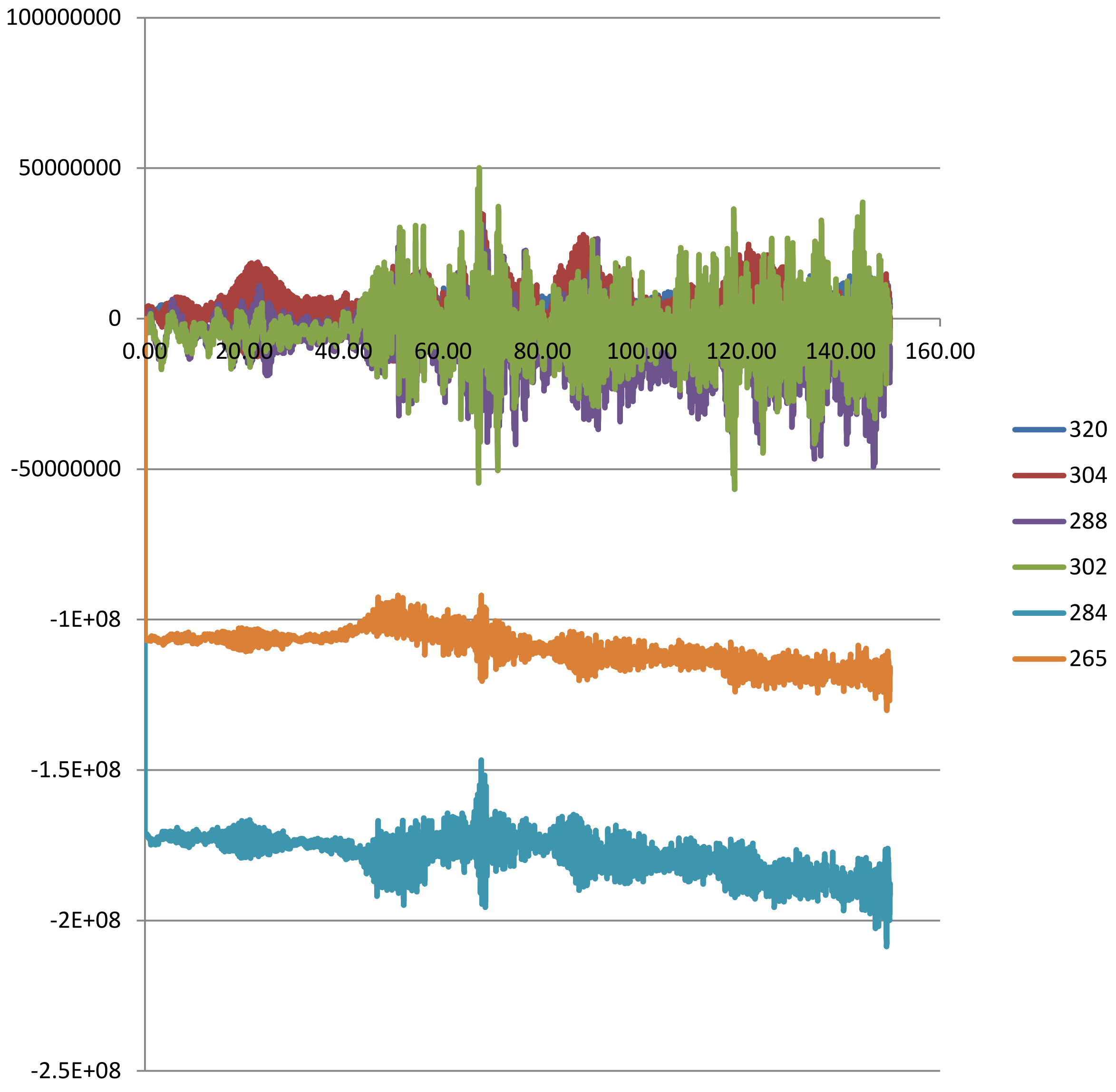
Mid Junction



Lwr Junction



11.18.5 Beam stresses



Max Beam Stresses

320	304	302	288	284	265	
2.49E+07	3.67E+07	5.01E+07	3.11E+07	2.84E+02	2.65E+02	*[Pa]*
24.861	36.706	50.122	31.102	0.000	0.000	*[MPa]*

Min Beam Stresses

320	304	302	288	284	265	
-1.22E+07	-3.03E+07	-5.67E+07	-4.88E+07	-1.96E+08	-1.24E+08	*[Pa]*
-12.154	-30.318	-56.669	-48.829	-195.626	-123.992	*[MPa]*

Delta sigma

320	304	302	288	284	265	
37.014	67.024	106.791	79.931	195.626	123.992	*[MPa]*

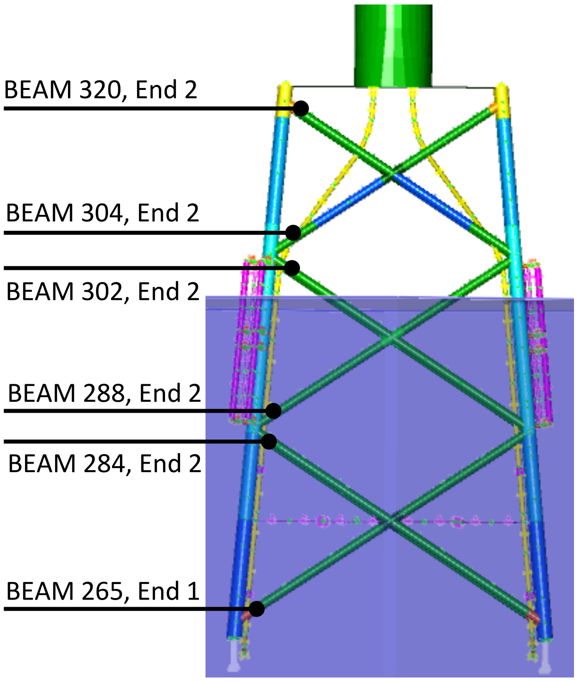
11.19 Sequentially Coupled Analysis- Event 1-M02-Sensitivity

11.19.1 Description

- Sensitivity analysis for mass [M] matrix. -10% of original value
- Fully coupled Usfos/Fedem analysis
- K, C, and M matrices replacing the interface node for Fedem analysis
    - DOF z is constrained in Fedem-Analysis
    - Tower and nacelle lumped mass in z direction for Usfos stress recovery analysis
- Linear springs along soil piles for soil properties
- Turbine setup in Fedem according to “Definition of a 5-MW Reference Wind Turbine for Offshore System Development”
    - J. Jonkman, S. Butterfield, W. Musial, and G. Scott
- Event 1

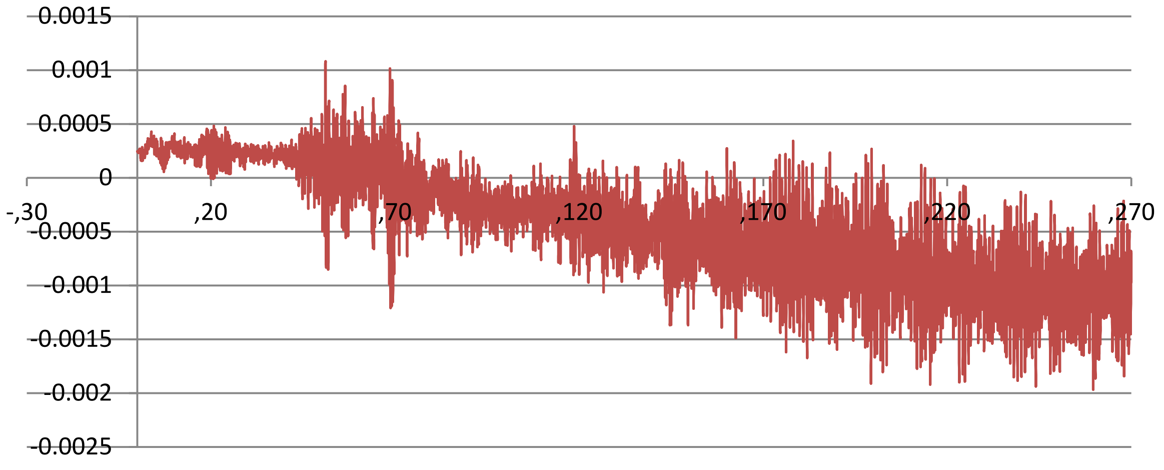
11.19.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	2.353E+01
Stress range	$\Delta\sigma$ [MPa]	3.830E+01
Damage		1.437E-06
Beam 304		
Max stress	σ_{max} [MPa]	3.617E+01
Stress range	$\Delta\sigma$ [MPa]	6.673E+01
Damage		6.799E-06
Beam 302		
Max stress	σ_{max} [MPa]	5.340E+01
Stress range	$\Delta\sigma$ [MPa]	1.148E+02
Damage		8.869E-05
Beam 288		
Max stress	σ_{max} [MPa]	3.658E+01
Stress range	$\Delta\sigma$ [MPa]	9.017E+01
Damage		3.704E-05
Beam 284		
Max stress	σ_{max} [MPa]	2.840E-04
Stress range	$\Delta\sigma$ [MPa]	2.066E+02
Damage		1.568E-05
Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	1.298E+02
Damage		3.486E-06

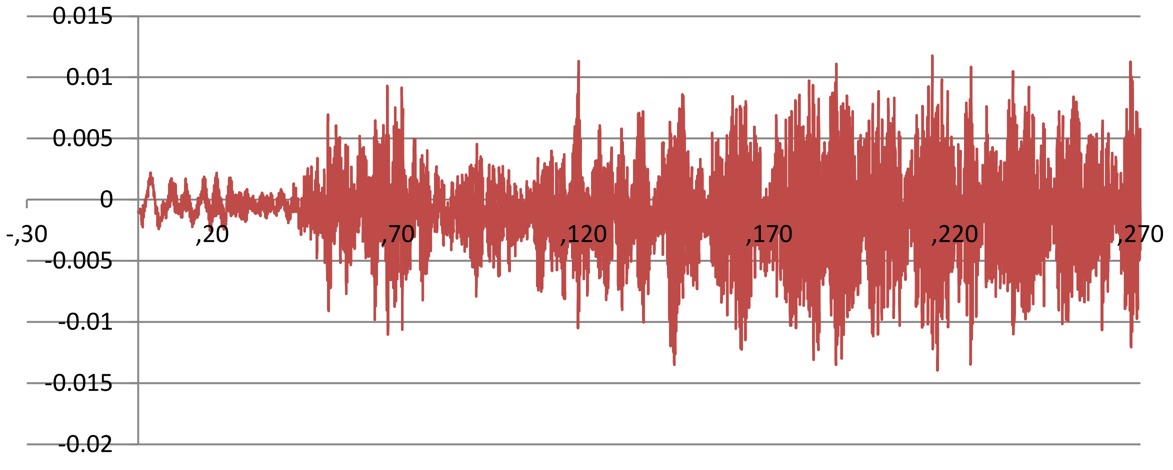


11.19.3 Bracer displacement

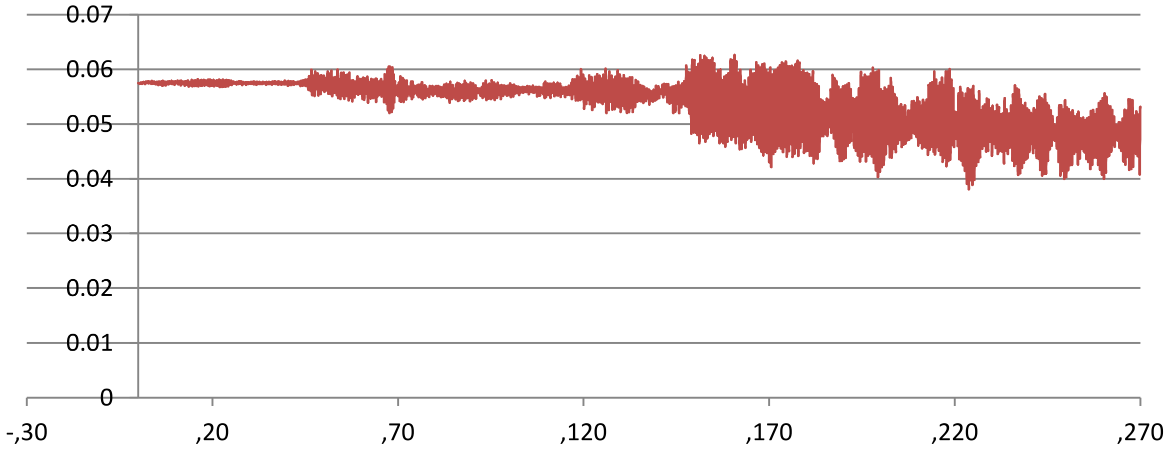
Upper Junction



Mid Junction

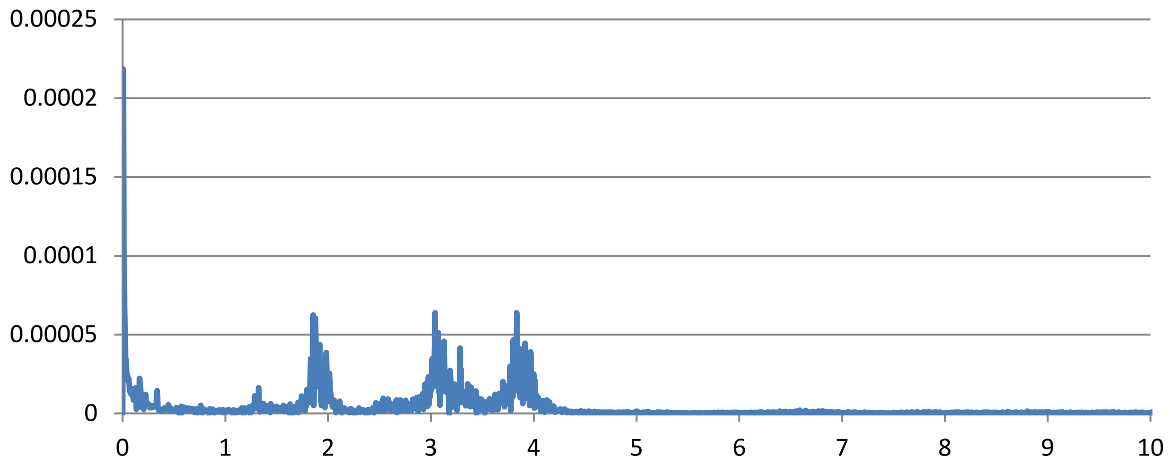


Lwr Junction

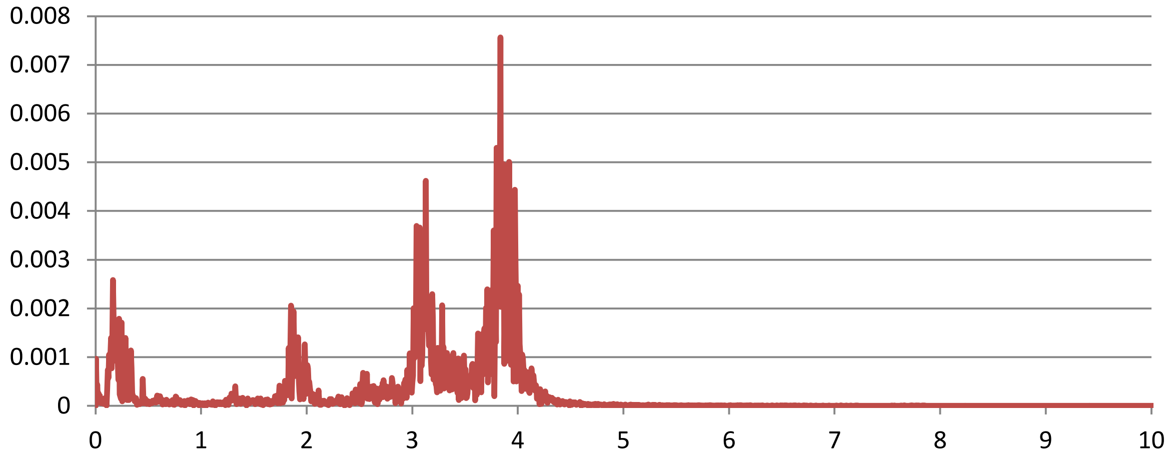


11.19.4 Bracer FFT Analysis

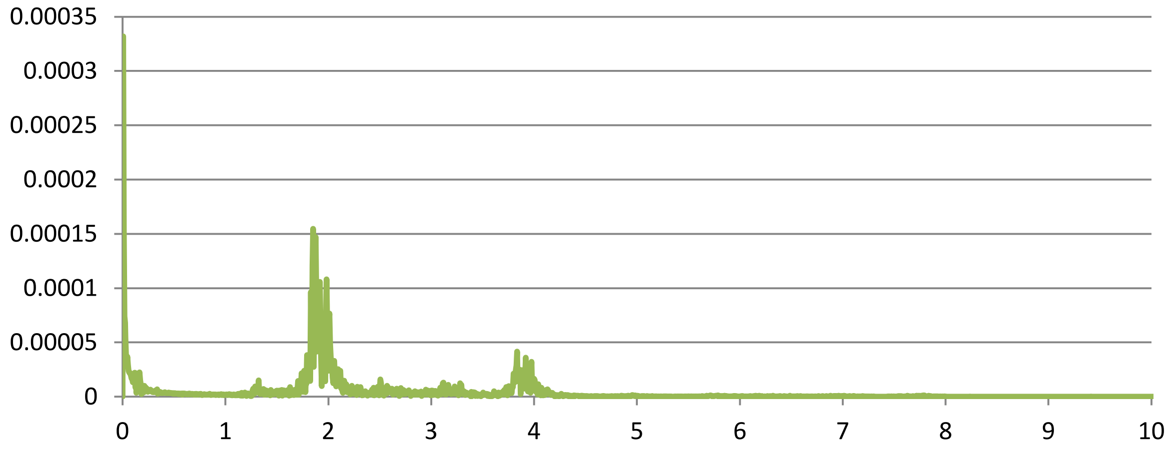
Upper Junction



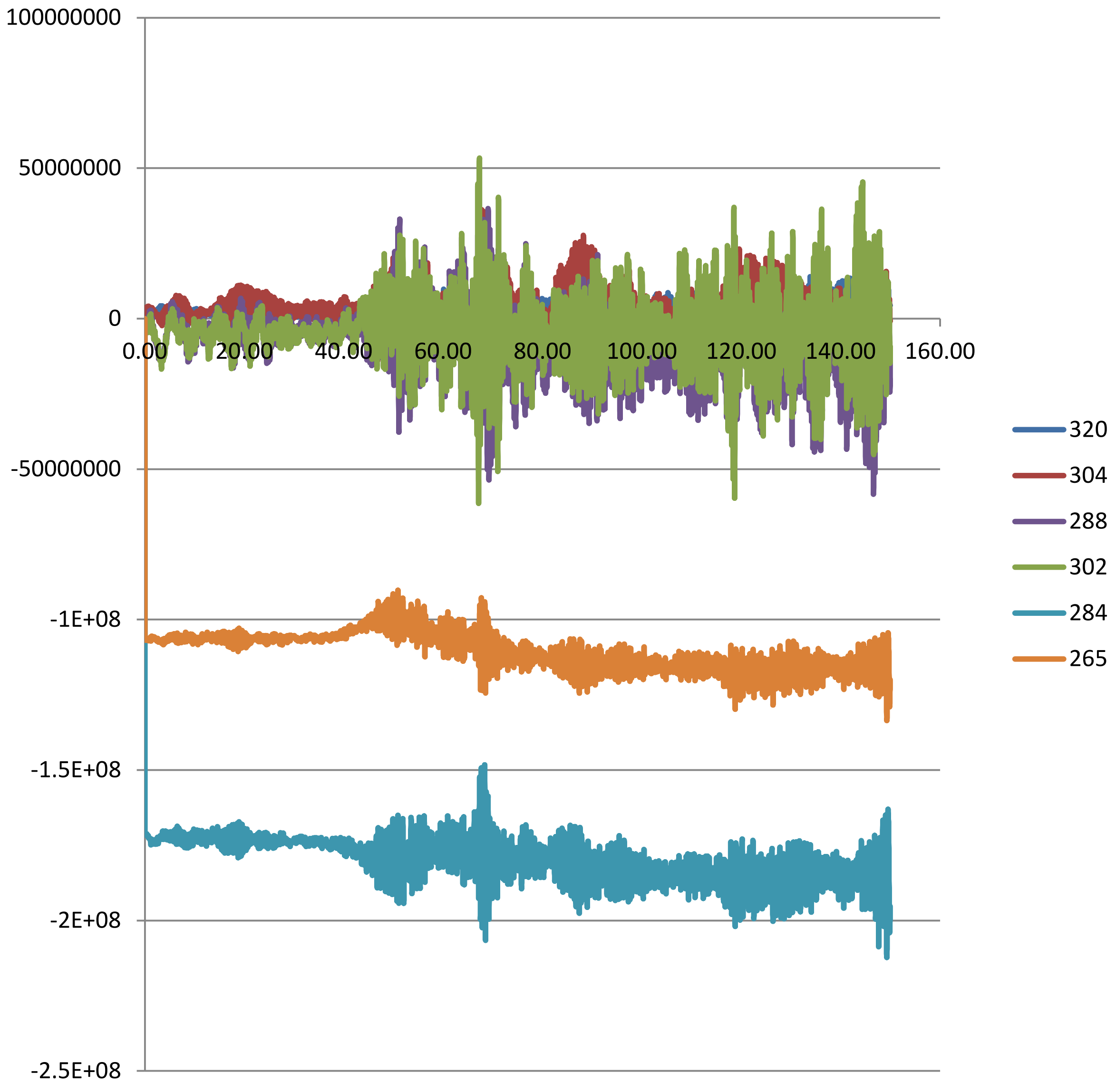
Mid Junction



Lwr Junction



11.19.5 Beam stresses



Max Beam Stresses

320	304	302	288	284	265	
2.35E+07	3.62E+07	5.34E+07	3.66E+07	2.84E+02	2.65E+02	*[Pa]*
23.534	36.166	53.403	36.577	0.000	0.000	*[MPa]*

Min Beam Stresses

320	304	302	288	284	265	
-1.48E+07	-3.06E+07	-6.14E+07	-5.36E+07	-2.07E+08	-1.30E+08	*[Pa]*
-14.762	-30.560	-61.369	-53.592	-206.591	-129.823	*[MPa]*

Delta sigma

320	304	302	288	284	265	
38.295	66.726	114.772	90.170	206.592	129.824	*[MPa]*

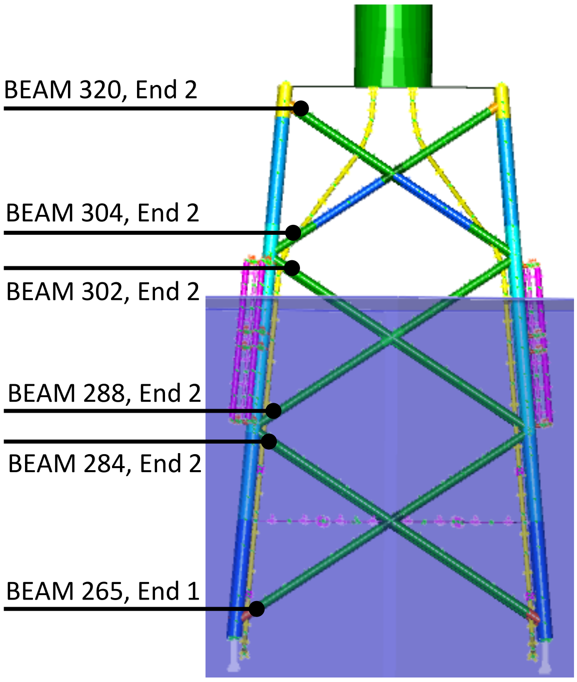
11.20 Sequentially Coupled Analysis- Event 1-M03-Sensitivity

11.20.1 Description

- Sensitivity analysis for mass [M] matrix. +s10% of original value
- Fully coupled Usfos/Fedem analysis
- K, C, and M matrices replacing the interface node for Fedem analysis
    - DOF z is constrained in Fedem-Analysis
    - Tower and nacelle lumped mass in z direction for Usfos stress recovery analysis
- Linear springs along soil piles for soil properties
- Turbine setup in Fedem according to “Definition of a 5-MW Reference Wind Turbine for Offshore System Development”
    - J. Jonkman, S. Butterfield, W. Musial, and G. Scott
- Event 1

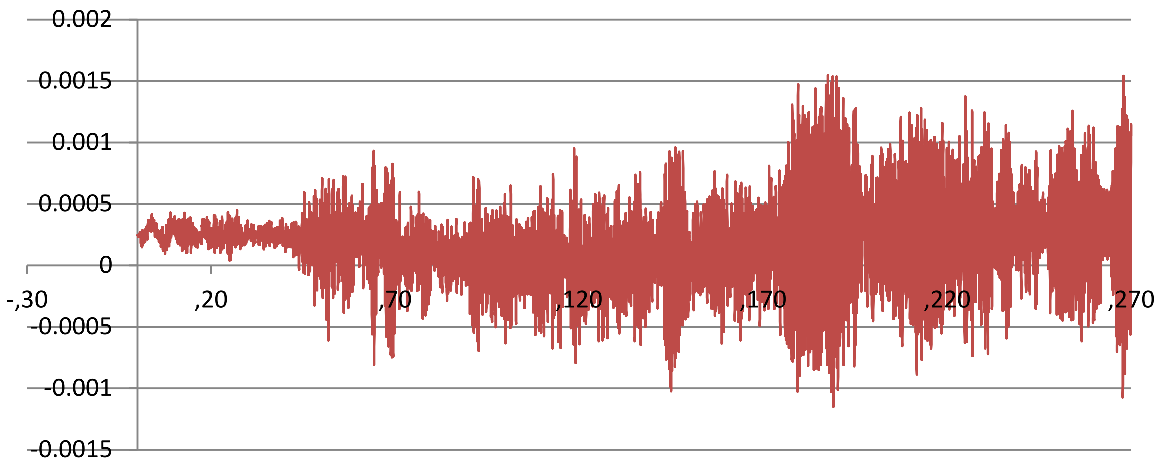
11.20.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	1.952E+01
Stress range	$\Delta\sigma$ [MPa]	3.420E+01
Damage		4.946E-07
Beam 304		
Max stress	σ_{max} [MPa]	3.172E+01
Stress range	$\Delta\sigma$ [MPa]	5.073E+01
Damage		1.530E-06
Beam 302		
Max stress	σ_{max} [MPa]	5.363E+01
Stress range	$\Delta\sigma$ [MPa]	1.145E+02
Damage		1.178E-04
Beam 288		
Max stress	σ_{max} [MPa]	4.521E+01
Stress range	$\Delta\sigma$ [MPa]	1.091E+02
Damage		1.246E-04
Beam 284		
Max stress	σ_{max} [MPa]	2.840E-04
Stress range	$\Delta\sigma$ [MPa]	2.069E+02
Damage		3.545E-06
Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	1.237E+02
Damage		9.023E-07

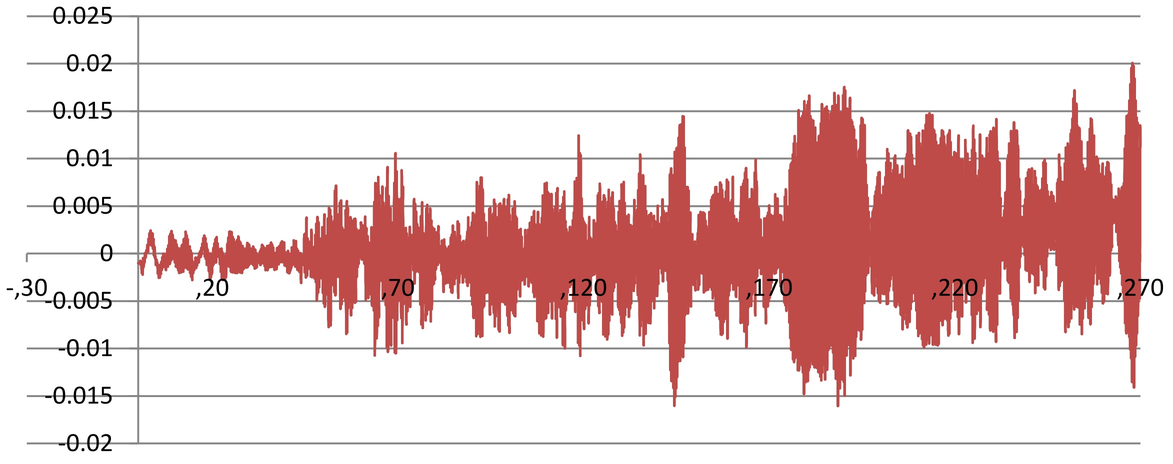


11.20.3 Bracer displacement

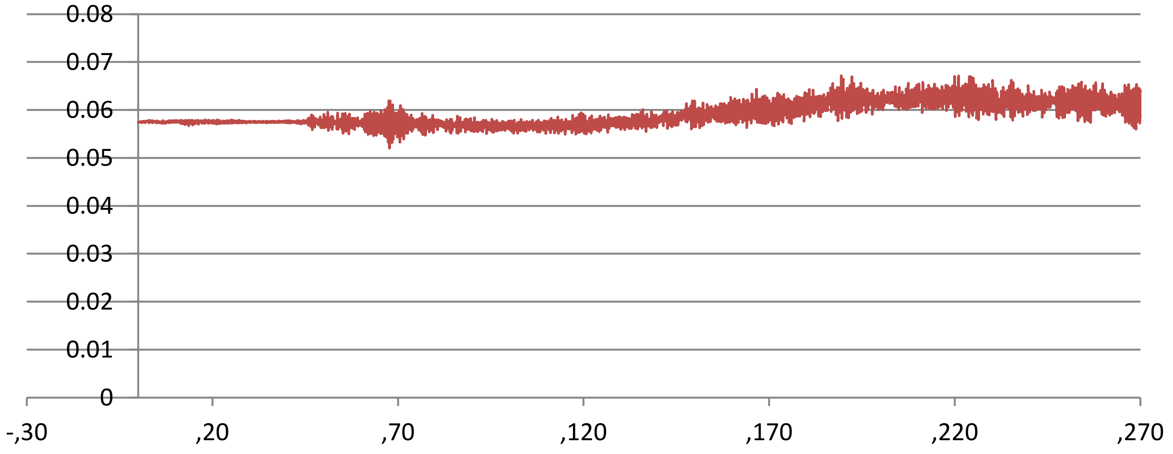
Upper Junction



Mid Junction

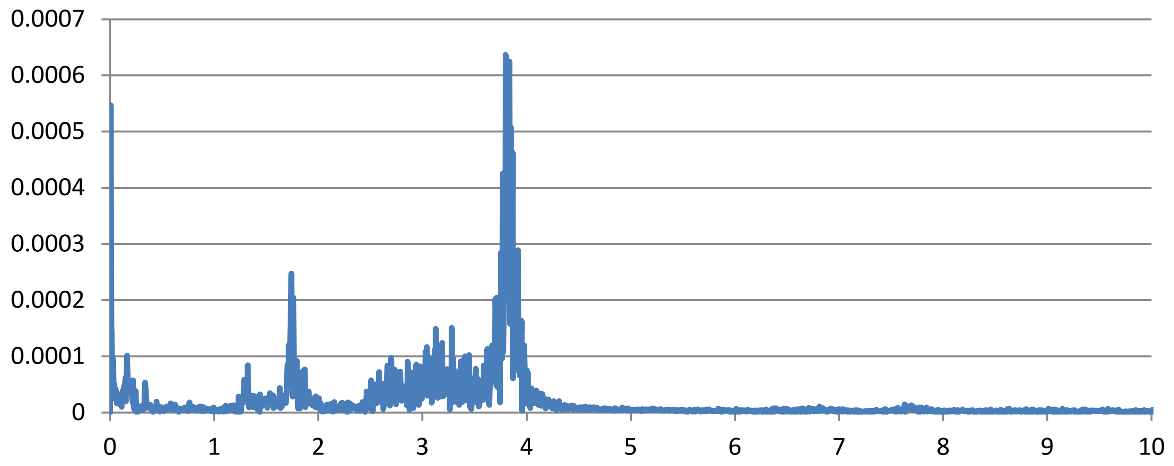


Lwr Junction

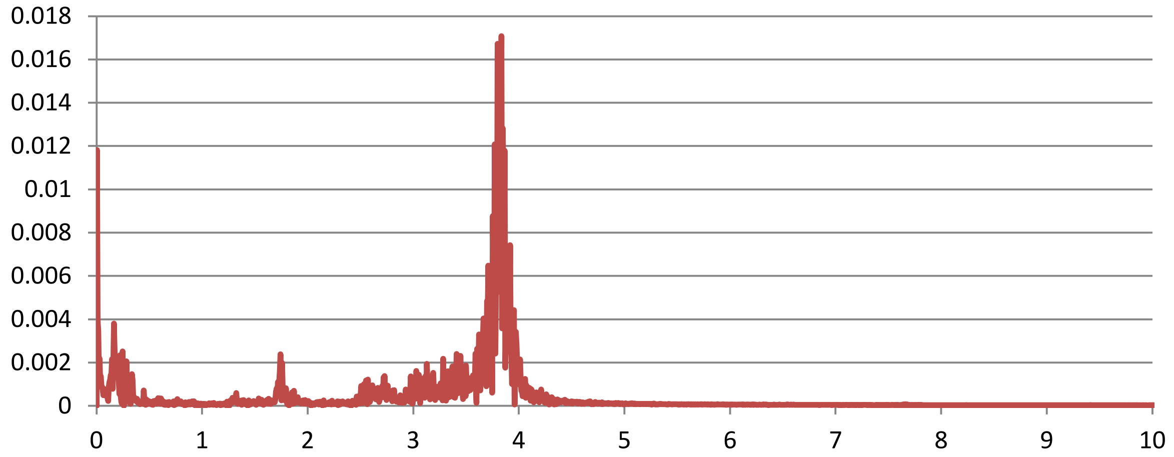


11.20.4 Bracer FFT Analysis

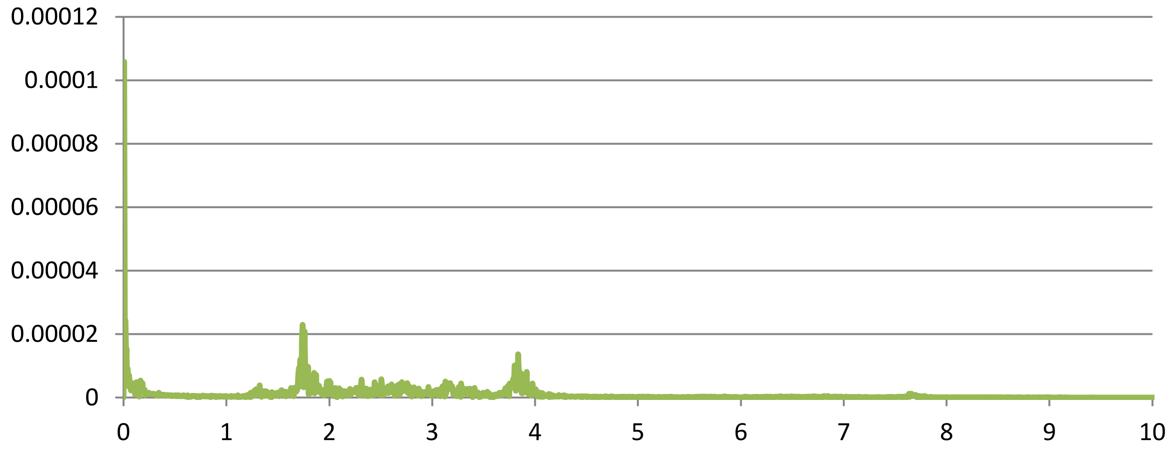
Upper Junction



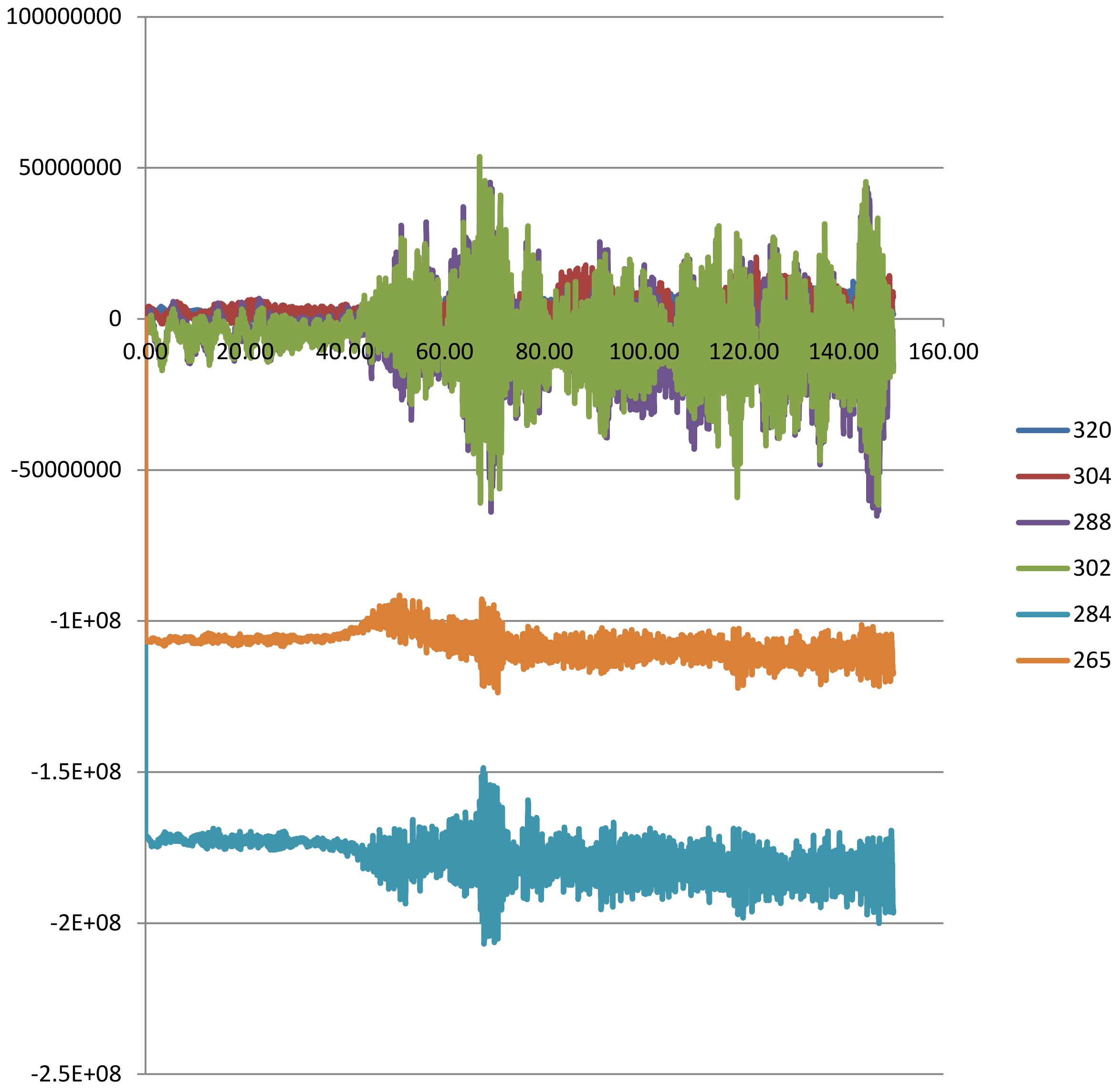
Mid Junction



Lwr Junction



11.20.5 Beam stresses



Max Beam Stresses

320	304	302	288	284	265	
1.95E+07	3.17E+07	5.36E+07	4.52E+07	2.84E+02	2.65E+02	*[Pa]*
19.515	31.725	53.633	45.207	0.000	0.000	*[MPa]*

Min Beam Stresses

320	304	302	288	284	265	
-1.47E+07	-1.90E+07	-6.09E+07	-6.39E+07	-2.07E+08	-1.24E+08	*[Pa]*
-14.680	-19.004	-60.911	-63.893	-206.929	-123.708	*[MPa]*

Delta sigma

320	304	302	288	284	265	
34.195	50.729	114.544	109.100	206.929	123.708	*[MPa]*

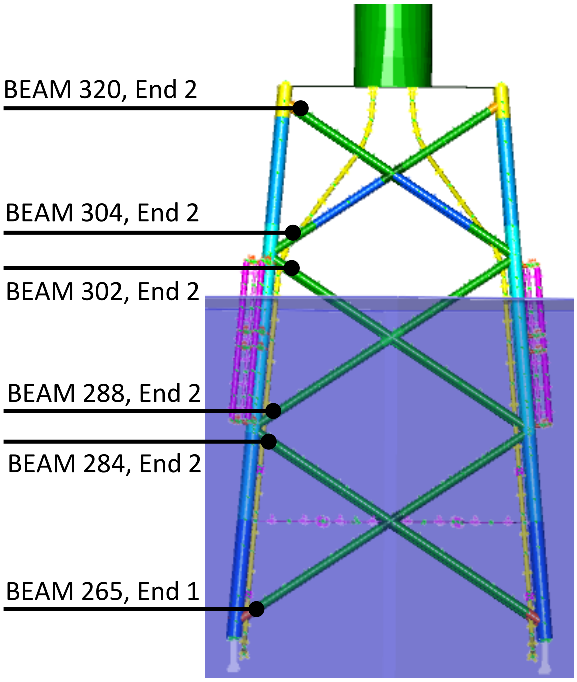
11.21 Sequentially Coupled Analysis- Event 1-M04-Sensitivity

11.21.1 Description

- Sensitivity analysis for mass [M] matrix. +20% of original value
- Fully coupled Usfos/Fedem analysis
- K, C, and M matrices replacing the interface node for Fedem analysis
    - DOF z is constrained in Fedem-Analysis
    - Tower and nacelle lumped mass in z direction for Usfos stress recovery analysis
- Linear springs along soil piles for soil properties
- Turbine setup in Fedem according to “Definition of a 5-MW Reference Wind Turbine for Offshore System Development”
    - J. Jonkman, S. Butterfield, W. Musial, and G. Scott
- Event 1

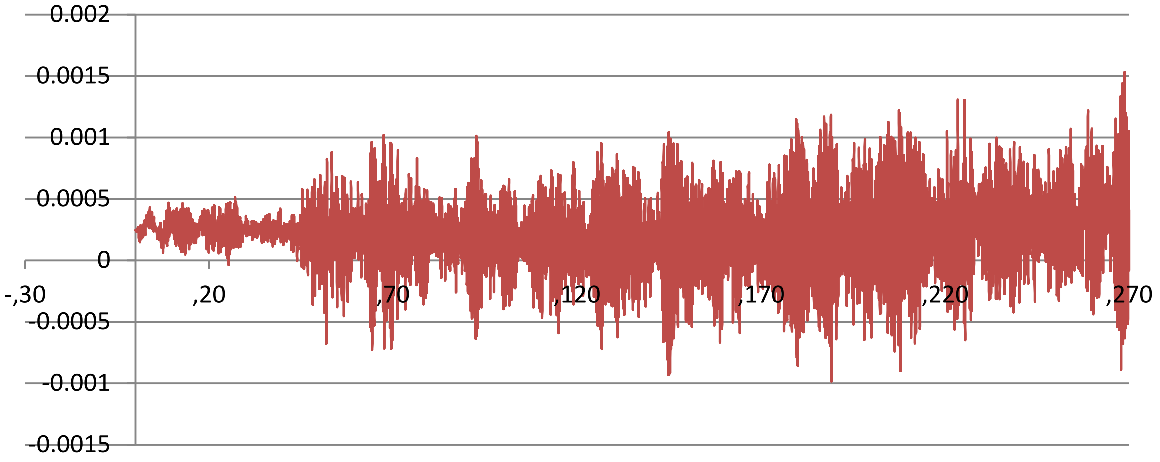
11.21.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	1.826E+01
Stress range	$\Delta\sigma$ [MPa]	2.909E+01
Damage		2.477E-07
Beam 304		
Max stress	σ_{max} [MPa]	2.527E+01
Stress range	$\Delta\sigma$ [MPa]	3.813E+01
Damage		1.121E-06
Beam 302		
Max stress	σ_{max} [MPa]	4.628E+01
Stress range	$\Delta\sigma$ [MPa]	1.020E+02
Damage		8.213E-05
Beam 288		
Max stress	σ_{max} [MPa]	4.618E+01
Stress range	$\Delta\sigma$ [MPa]	1.054E+02
Damage		1.015E-04
Beam 284		
Max stress	σ_{max} [MPa]	2.840E-04
Stress range	$\Delta\sigma$ [MPa]	1.978E+02
Damage		2.879E-06
Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	1.196E+02
Damage		6.935E-07

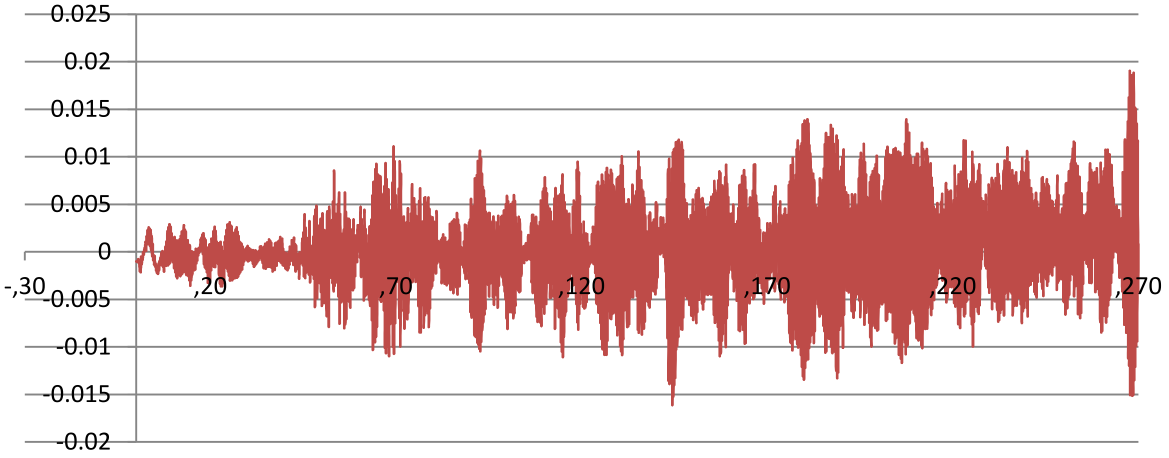


11.21.3 Bracer displacement

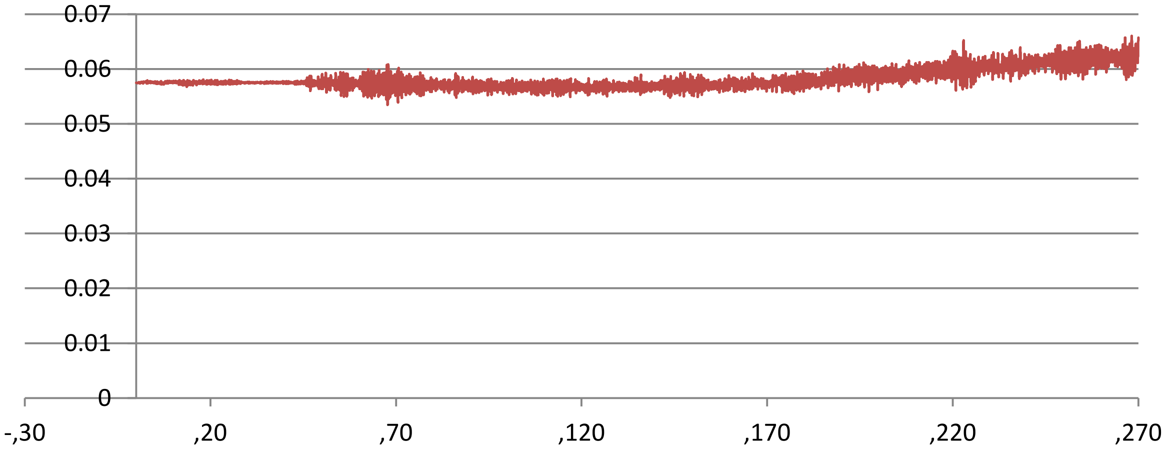
Upper Junction



Mid Junction

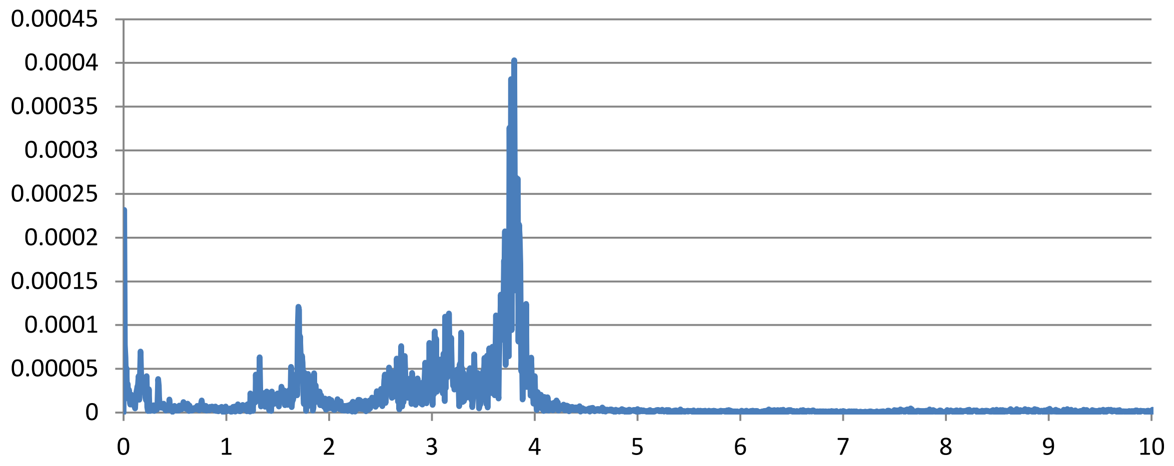


Lwr Junction

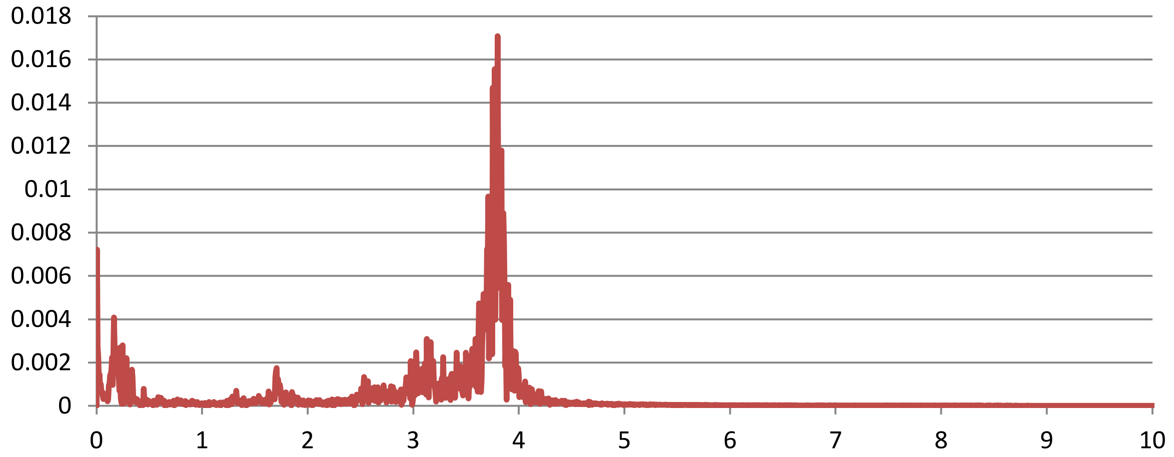


11.21.4 Bracer FFT Analysis

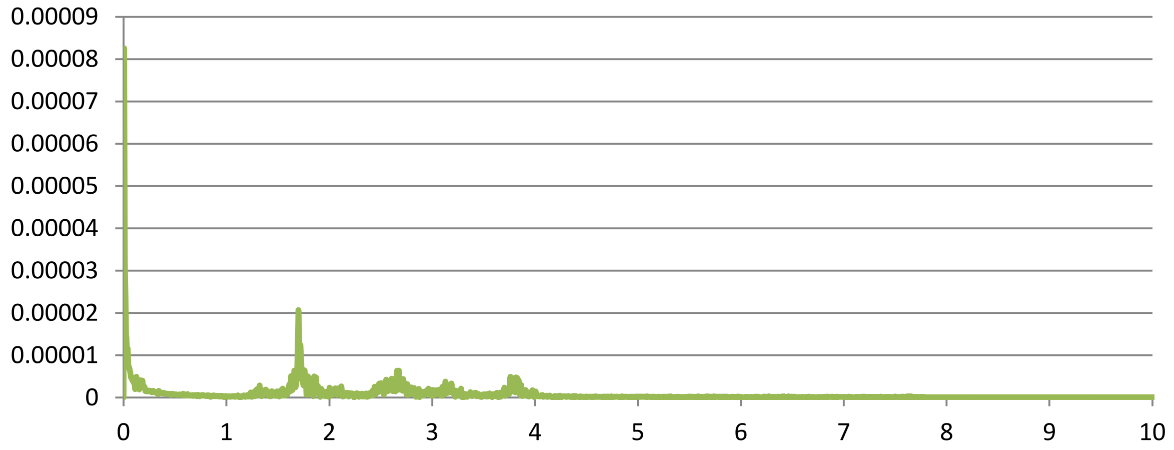
Upper Junction



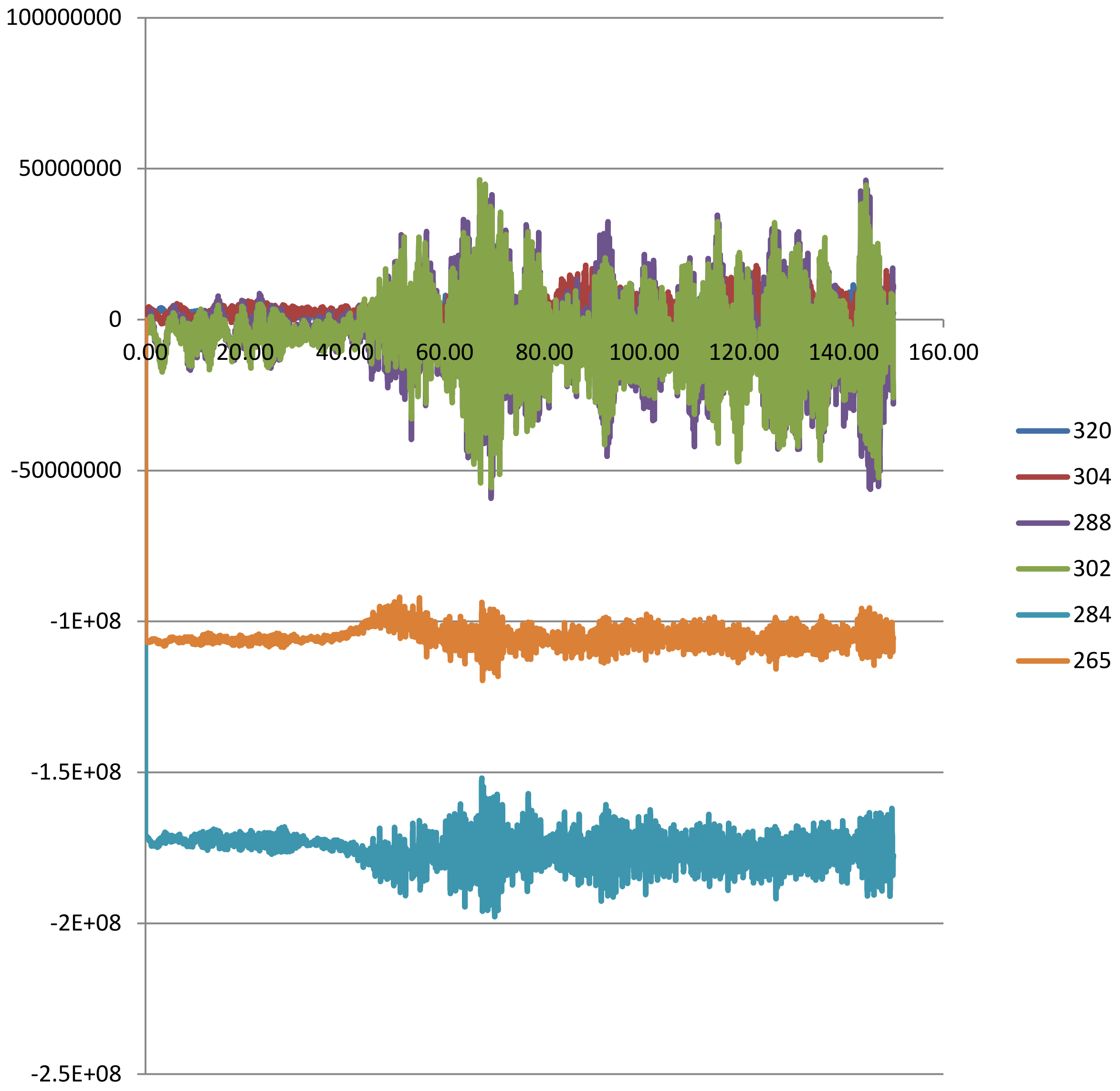
Mid Junction



Lwr Junction



11.21.5 Beam stresses



Max Beam Stresses

320	304	302	288	284	265	
1.83E+07	2.53E+07	4.63E+07	4.62E+07	2.84E+02	2.65E+02	*[Pa]*
18.258	25.273	46.278	46.183	0.000	0.000	*[MPa]*

Min Beam Stresses

320	304	302	288	284	265	
-1.08E+07	-1.29E+07	-5.57E+07	-5.92E+07	-1.98E+08	-1.20E+08	*[Pa]*
-10.831	-12.857	-55.719	-59.242	-197.767	-119.581	*[MPa]*

Delta sigma

320	304	302	288	284	265	
29.089	38.130	101.997	105.425	197.768	119.581	*[MPa]*

11.22 Fully Coupled Analysis-0.01s increment- Event 1 - Fedem

11.22.1 Description

- Fully coupled Fedem analysis
- K, C, and M matrices replacing the soil piles properties
- Turbine setup according to “Definition of a 5-MW Reference Wind Turbine for Offshore System Development”
    - J. Jonkman, S. Butterfield, W. Musial, and G. Scott
- Time increment used is 0.01 seconds (used in Sequentially Coupled Analysis)
- Event 1

11.22.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	1.522E+00
Stress range	$\Delta\sigma$ [MPa]	8.125E+00
Damage		5.541E-10

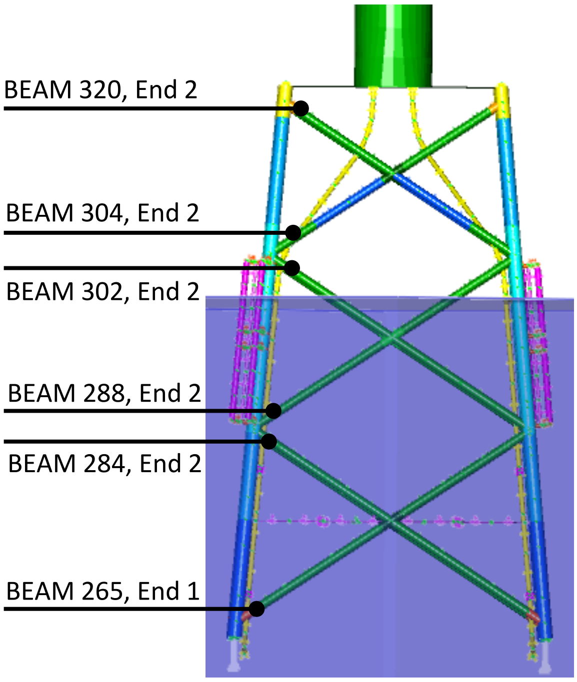
Beam 304		
Max stress	σ_{max} [MPa]	2.895E+00
Stress range	$\Delta\sigma$ [MPa]	1.310E+01
Damage		2.696E-09

Beam 302		
Max stress	σ_{max} [MPa]	1.214E+01
Stress range	$\Delta\sigma$ [MPa]	2.085E+01
Damage		3.421E-09

Beam 288		
Max stress	σ_{max} [MPa]	1.171E+01
Stress range	$\Delta\sigma$ [MPa]	1.525E+01
Damage		4.103E-09

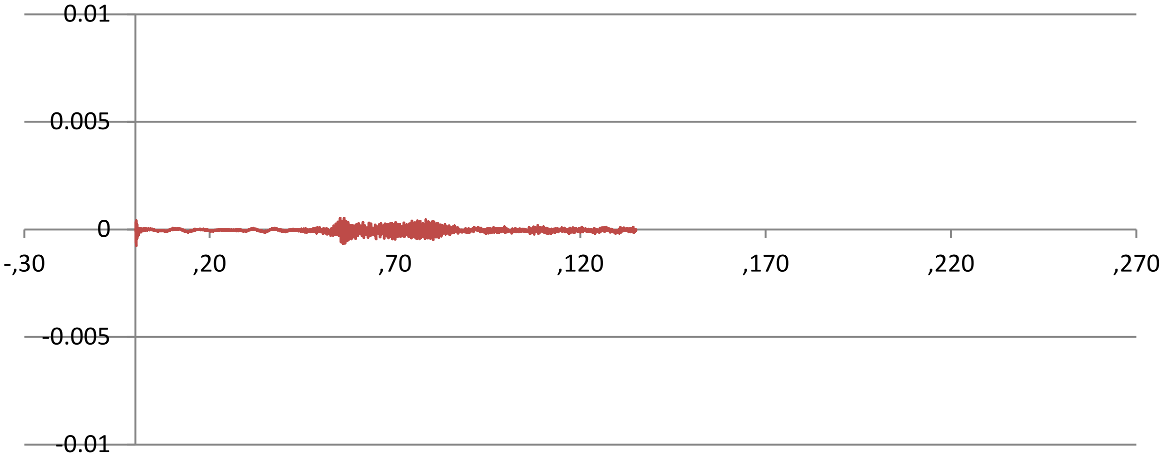
Beam 284		
Max stress	σ_{max} [MPa]	1.211E+02
Stress range	$\Delta\sigma$ [MPa]	3.295E+01
Damage		3.218E-08

Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	1.063E+02
Damage		1.256E-08

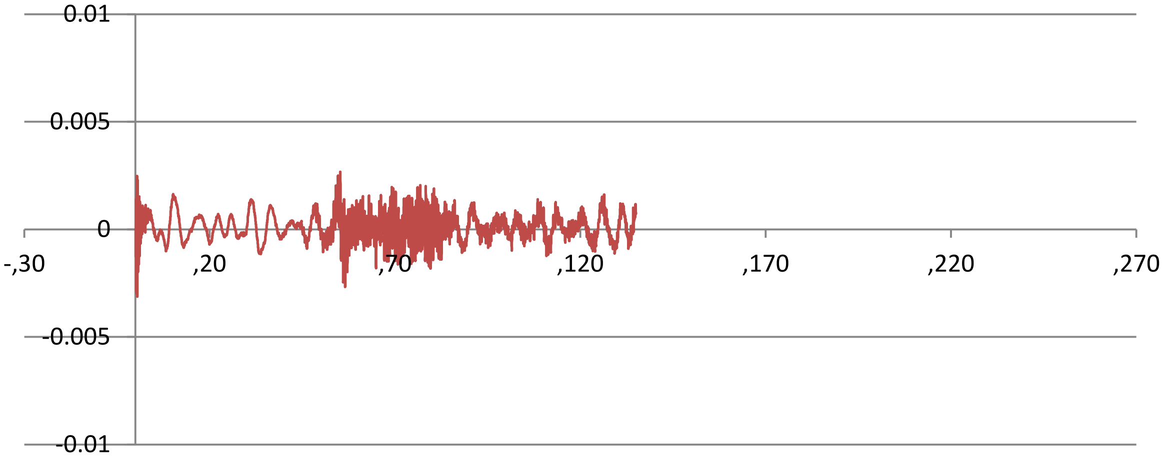


11.22.3 Bracer displacement

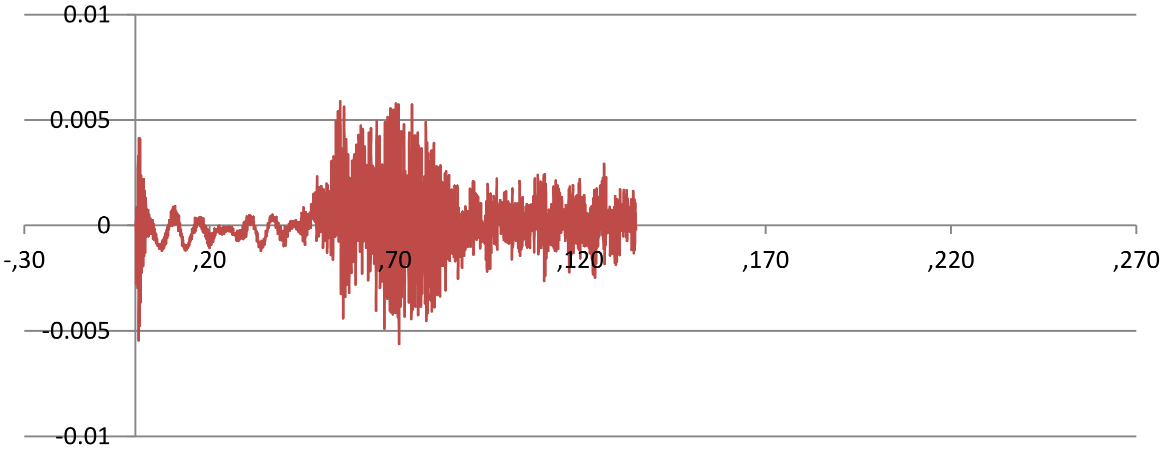
Upper Junction



Mid Junction

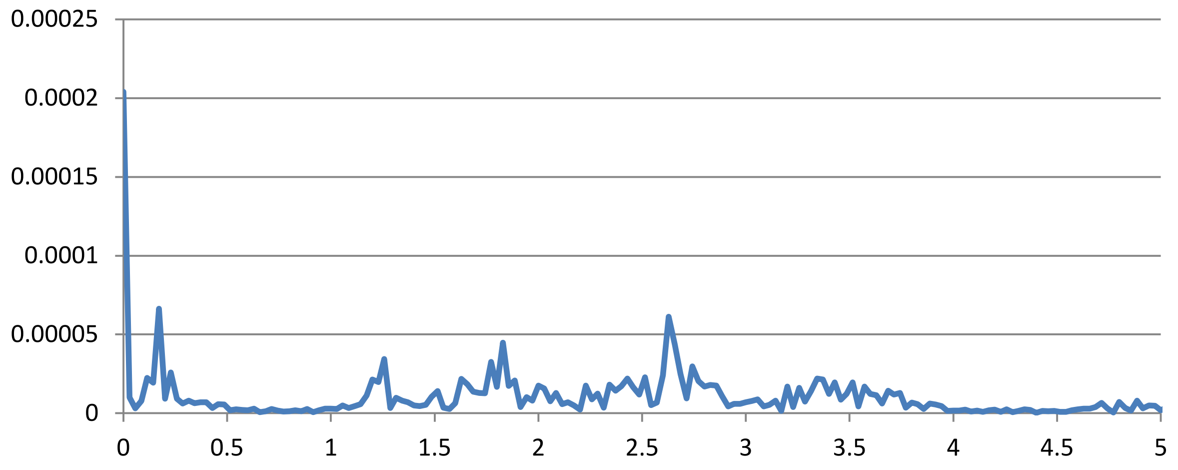


Lwr Junction

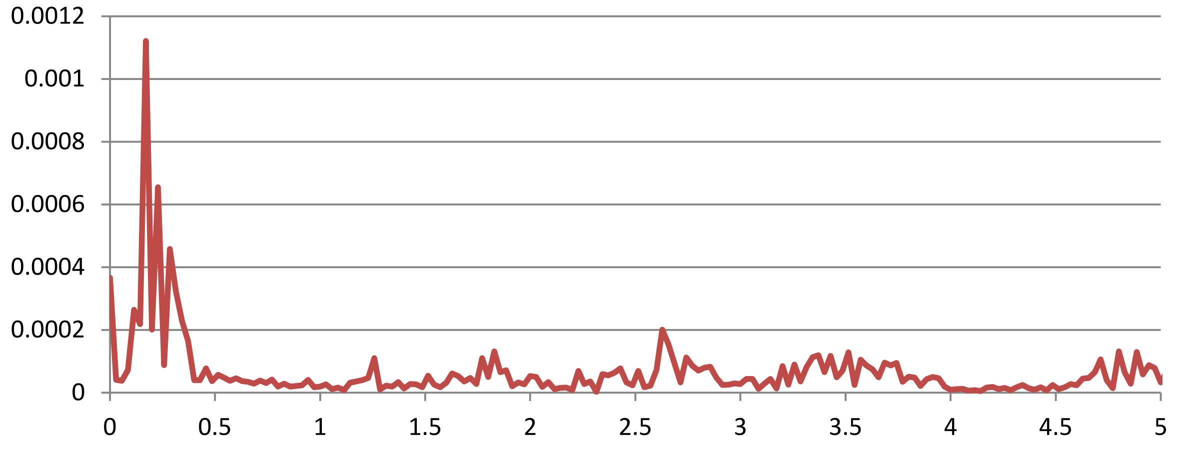


11.22.4 Bracer FFT Analysis

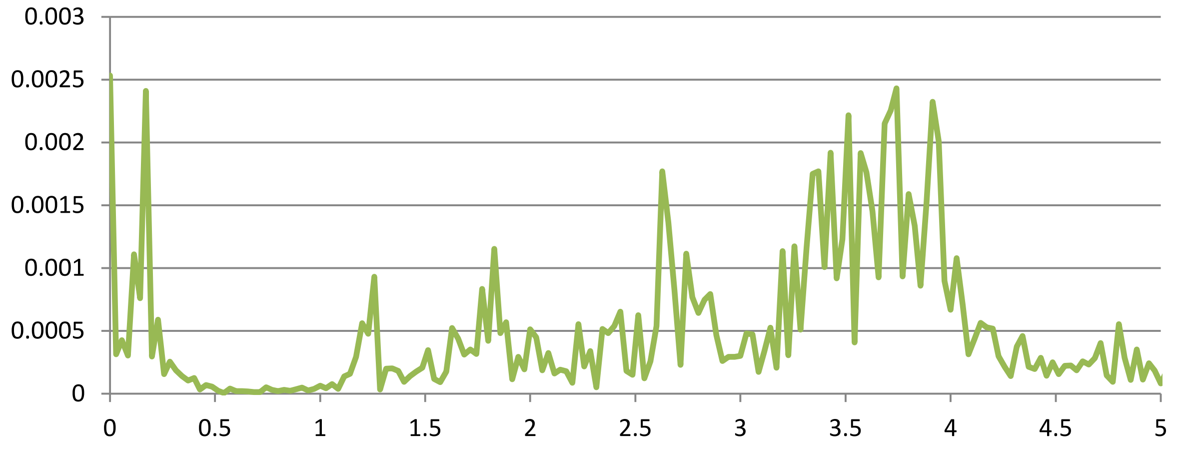
Upper Junction



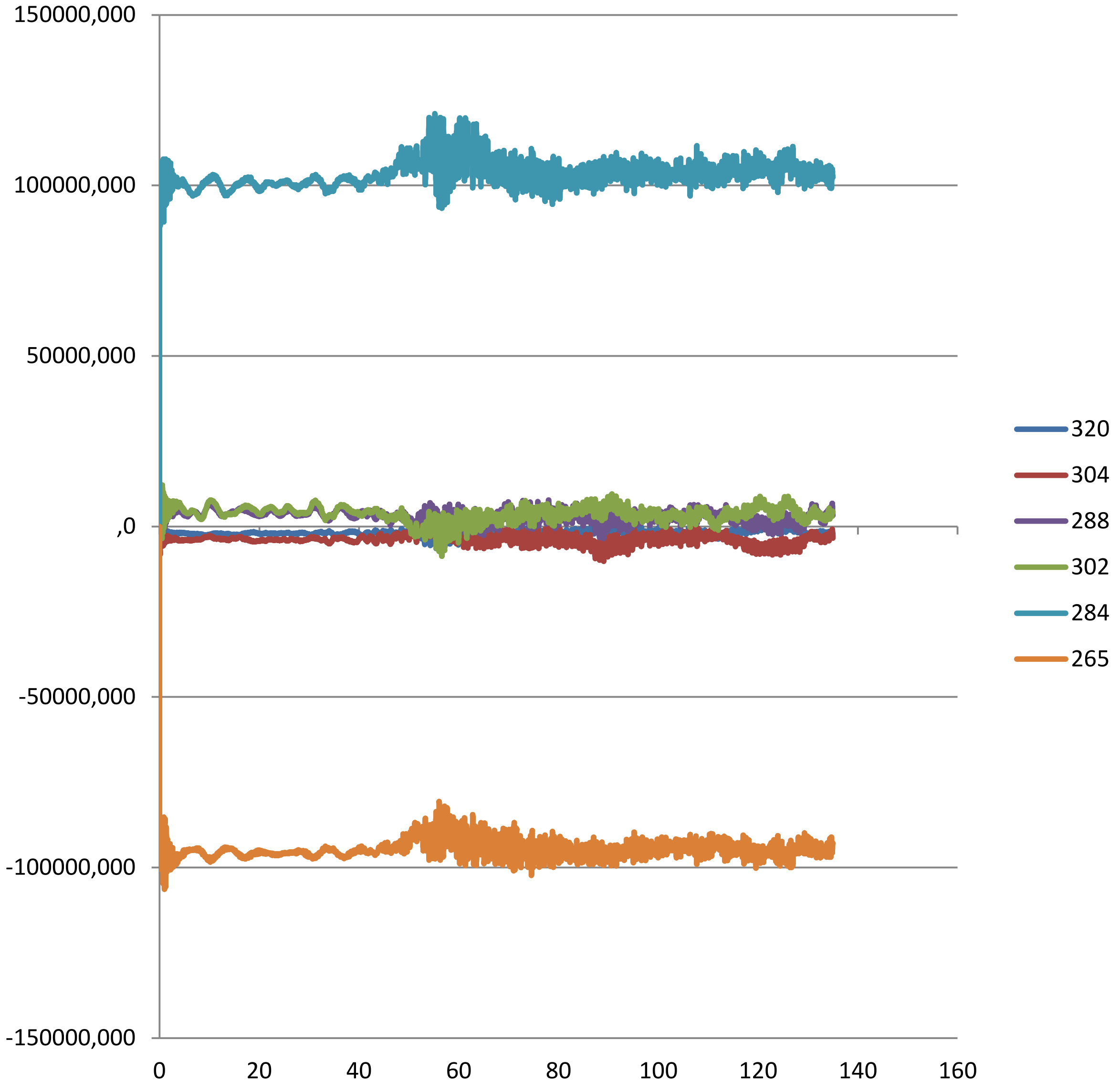
Mid Junction



Lwr Junction



11.22.5 Beam stresses



Max Beam Stresses

320	304	302	288	284	265	
1.52E+06	2.90E+06	1.21E+07	1.17E+07	1.21E+08	2.65E+02	*[Pa]*
1.522	2.895	12.136	11.714	121.092	0.000	*[MPa]*

Min Beam Stresses

320	304	302	288	284	265	
-6.60E+06	-1.02E+07	-8.71E+06	-3.53E+06	8.81E+07	-1.06E+08	*[Pa]*
-6.604	-10.202	-8.710	-3.534	88.147	-106.335	*[MPa]*

Delta sigma

320	304	302	288	284	265	
8.125	13.097	20.846	15.247	32.945	106.335	*[MPa]*

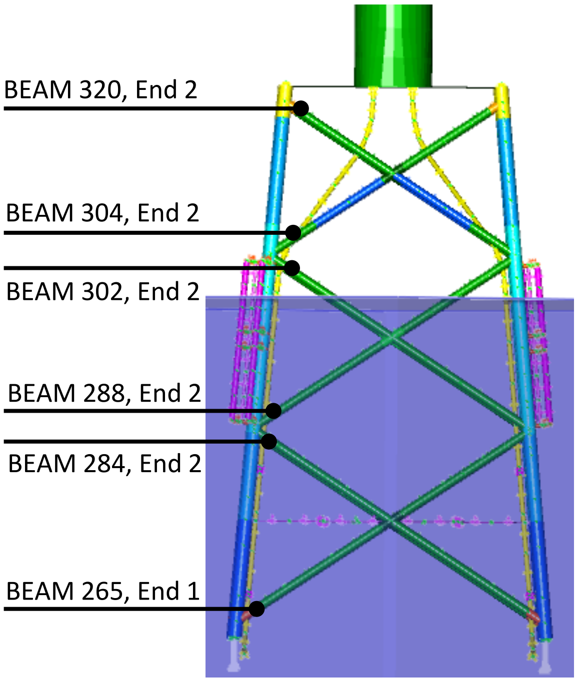
11.23 Fully Coupled Analysis-0.02s increment- Event 1 - Fedem

11.23.1 Description

- Fully coupled Fedem analysis
- K, C, and M matrices replacing the soil piles properties
- Turbine setup according to “Definition of a 5-MW Reference Wind Turbine for Offshore System Development”
    - J. Jonkman, S. Butterfield, W. Musial, and G. Scott
- Time increment used is 0.02 seconds (used in Fully Coupled Analysis in FEDEM)
- Event 1

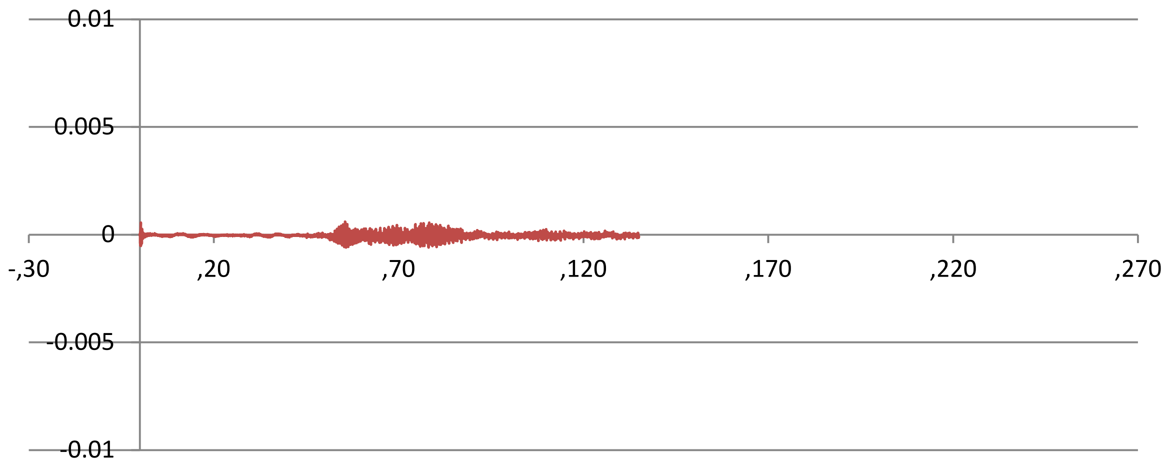
11.23.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	1.719E+00
Stress range	$\Delta\sigma$ [MPa]	7.965E+00
Damage		6.185E-10
Beam 304		
Max stress	σ_{max} [MPa]	1.930E+00
Stress range	$\Delta\sigma$ [MPa]	1.225E+01
Damage		2.816E-09
Beam 302		
Max stress	σ_{max} [MPa]	1.166E+01
Stress range	$\Delta\sigma$ [MPa]	1.925E+01
Damage		3.583E-09
Beam 288		
Max stress	σ_{max} [MPa]	1.168E+01
Stress range	$\Delta\sigma$ [MPa]	1.601E+01
Damage		5.056E-09
Beam 284		
Max stress	σ_{max} [MPa]	1.229E+02
Stress range	$\Delta\sigma$ [MPa]	1.229E+02
Damage		3.319E-08
Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	1.063E+02
Damage		1.214E-08

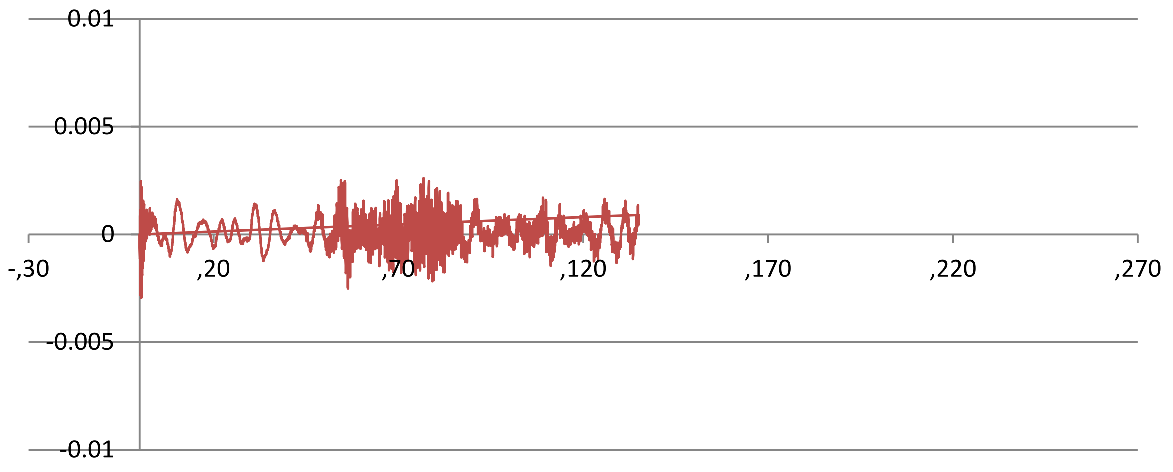


11.23.3 Bracer displacement

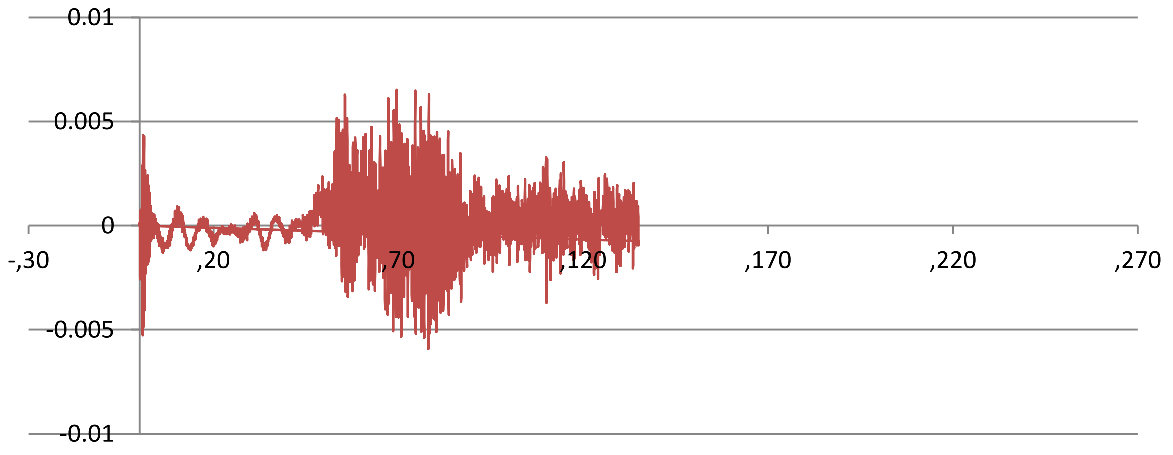
Upper Junction



Mid Junction

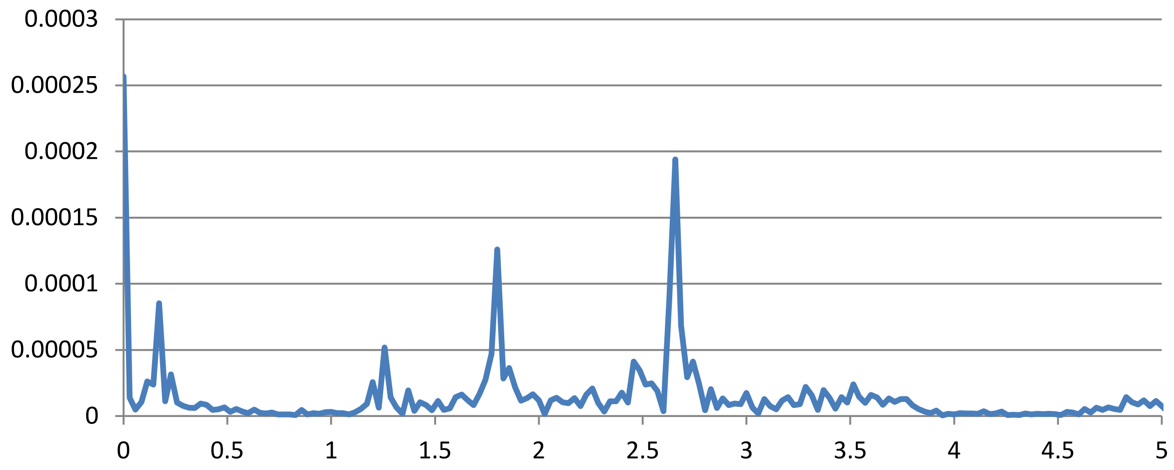


Lwr Junction

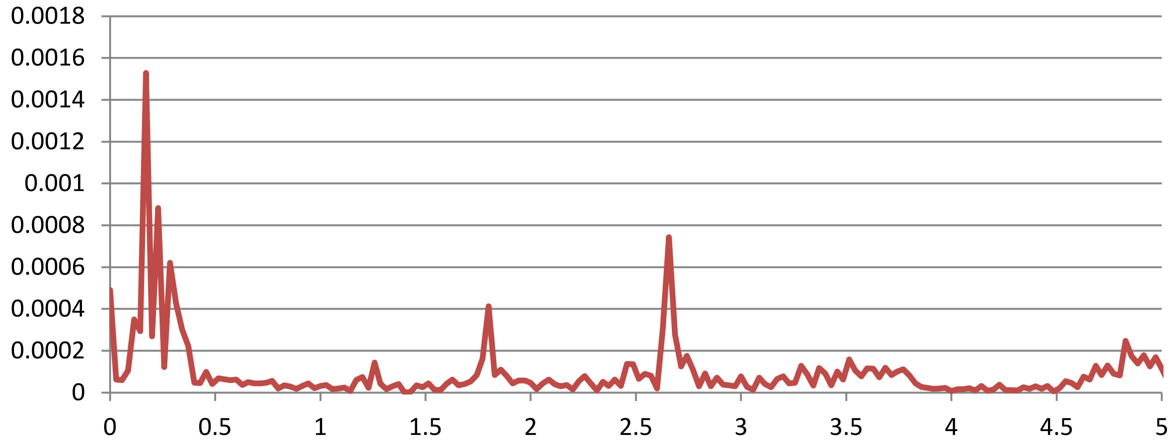


11.23.4 Bracer FFT Analysis

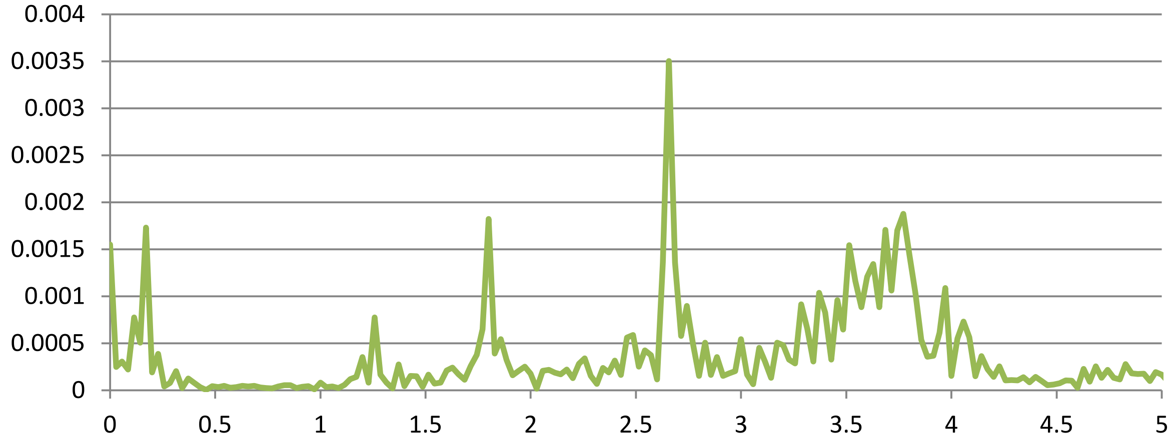
Upper Junction



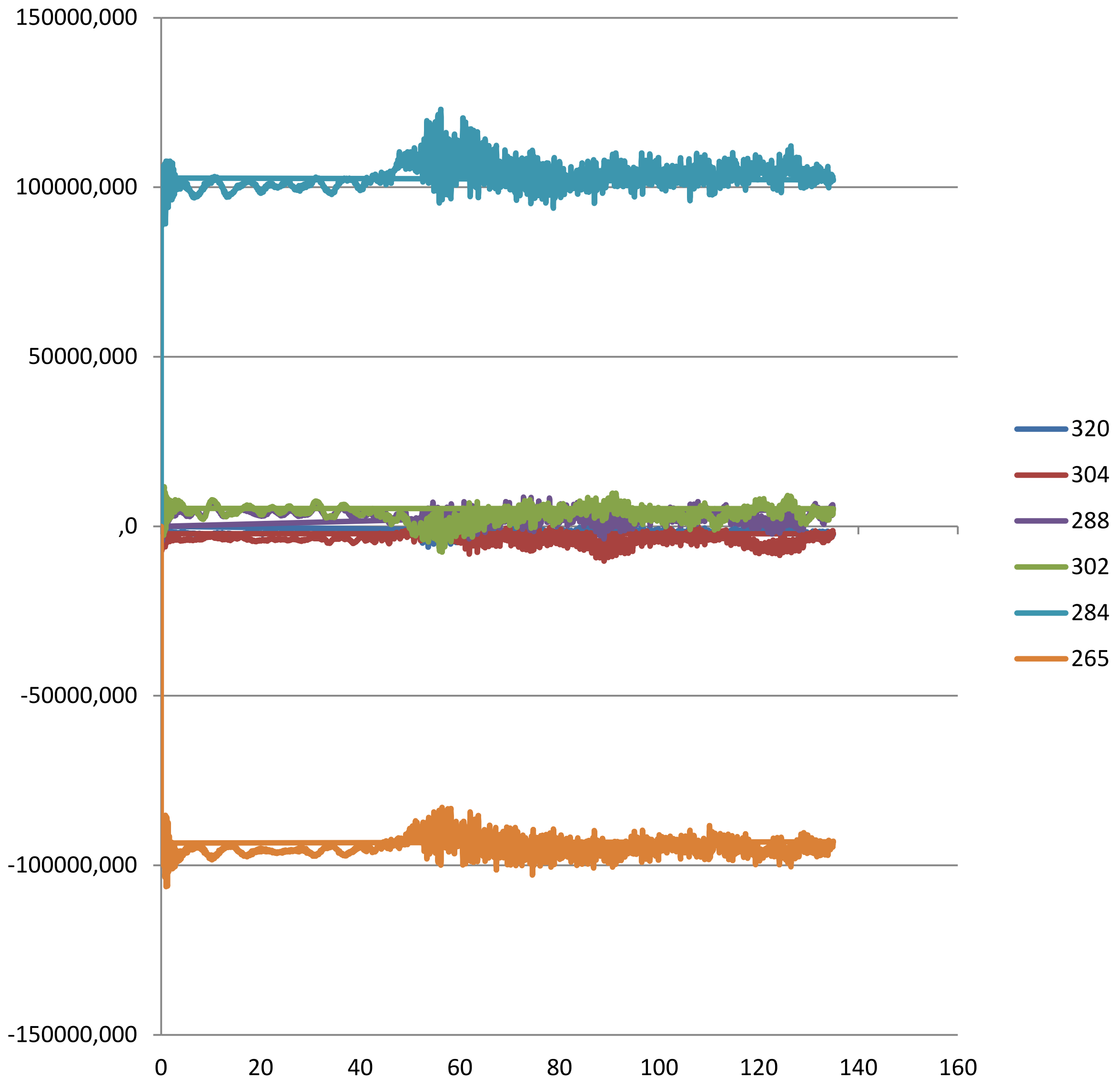
Mid Junction



Lwr Junction



11.23.5 Beam stresses



Max Beam Stresses

320	304	302	288	284	265	
1.72E+06	1.93E+06	1.17E+07	1.17E+07	1.23E+08	2.65E+02	*[Pa]*
1.719	1.930	11.661	11.681	122.932	0.000	*[MPa]*

Min Beam Stresses

320	304	302	288	284	265	
-6.25E+06	-1.03E+07	-7.59E+06	-4.33E+06	0.00E+00	-1.06E+08	*[Pa]*
-6.246	-10.321	-7.589	-4.326	0.000	-106.303	*[MPa]*

Delta sigma

320	304	302	288	284	265	
7.965	12.251	19.250	16.006	122.932	106.303	*[MPa]*

11.24 Fully Coupled Analysis-0.04s increment- Event 1 - Fedem

11.24.1 Description

- Fully coupled Fedem analysis
- K, C, and M matrices replacing the soil piles properties
- Turbine setup according to "Definition of a 5-MW Reference Wind Turbine for Offshore System Development"
    - J. Jonkman, S. Butterfield, W. Musial, and G. Scott
- Time increment used is 0.04 seconds (used in Fully Coupled Analysis by TDA)
- Event 1

11.24.2 Summary

Beam 320		
Max stress	σ_{max} [MPa]	2.121E+00
Stress range	$\Delta\sigma$ [MPa]	8.608E+00
Damage		5.197E-10

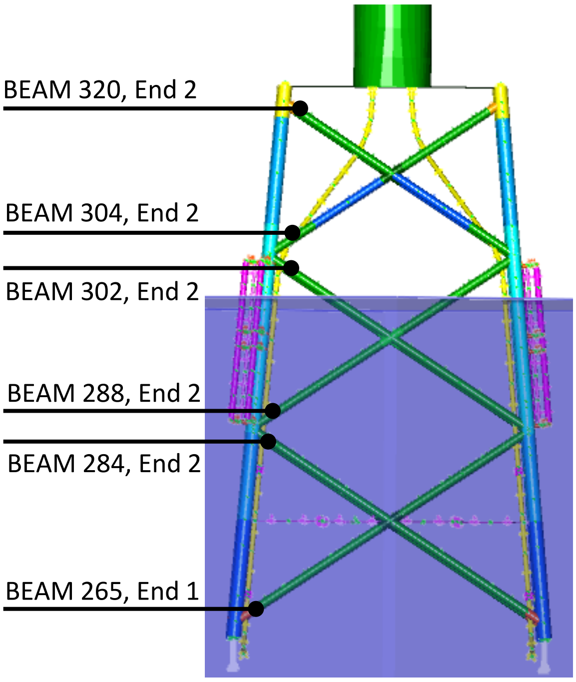
Beam 304		
Max stress	σ_{max} [MPa]	2.347E+00
Stress range	$\Delta\sigma$ [MPa]	1.267E+01
Damage		2.440E-09

Beam 302		
Max stress	σ_{max} [MPa]	1.036E+01
Stress range	$\Delta\sigma$ [MPa]	1.725E+01
Damage		3.043E-09

Beam 288		
Max stress	σ_{max} [MPa]	1.008E+01
Stress range	$\Delta\sigma$ [MPa]	1.548E+01
Damage		4.359E-09

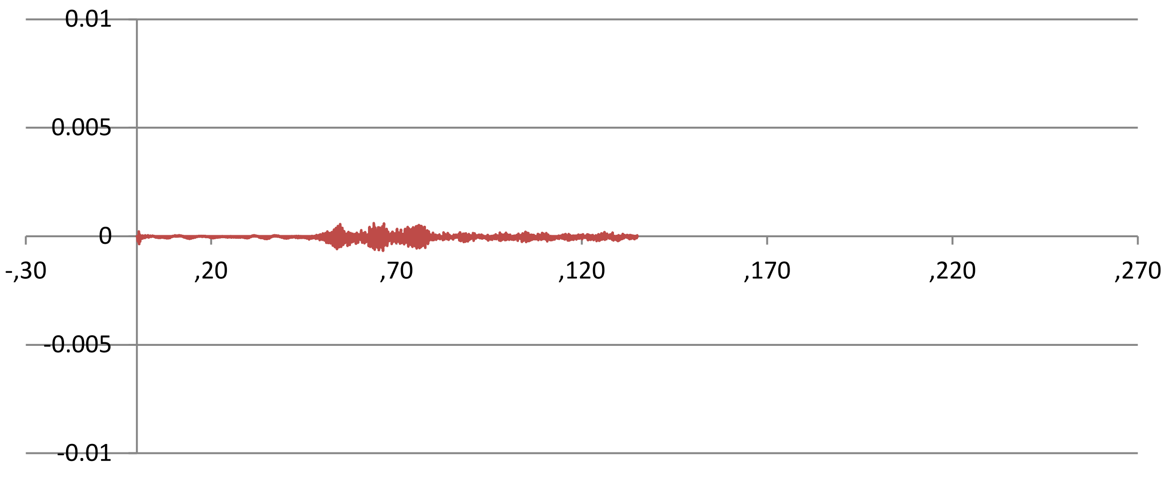
Beam 284		
Max stress	σ_{max} [MPa]	1.225E+02
Stress range	$\Delta\sigma$ [MPa]	1.225E+02
Damage		3.275E-08

Beam 265		
Max stress	σ_{max} [MPa]	2.650E-04
Stress range	$\Delta\sigma$ [MPa]	1.050E+02
Damage		1.441E-08

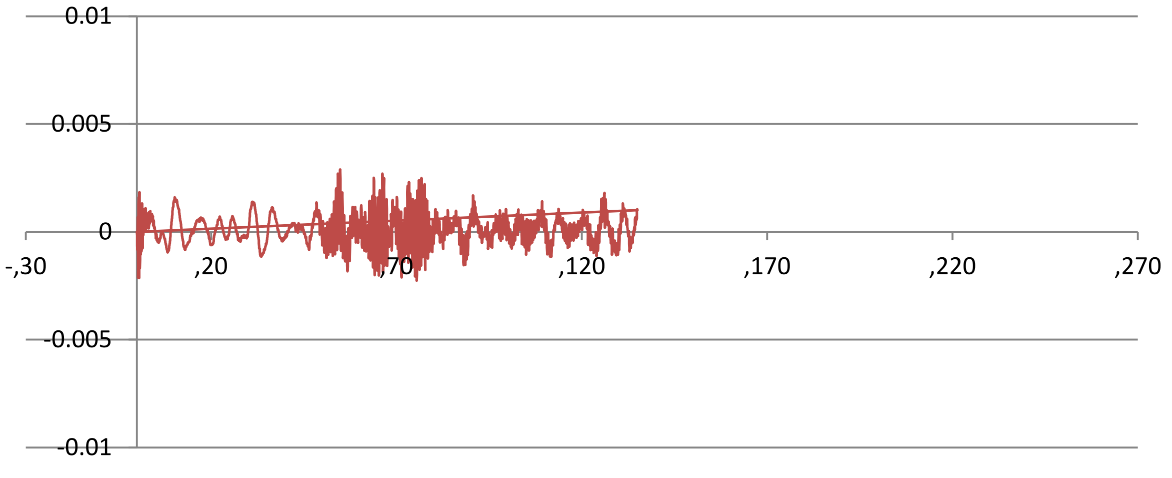


11.24.3 Bracer displacement

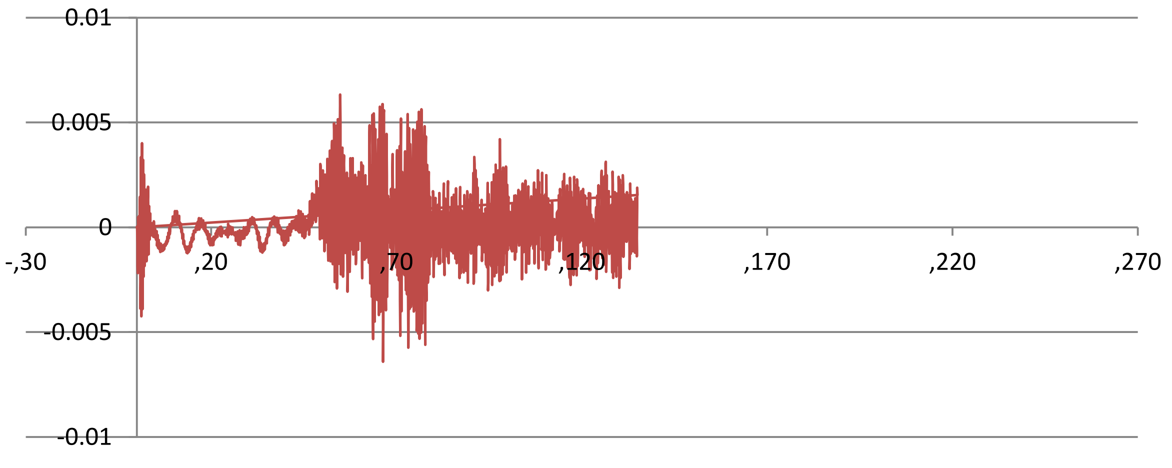
Upper Junction



Mid Junction

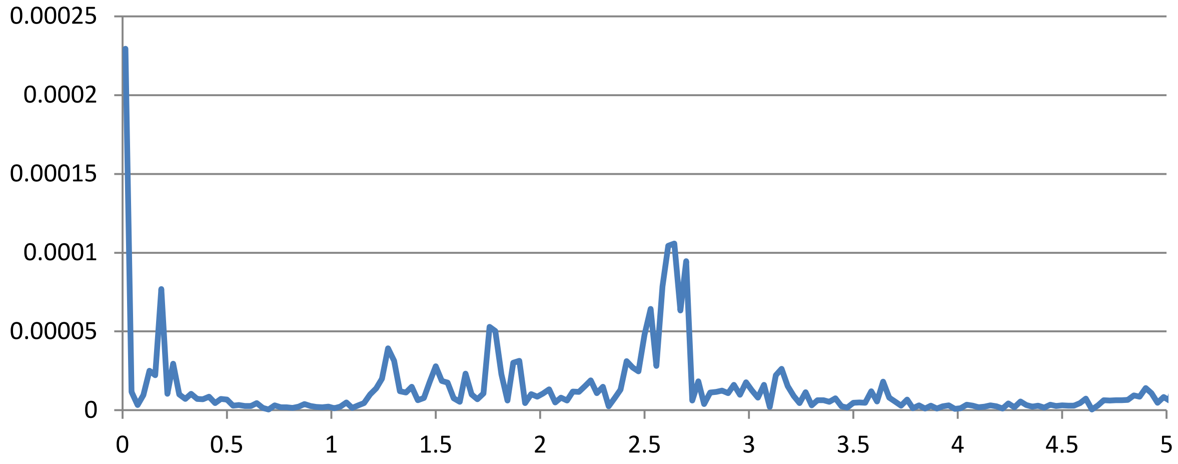


Lwr Junction

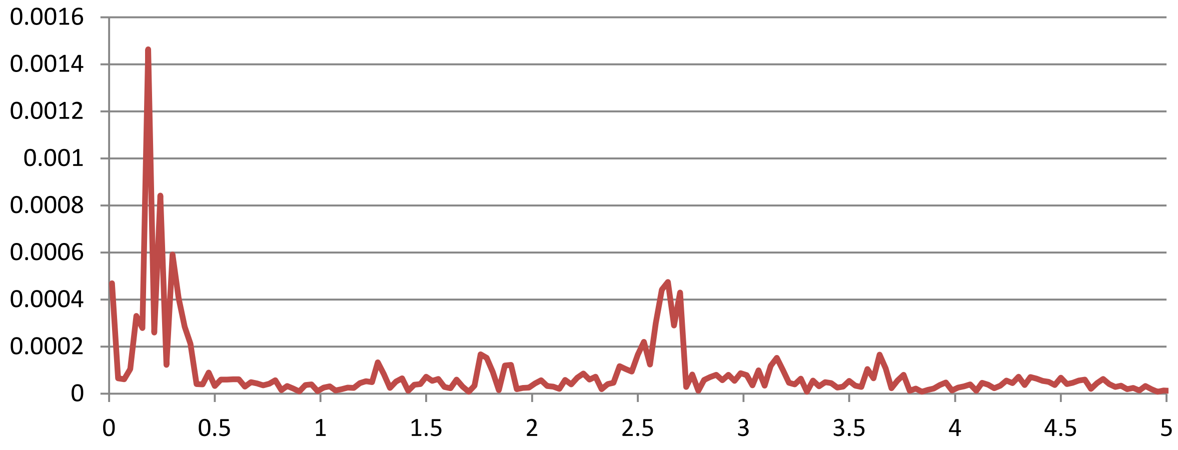


11.24.4 Bracer FFT Analysis

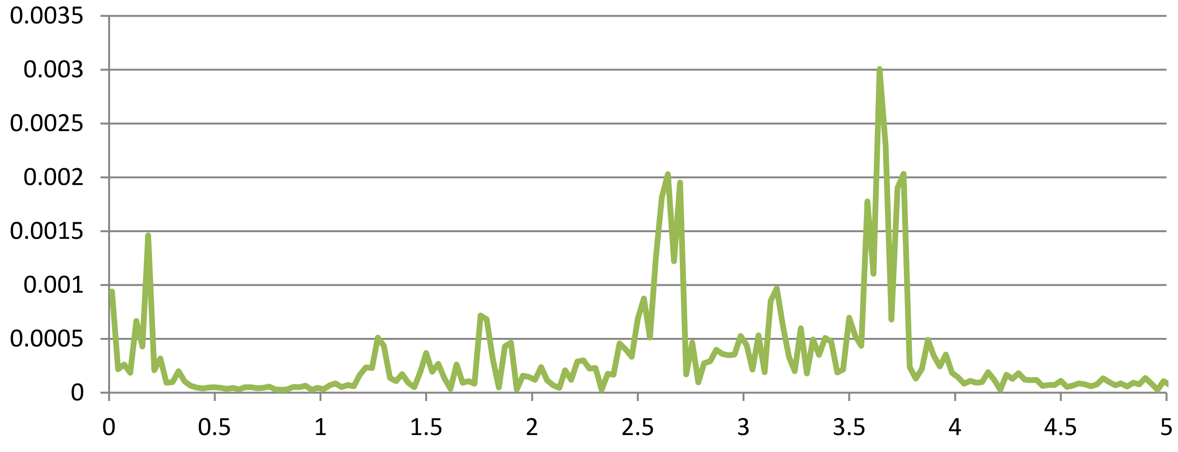
Upper Junction



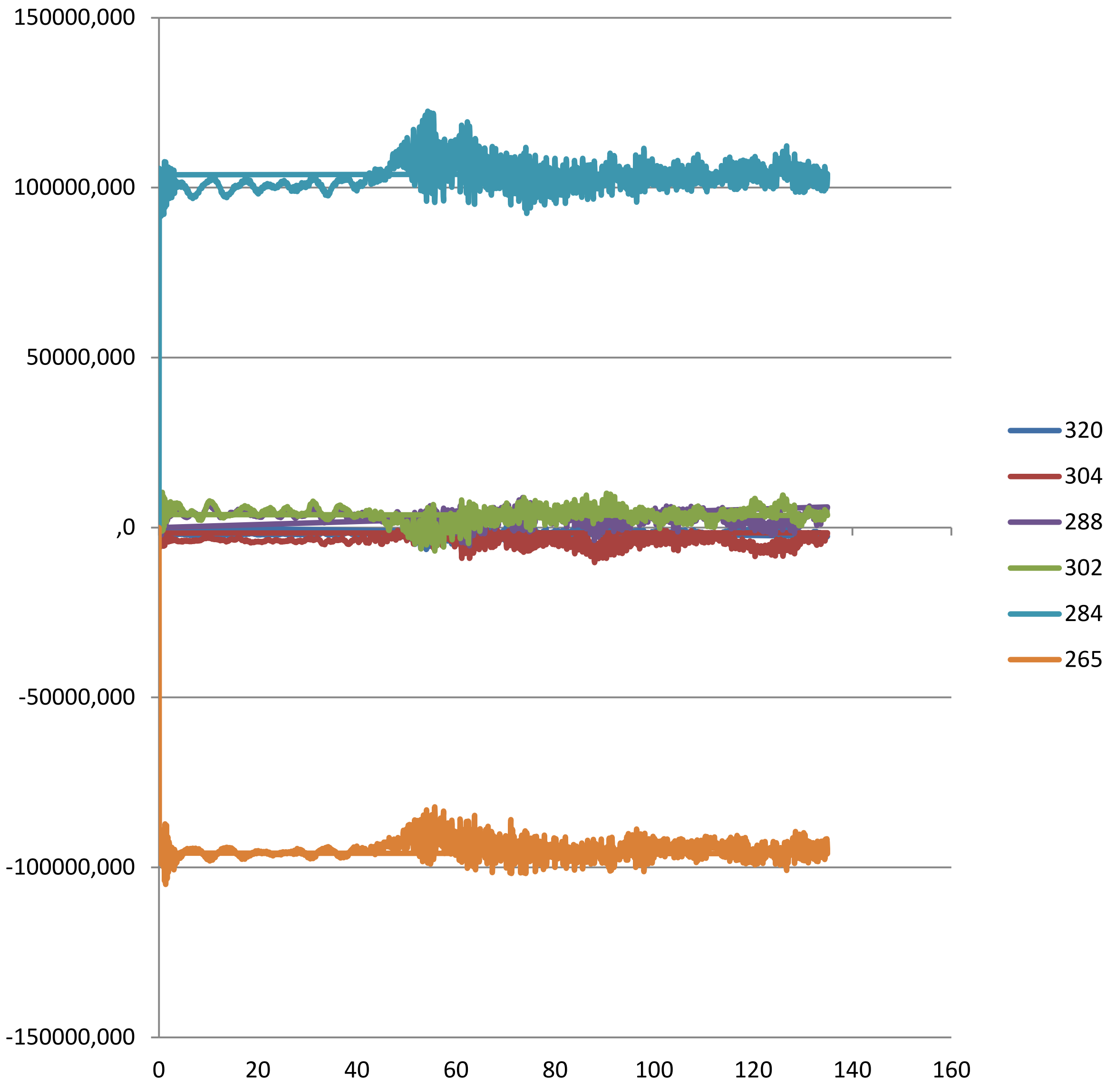
Mid Junction



Lwr Junction



11.24.5 Beam stresses



Max Beam Stresses

320	**304**	**302**	**288**	**284**	**265**	
2.12E+06	2.35E+06	1.04E+07	1.01E+07	1.22E+08	2.65E+02	*[Pa]*
2.121	2.347	10.356	10.078	122.497	0.000	*[MPa]*

Min Beam Stresses

320	**304**	**302**	**288**	**284**	**265**	
-6.49E+06	-1.03E+07	-6.90E+06	-5.40E+06	0.00E+00	-1.05E+08	*[Pa]*
-6.487	-10.318	-6.898	-5.405	0.000	-105.025	*[MPa]*

Delta sigma

320	**304**	**302**	**288**	**284**	**265**	
8.608	12.665	17.254	15.483	122.497	105.026	*[MPa]*

11.25 Study on numerical noise

11.25.1 Vibrations caused by the analysis algorithm

The phenomenon: When doing tests in USFOS it was observed that certain conditions lead to vibrations that would not decay over time. Common for all analysis showing this phenomenon were:

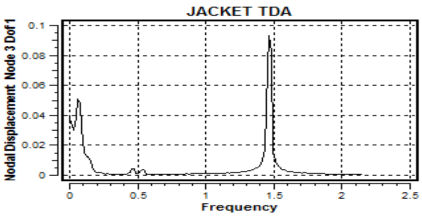
- Relative velocity (REL_VELO) between structure and water particles was activated.
- Regular wave defined (excitation)
- Current defined

FEA setup: Beam element model. One end clamped and one end free. Submerged 5 meters below surface. Subjected to one regular wave and current. The beam dimensions were randomly chosen.

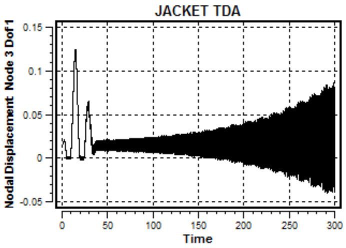
Shown results:

- FFT of vibrations in wave direction
- Vibration amplitude versus Time in wave direction

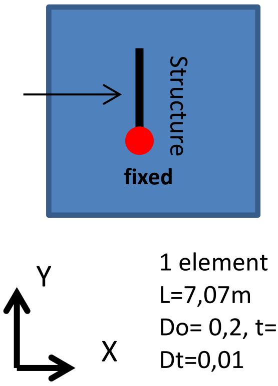
11.25.2 Simple structure – One element – Escalating vibrations over time, dt=0.01



U1: Mod90 head13d

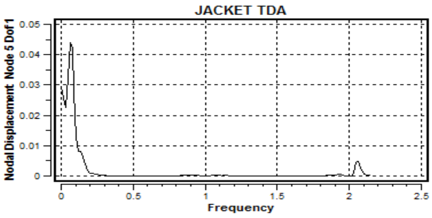


Top View

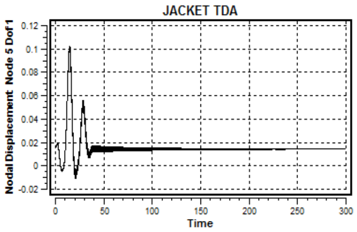


1 element
L=7,07m
Do= 0,2, t=0,03
Dt=0,01

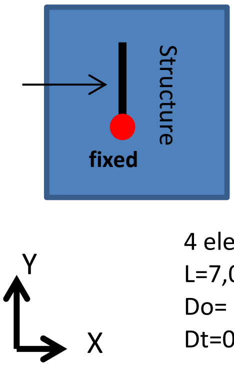
11.25.3 Simple structure –four elements – decaying vibrations over time, dt=0.01



U1:



Top View



4 element
L=7,07m
Do= 0,2, t=0,03
Dt=0,01

11.25.4 Observations:

- Relative velocity will under special circumstances cause escalating high frequency vibrations.
- Phenomenon observed to be present when model consisted of one beam element.
- Phenomenon observed to disappear when model consisted of four beam elements.
- Increasing the speed of the current will increase vibration amplitudes.

The cause to the phenomenon is not known.

11.25.5 USFOS Model file and Head file for setup to “U1: MOD90_head13d

```
‘HEAD FILE: head13d
HEAD                  JACKET TDA
              U S F O S  progressive collapse analysis
                    Usfos AS
XfosFull
'       Interval   Interval  Interval
'       Restart    Result    .out-file
 CSAVE       0       -3        -10
'       nloads   npostp   mxpstp   mxpdis
 CUSFOS     10       50      1.00     2.5
'       lcomb    lfact    mxld     nstep     minstp
          1       1.0      5.0       0        0.05
          1       0.5      6.0       0        0.01
          1       0.1      8.0      80        0.001
CNODES     1
'       nodex    idof     dfact
         3        1        1.
'RAYLDAMP           alpha1       alpha2
 RAYLDAMP           .01           .0001
'REL_VELO           nAvrg
 REL_VELO            0
'         End_Time   Delta_T   Dt_Res  Dt_Pri
 Static       1.0       1.0        1.0    1.0
 Dynamic     300.0       0.01       0.5    0.5
'       ID   <type>  Dtime Factor   Start_time
 TIMEHIST  2   Switch  1.0     1.0       0.0
'       ID  <type>
 TIMEHIST  1   Points
'      Time           Factr
       0.0            0.0
       1.0            1.0
      10.0             1.0
'         Ildcs   Tim Hist
 LOADHIST     1       1
 LOADHIST     2       2
'RegWave     Ildcs  <type>   H    Period   Direction Phase  Surf_Lev Depth
 WAVEDATA      2    Stoke  20.0    16.0       0       0.0     0     100
#                          Number of Ini_Points      X   Fac
                                  7           -10000  0.0
                                              -1000  0.0
                                               -600  1.0
                                               -400  1.0
                                               -200  1.0
                                                -20  0.0
                                               1000  0.0
'         Ildcs  Speed Direction Surf_Lev Depth    [Profile]
 CURRENT       2    2      0        0    100         0.0  1.0
                                            -20.0  1.0
                                           -100.0  0.0
                                           -110.0  0.0
'             Z    Add_Thick
 M_GROWTH       0.0     0.1
           -20.0     0.1
           -50.0     0.03
          -100.0     0.0
          -110.0     0.0
 BUOYANCY
```

```
‘MODEL FILE: MOD90
NODE      3            0            0            -5
NODE      7            0            -7.07        -5           1            1            1            1            1            1
BEAM      6            3            7                         2            4            6
UNITVEC5               1            0            0
UNITVEC6               1            0            0
PIPE      4            0.2          0.03
MISOIEP  2             2.1E+11      0.3          3.24E+08  7850         0
GRAVITY                1            0            0            -9.81
DYNRES_N               Disp         3            1
DYNRES_N               Disp         3            2
DYNRES_N               Disp         3            3
```

11.26 RUN_Script

These are the scripts written for running and handling files for the Usfos software analyses.

11.26.1 _00_00_CleanBat

Line 1 deletes all usfos analysis batch redir text files, line 2 deletes the files for DynRes, line 3 for Postfos, and 4 deletes all batch files.

Code 1 - _00_00_CleanBat

```
1   DEL *UAR.txt
2   DEL *DRAR.txt
3   DEL *PAR.txt
4   DEL     *bat
```

11.26.2 _01_01_RunUsfos

Running of Usfos cmd.

Code 2 - _01_01_RunUsfos

```
cd /d %~dp0
dir
C:\USFOS\bin\USFOS.EXE
pause
```

11.26.3 _01_02_RunDynres

Running of DynRes cmd.

Code 3 - _01_02_RunDynres

```
cd /d %~dp0
dir
C:\USFOS\bin\dynRes.EXE
pause
```

11.26.4 _01_03_RunPostfos

Running of Postfos cmd.

Code 4 - _01_03_RunPostfos

```
cd /d %~dp0
dir
C:\USFOS\bin\Postfos.EXE
pause
```

11.26.5 _01_04_RunXact

Running of Xact GUI.

Code 5 - _01_04_RunXact

```
cd /d %~dp0
C:\USFOS\bin\xact.EXE
```

11.26.6 _02_00_RunAllAnalyses

This script calls all “run” analysis scripts one by one.

Code 6 - _02_00_RunAllAnalyses

```
::
:   Run all analyses
:
:   RunUsfosAnalysis
:   Command BatFile
    cd /d %~dp0
    Call    _02_01_RunUsfosAnalysis
:
:   RunDynResAnalysis
:   Command BatFile
    cd /d %~dp0
    Call    _02_02_RunDynResAnalysis
:
:   RunPostfosAnalysis
:   Command BatFile
    cd /d %~dp0
    Call    _02_03_RunPostfosAnalysis
:
:
```

11.26.7 _02_01_RunUsfosAnalysis

This script run all Usfos analyses in the folder. The analyse package has to have "*UA.bat" (UsfosAnalysis.bat) as extension. Line 5 makes working directory as the script folder directory. Line 6 lists all files in the folder with the specific extension, and executes the batch files listed in sequence.

Code 7 - _02_01_RunUsfosAnalysis

```
1   :
2   :   Run all Usfos analyses in folder
3   :
4   :   Command FileName    Execute command
5       cd /d %~dp0
6       FORFILES /M *UA.bat /C "cmd /c call @file"
7   :
8   :
```

11.26.8 _02_02_RunDynresAnalyses

This script run all DynRes analyses in the folder. The analyse package has to have "*DRA.bat" (DynResAnalysis.bat) as extension. Line 5 makes working directory as the script folder directory. Line 6 lists all files in the folder with the specific extension, and executes the batch files listed in sequence.

Code 8 - _02_02_RunDynresAnalyses

```
01  :
02  :   Run all DynRes analyses in folder
03  :
04  :   Command FileName    Execute command
05  :
06      cd /d %~dp0
07      FORFILES /M *DRA.bat    /C "cmd /c call @file"
08  :
09  :
```

11.26.9 _02_03_RunPostfosAnalyses

This script run all Postfos analyses in the folder. The analyse package has to have “*PA.bat” (PostfosAnalysis.bat) as extension. Line 5 makes working directory as the script folder directory. Line 6 lists all files in the folder with the specific extension, and executes the batch files listed in sequence.

Code 9 - _02_03_RunPostfosAnalyses

```
1    :
2    :   Run all Postfos analyses in folder
3    :
4    :   Command FileName    Execute command
5        cd /d %~dp0
6        FORFILES /M *PA.bat /C "cmd /c call @file"
7    :
8    :
```

11.26.10 _03_00_CleanAllResults

This script calls all “clean” analysis scripts one by one.

Code 10 - _03_00_CleanAllResults

```
:
:   Clean all analyses results
:
:   CleanUsfosResults
:   Command BatFile
    cd /d %~dp0
    Call    _03_01_CleanUsfosResults
:
:   CleanDynResResults
:   Command BatFile
    cd /d %~dp0
    Call    _03_02_CleanDynResResults
:
:   CleanPostfosResults
:   Command BatFile
    cd /d %~dp0
    Call    _03_03_CleanPosfosResults
:
:
```

11.26.11 _03_01_CleanUsfosResults

This script cleans all analysis and result files related to the Usfos analysis.

Code 11 - _03_01_CleanUsfosResults

```
:
:   Clean Usfos Results
:
cd /d %~dp0
DEL     *out
DEL     *raf
DEL     *jnt
DEL     .xact_errormsg
DEL     fort.17
DEL     *status.text
DEL     fort.94
:
:
```

11.26.12 _03_01_CleanDynresResults

This script cleans all result files related to the Dynres analysis.

Code 12 - _03_01_CleanDynresResults

```
:
:   Clean DynRes Results
:
cd /d %~dp0
DEL     *.dyn
DEL     *.fil
:
:
```

11.26.13 _03_01_CleanPostfosResults

This script cleans all result files related to the Postfos analysis.

Code 13 - _03_01_CleanPostfosResults

```
:
:   Clean Postfos Results
:
cd /d %~dp0
DEL     postfos.log
:
:
```

11.27 FFT Tool in Matlab – Code description

FFT (Fast Fourier Transformation) is a tool used for signal analyzing. It is used to calculate in which frequency range the signal amplitudes belong. To process signals the numerical computing environment Matlab R2012a is used.

The FFT tool code written for Mathlab is given by “###Code 14###” to “###Code 17###”. The code is written to automatically process and preform a FFT on every file in the folder with a specific file extension.

The “###Code 14 - FFT in Matlab –Part1/4 ###” the input data is given. Line 012-013 contain the input signal extension. As default “.IFFT” (Input Fast Fourier Transformation) is coosen. The input signals is the signals to be processed by this code. Line 015-016 contain the output signal extension. As default “.OFFT” (Output Fast Fourier Transformation) is coosen. The output signals is the FFT results for the input signals. Line 018-019 is the input signal sampling time. Line 022-023 specifies the samples time range where the average amplitude are extracted. This has to be set manually before the code is executed.

Code 14 - FFT in Matlab –Part1/4

```
001 % FFT Tool in Matlab
002 % 14 May 2013
003 % Version 1.0
004
005 clear all
006 close all
007
008  %%%%%%%%%%%%%%%%%%%%%%%%%%%%
009 %%%%        INPUT         %%%%
010  %%%%%%%%%%%%%%%%%%%%%%%%%%%%
011
012 %Insert name of input signal file extension
013 Input_signal_extension='.IFFT';
014
015 %Insert name of output signal file extension
016 Output_signal_extension='.OFFT';
017
018 % Samples time
019 % Tmax=270;
020 Ts=0.02;
021
022 % Samples time average range
023 STAR=100;
```

The “###Code 15 - FFT in Matlab –Part 2/4###” the output data is given. Line 029-039 contain the input file processing. Its function is to list all files with extension “.IFFT” and put them without extension as raw names in a matrix named “outNames”. The line 041-042 is the code loop that loops through the input files. Line 044-048 writes the full input names, while line 050-053 writes the full output names.

Code 15 - FFT in Matlab –Part2/4

```
025  %%%%%%%%%%%%%%%%%%%%%%%%%%%%%
026 %%%%          OUTPUT          %%%%
027  %%%%%%%%%%%%%%%%%%%%%%%%%%%%%
028 
029 % Listning files to run
030 dirList = dir('*.IFFT');
031 names = {dirList.name};
032 outNames = {};
033     for i=1:numel(names)
034         name = names{i};
035         if ~isequal(name,'.') && ~isequal(name,'..')
036             [~,name] = fileparts(names{i});
037             outNames{end+1} = name;
038         end
039     end
040 
041 % Running Rainflow Counting
042 for c=1:numel(names)
043 
044     % Input signal name:
045 inp1=outNames{c};
046 inp2=Input_signal_extension;
047 inp=[inp1 inp2];
048 Input_signal=load(inp);
049 
050 % Output signal name:
051 outp1=outNames{c};
052 outp2=Output_signal_extension;
053 outp=[outp1 outp2];
```

The “###Code 16 - FFT in Matlab –Part 3/4###” the FFT is performed for the input signal. The lines 062-074 the max and min amplitude for the input signal is found. The code checks the maximum amplitude at a range of values, and then compares the value with the last range block. The largest is retained for later in the code. This is later used to scale the signal. The lines 076-089 the different FFT paramerers for the signal is calculated. Line 092 the FFT is executed as imaginary numbers. Line 093 calculates the unscaled magnitude of the FFT calculation. Line 094 shifts the FFT curve to f=0. In this way the full FFT is presented, where the negative frequency range is a mirror of the positive. Line 096-098 scales the FFT signal to 1, while line 100-101 scale it to represent the input signal amplitude.

Code 16 - FFT in Matlab –Part3/4

```
055  %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
056 %%%% Perform FFT on Input_signal %%%%
057  %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
058
059 AmpValueOld=0;
060
061
062 for k=1:(length(Input_signal)-STAR)
063 %Finding limits of input signal
064
065     MaxValue=max(Input_signal(k:(k+STAR-1)));
066     MinValue=min(Input_signal(k:(k+STAR-1)));
067     AmpValueNew=(MaxValue-MinValue)/2;
068
069     if AmpValueNew>AmpValueOld
070         AmpValueOld=AmpValueNew;
071     end
072 end
073
074 AvrAmp_signal=AmpValueOld
075
076 %Input signal time
077 Tmax=length(Input_signal)*Ts;
078
084 %CalculatingParameters
085 fs=(1/Ts);
086 N=Tmax/Ts;
087
088 %Frequency vector
089 f=[-fs/2:fs/(N-1):fs/2]';
090
091 %FFT on Input_signal
092 X=fft(Input_signal);
093 X=abs(X);
094 X=fftshift(X);
095
096 %Scaling FFT to 1
097 MaxAmp_FFT=max(X);
098 X=X/MaxAmp_FFT;
099
100 %Scaling FFT to Amplitude
101 X=X*abs(AvrAmp_signal);
```

The “###Code 17 - FFT in Matlab –Part 4/4###” the output signal table is printet to file. Line 112-114 assembles the FFT matrix. Line 116-118 creates and writes the FFT martrix to the output file with the same raw input file name with file extension “.OFFT”.

Code 17 - FFT in Matlab –Part4/4

```
106  %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
107 %%%% Printing FFT Table to File %%%%
108  %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
109
110 %Write output data to text file
111
112 A=[f];
113 B=[X];
114 C=[A,B]';
115
116     fileID = fopen(outp ,'w');
117     fprintf(fileID,'%8.5f %8.15f\n',C);
118     fclose(fileID);
119
120 end
```

11.28 Rainflow Counting Tool in Matlab – Code description

RFC (RainFlow Counting) is a tool used for signal analyzing. It is used to count the number of half and whole cycluses at a spesific amplitude range of the input signal. To process signals the numerical computing environment Matlab R2012a is used.

The RFC tool code written for Mathlab is given by “###Code 1###” to “###Code 4###”. The code is written to automatically process and preform a RFC on every file in the folder with a specific file extension.

The “###Code 18 - FFT in Matlab –Part1/5 ###” the input data is given. Line 013-014 contain the input signal extension. As default “.IRFC” (Input RainFlow Counting) is coosen. The input signals is the signals to be processed by this code. Line 016-017 contain the output signal extension. As default “.ORFC” (Output RainFlow Counting) is coosen. The output signals is the RFC results for the input signals. Line 019-020 is the output signal bin number. Line 022-026 gives how the signal is processed according to limit values of the output. Line 028-029 specify if the output bin values is given by amplitude or range. Line 031-038 specifies if the results is given by different plot options.

Code 18 - RFC in Matlab –Part1/5

```
001 % Rainflow Counting in Matlab
002 % 14 May 2013
003 % Version 1.0
004
005     clear all
006     close all
007
008
009  %%%%%%%%%%%%%%%%%%%%%
010 %%%     INPUT       %%%
011  %%%%%%%%%%%%%%%%%%%%%
012
013 %Insert name of input signal file extension
014     Input_signal_extension='.IRFC';
015
016 %Insert name of output signal file extension
017     Output_signal_extension='.ORFC';
018
019 %Insert number of bins
020     Number_of_bins=1000;
021
022 %Use of maximum cycle value or pre-set value for bin defenision
023     Use_max_value='Y';
024     Use_min_value='Y';
025     Bin_end_size=1;
026     Bin_start_size=0;
027
028 %Insert '1' for amplitude or '2' for range counting
029     Amplitude_or_range=1;
030
031 %Plot of histogram Y/N:
032     Histogram='N';
033
034 %Plot of diagram
035     Diagram='N';
036
037 %Plot of diagram with log-scale
038     LogDiagram='N';
```

The “###Code 19 - RFC in Matlab –Part 2/5###” the output data is given. Line 045-055 contain the input file processing. Its function is to list all files with extension “.IRFC” and put them without extension as raw names in a matrix named “outNames”. The line 062 is the code loop that loops through the input files. Line 063-067 writes the full input names, while line 069-072 writes the full output names. Line 074-082 specify different paramters for futher calculations. Line 068 run the sig2ext code for the input signal. This code calculates the turning points (tp) of the input signal. Line 089 runs the rainflow counting on the turning point matrix of the input signal.

Code 19 - RFC in Matlab –Part2/5

```
041  %%%%%%%%%%%%%%%%%%%%%
042 %%%     OUTPUT      %%%
043  %%%%%%%%%%%%%%%%%%%%%
044
045 % Listning files to run
046     dirList = dir('*.IRFC');
047     names = {dirList.name};
048     outNames = {};
049         for i=1:numel(names)
050             name = names{i};
051             if ~isequal(name,'.') && ~isequal(name,'..')
052                 [~,name] = fileparts(names{i});
053                 outNames{end+1} = name;
054             end
055         end
056
057
058  %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
059 %%% Running Rainflow Counting %%%
060  %%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
061
062     for c=1:numel(names)
063 % Input signal name:
064     inp1=outNames{c};
065     inp2=Input_signal_extension;
066     inp=[inp1 inp2];
067     Input_signal=load(inp);
068
069 % Output signal name:
070     outp1=outNames{c};
071     outp2=Output_signal_extension;
072     outp=[outp1 outp2];
073
074 % Read force history
075 % Muliply by two for full range data, and one for amplitude
076     INPUT = Input_signal*Amplitude_or_range;
077
078 %Number of bins
079     n_bins=Number_of_bins;
080
081 % set signal to be rainflow counted
082     s = INPUT;
083
084 % Using of sig2ext function for calculating turning points
085 % Counting procedure only works properly on turning points
086     tp=sig2ext(s);
087
088 %rfhist(bins)
089     rf=rainflow(tp);
```

The “Code 20 – RFC in Matlab – Part 3/5” calculates the bins of the rainflow counting signal. Line 096-106 specifies the maximum value, and line 108-118 specifies the minimum value. Line 120-141 specifies which of the maximum and minimum bin values to be used in the matrix.

Code 20 - RFC in Matlab –Part3/5

```
 %%%%%%%%%%%%%%%%%%%%
%%% Writing matrix %%%
 %%%%%%%%%%%%%%%%%%%%

%Max value
    rf1=rf';
        rf2=rf1(:,1);

        list=[rf2];
        greatestSoFar=list(1);
        for i=1:length(list)
            if (list(i)>greatestSoFar)
            greatestSoFar=list(i);
            end
        end

%Min value
    rf1=rf';
        rf2=rf1(:,1);

        list=[rf2];
        smallestSoFar=list(1);
        for j=1:length(list)
            if (list(j)<smallestSoFar)
            smallestSoFar=list(j);
            end
        end

%Bin matrix vector
    if Use_max_value=='Y'
        if Use_min_value=='Y'
            bins=n_bins;
        elseif Use_min_value=='N'
            bin_range=greatestSoFar-Bin_start_size;
            bins_diff=(greatestSoFar-Bin_start_size)/n_bins;
bins=(Bin_start_size+(bin_range/(2*n_bins)):bins_diff:greatestSoFar-
(bin_range/(2*n_bins)));
        end
    end

    if Use_max_value=='N'
        if Use_min_value=='Y'
            bin_range=Bin_end_size-smallestSoFar;
            bins_diff=(Bin_end_size-smallestSoFar)/n_bins;
bins=(smallestSoFar+(bin_range/(2*n_bins)):bins_diff:Bin_end_size-
(bin_range/(2*n_bins)));
        elseif Use_min_value=='N'
            bin_range=Bin_end_size-Bin_start_size;
            bins_diff=(Bin_end_size-Bin_start_size)/n_bins;
bins=(Bin_start_size+(bin_range/(2*n_bins)):bins_diff:Bin_end_size-
(bin_range/(2*n_bins)));
        end
    end
```

The “###Code 21 - RFC in Matlab –Part 4/5###” the output signal table is printet to file. Line 143-144 assembles the RFC matrix. Line 146-151 creates and writes the RFC martrix to the output file with the same raw input file name with file extension “.ORFC”.

Code 21 - RFC in Matlab - Part 4/5

```
143 %Cycles matrix
144     [no,xo] = rfhist(rf,bins);
145
146 %Write output data to text file
147     A=[xo;no];
148
149     fileID = fopen(outp ,'w');
150     fprintf(fileID,'%6.2f %12.8f\n',A);
151     fclose(fileID);
```

The “###Code 22 - RFC in Matlab –Part 5/5###” the differet plots specified in in the input section is executed.

Code 22 - RFC in Matlab –Part5/5

```
 %%%%%%%%%%%%%%%%%%%%%
%%% Result Plotting %%%
 %%%%%%%%%%%%%%%%%%%%%

%Plot of histogram
    if (Histogram=='Y')
        rfhist(rf,bins)
    else
    end

%Plot of diagram
    if (Diagram=='Y')
    figure
    plot(xo,no)
    legend('MATLAB');
```

11.29 Rainflow for MATLAB by Adam Nieslony

Version date: 2010-04-03

11.29.1 Introduction

The rainflow cycle counting algorithm is widely used while fatigue life assessment of machine components or structures under non-constant amplitude loading. Usually, the algorithm extract cycles from load, stress or strain history obtained from measurement or simulation. As a results of the counting several cycles and half-cycles with different amplitude and mean value are obtained. With the advantage of fatigue damage accumulation hypothesis, like Miners rule, the algorithm gives possibility to compute the expected fatigue life under random loading conditions. Theoretically, of course ;-)

The small toolbox RAINFLOW includes rainflow cycle counting algorithm prepared for using in the MATLAB® environment. The main function has been translated from Turbo Pascal into C language and compiled to the MEX function. The algorithm code has been written according to the ASTM standard [1] and optimized considering the calculation speed. Some other details about the usability of the function with practical application can be found in [2]. The toolbox was tested on PCs under MS Windows® XP32 and Vista64 operating systems, however I guess that they will also works properly on other OS.

In order to improve the RAINFLOW toolbox please sends all your remarks, suggestions and questions to me through the MATLAB Central contact data. All functions in the RAINFLOW toolbox where elaborated by myself. At present I am working as assistant professor on the Opole University of Technology, Department of Mechanics and Machine Design (Technical University of Opole, Department of Mechanics and Machine Design) and I am open for cooperation in the field of fatigue live assessment of machine components under multiaxial random loading.

11.29.2 Terms of use

The toolbox is copyrighted according to the MATLAB Central File Exchange terms. Additionally, I have a following, not mandatory, request to the users of my toolbox - if any scientific publication they have been made using my software, please refer to the publication [2] of my authorship, where the RAINFLOW toolbox have been first officially presented and described.

11.29.3 Installation

The RAINFLOW toolbox is a typical “one directory” MATLAB toolbox. Following files there:

`File`		Description
`rainflow.m`	-	short description of the rainflow function, used by MATLAB for build-in help engine, this is not the “working” rainflow function!
`rainflow.c`	-	C MEX code of the rainflow function
`rainflow.mexw32`	-	compiled "rainflow.c" file (MATLAB R2007a, Windows XP Prof. 32Bit)
`rainflow.mexw64`	-	compiled "rainflow.c" file (MATLAB R2007a, Windows Vista 64Bit)
`rfhist.m`	-	histograms of rainflow data
`rfmatrix.m`	-	rainflow matrix calculation and visualization
`rfdemo1.m`	-	demo for cycle counting, recommended for short signals
`rfdemo2.m`	-	demo for rainflow matrix and histograms, recommended for long signals
`sig2ext.m`	-	auxiliary function for rfdemo, it searches turning points from signals

In order to install the rainflow function, you must realise standard tasks, necessary during addition of new toolboxes to the MATLAB environment, i.e.:

1. Download the archive file rainflow.zip from File Exchange web side.
2. Create a new local directory for files of the RAINFLOW toolbox.
3. Unpack all files from the archive to that directory.
4. Add a path to that directory in MATLAB, using the command "Edit Path" from main menu.

Additionally, if the compiled rainflow MEX file (`rainflow.mexw32` for 32Bit MS Windows and `rainflow.mexw64` for 64Bit MS Windows) is out of date, or you are user of OS unlike MS Windows, you need to recompile it. Use the `rainflow.c` file with source code for this task. Please, look into the MATLAB help and search the topic “mex” for more details.

11.29.4 Quick start

11.29.4.1 Lesson 1 – using the demo files

There are two demo files:

1. rfdemo1 – show how the algorithm are working plotting the extracted cycles and half-cycles on the time history, recommended for short time signals,
2. rfdemo2 – make rainflow histograms and rainflow matrix, recommended for long time signals.

To run this demos type on the MATLAB command window following commands:

```
rfdemo1
```

or

```
rfdemo2
```

11.29.4.2 Lesson 2 – rainflow counting on user data

This example show how to perform rainflow counting on self generated signal (or self loaded to the MATLAB Command Window):

```
clear all
lengths=10000;
s=10*randn(lengths,1)+rand(lengths,1);
% or load the signal from file --> s=my_signal;
tp=sig2ext(s);
% use the sig2ext function ever before you run rainflow,
% because the counting procedure works properly only on
% turning points
rf=rainflow(tp);
rfm=rfmatrix(rf,20,20);
surf(rfm)
```

11.29.4.3 Lesson 3 – fatigue life calculation

The script presented bellow realize fatigue calculation in the case of uniaxial random loading without any additional influence of notch, stress gradient, environment etc. In this example linear hypothesis of damage accumulation is used (full range Miner rule without omission of the cycles with amplitudes lover then endurance limit) combined with typical S-N curve (Wöhler curve).

```
clear all

% S-N curve parameters:
```

```
sigaf=250;   % endurance limit
m=8;         % slope of the curve
Nk=2e6;      % number of cycle for knee point

% random stress time history generated from random generator
% can be replaced by another signal: sigt=my_stress;
sigt=100*randn(1000,1);

% length in second of the time history
To=500;

tp=sig2ext(sigt);    % turning points
rf=rainflow(tp);     % rainflow
CycleRate=rf(3,:);   % number of cycles
siga=rf(1,:);        % cycle amplitudes

% calculation of the damage
damage=sum((CycleRate/Nk).*((siga/sigaf).^m));

% expected time to failure in seconds
T=To/damage;

% expected time to failure in days
disp(['Calculated fatigue life in days: ' num2str(T/3600/24)])
```

Hint: copy and paste the text in blue into the Script Editor and press the “run button”

11.29.5 References

[1] ASTM E 1049-85 (Reapproved 1997), *Standard practices for cycle counting in fatigue analysis*, in: Annual Book of ASTM Standards, Vol. 03.01, Philadelphia 1999, pp. 710-718

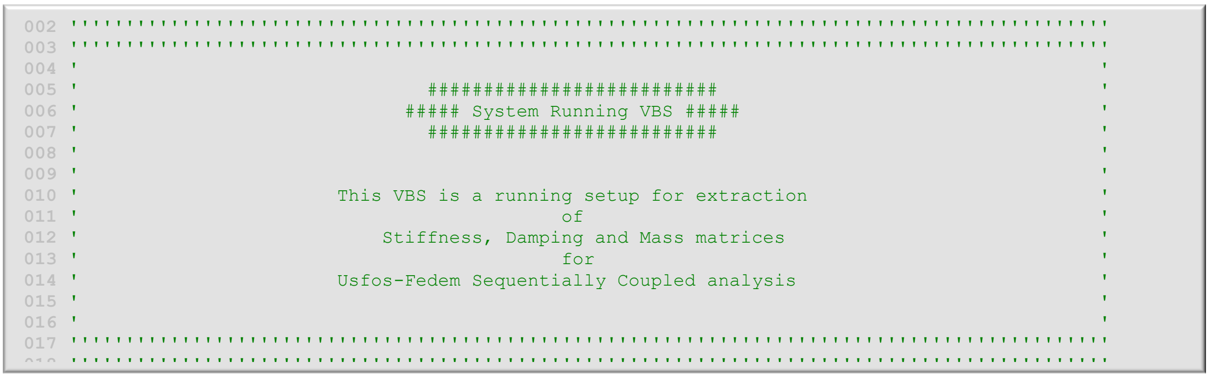
[2] NIESŁONY A., *Determination of fragments of multiaxial service loading strongly influencing the fatigue of machine components*, Mechanical Systems and Signal Processing, Vol. 23(8), 2009, pp. 2712-2721

11.30 Matrices extraction tool scripts

11.30.1 01_01_SRM

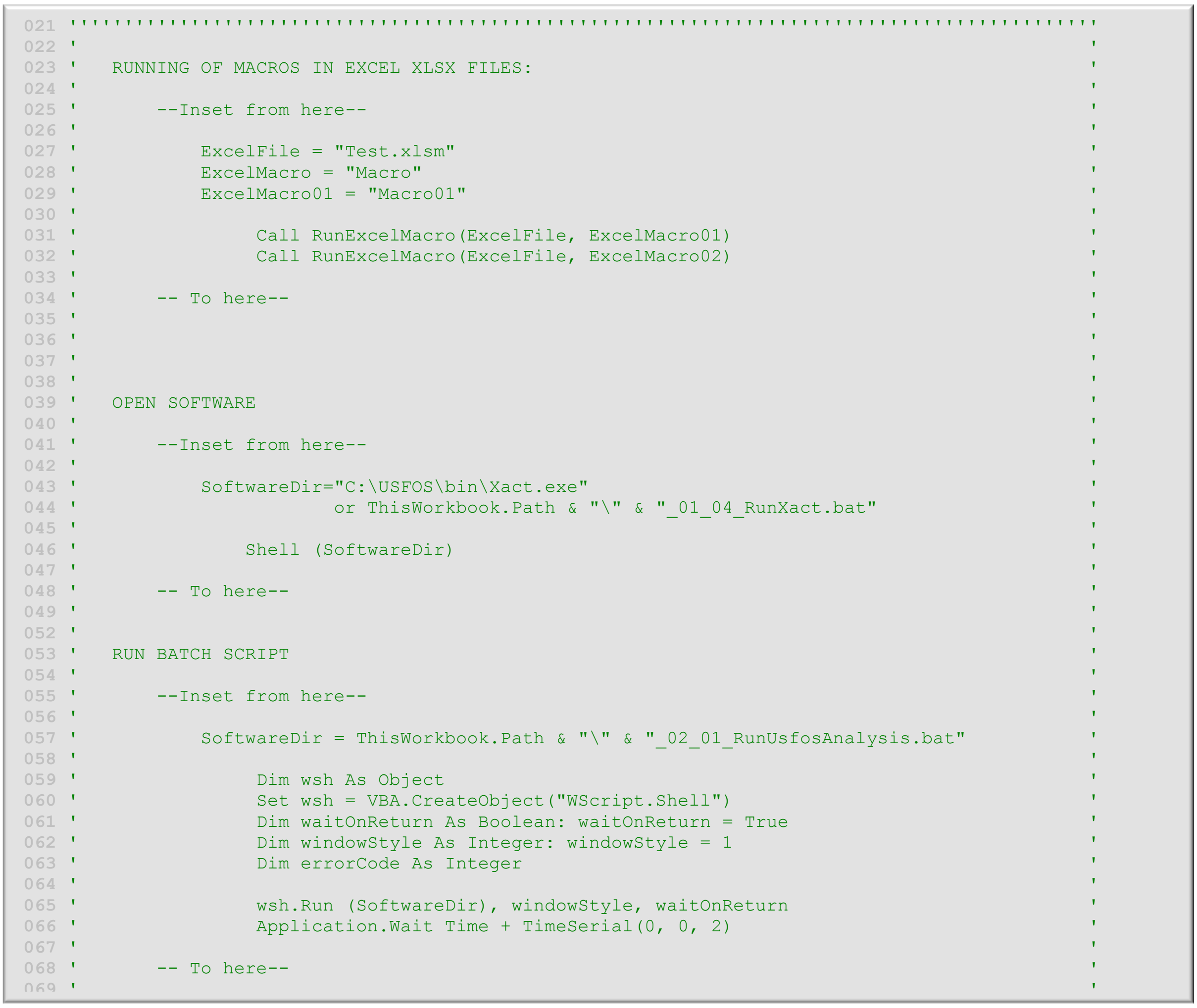
The SRM script (SystemRunMacro) is the macro calling and running all parts of the matrices extraction tools in right sequence.

Code 23 - SRM part 1



```
'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
'                                                                                           '
'                               ##########################                                  '
'                             ##### System Running VBS #####                                '
'                               ##########################                                  '
'                                                                                           '
'                                                                                           '
'                       This VBS is a running setup for extraction                          '
'                                           of                                              '
'                           Stiffness, Damping and Mass matrices                            '
'                                           for                                             '
'                       Usfos-Fedem Sequentially Coupled analysis                           '
'                                                                                           '
'                                                                                           '
'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
```

Code 24 – SRM part 2



```
'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
'                                                                                           '
'   RUNNING OF MACROS IN EXCEL XLSX FILES:                                                  '
'                                                                                           '
'       --Inset from here--                                                                 '
'                                                                                           '
'           ExcelFile = "Test.xlsm"                                                         '
'           ExcelMacro = "Macro"                                                            '
'           ExcelMacro01 = "Macro01"                                                        '
'                                                                                           '
'                Call RunExcelMacro(ExcelFile, ExcelMacro01)                                '
'                Call RunExcelMacro(ExcelFile, ExcelMacro02)                                '
'                                                                                           '
'       -- To here--                                                                        '
'                                                                                           '
'                                                                                           '
'                                                                                           '
'                                                                                           '
'   OPEN SOFTWARE                                                                           '
'                                                                                           '
'       --Inset from here--                                                                 '
'                                                                                           '
'           SoftwareDir="C:\USFOS\bin\Xact.exe"                                             '
'                       or ThisWorkbook.Path & "\" & "_01_04_RunXact.bat"                   '
'                                                                                           '
'               Shell (SoftwareDir)                                                         '
'                                                                                           '
'       -- To here--                                                                        '
'                                                                                           '
'                                                                                           '
'   RUN BATCH SCRIPT                                                                        '
'                                                                                           '
'       --Inset from here--                                                                 '
'                                                                                           '
'           SoftwareDir = ThisWorkbook.Path & "\" & "_02_01_RunUsfosAnalysis.bat"           '
'                                                                                           '
'                Dim wsh As Object                                                          '
'                Set wsh = VBA.CreateObject("WScript.Shell")                                '
'                Dim waitOnReturn As Boolean: waitOnReturn = True                           '
'                Dim windowStyle As Integer: windowStyle = 1                                '
'                Dim errorCode As Integer                                                   '
'                                                                                           '
'                wsh.Run (SoftwareDir), windowStyle, waitOnReturn                           '
'                Application.Wait Time + TimeSerial(0, 0, 2)                                '
'                                                                                           '
'       -- To here--                                                                        '
'                                                                                           '
```

Code 25 - SRM part 3

```
'                                                                                            '
'   TIMER                                                                                    '
'                                                                                            '
'       --Inset from here--                                                                  '
'                                                                                            '
'           Application.Wait Time + TimeSerial(0,0,2)                                        '
'                                                                                            '
'       -- To here--                                                                         '
'                                                                                            '
'                                                                                            '
'                                                                                            '
'   KEYS                                                                                     '
'                                                                                            '
'       --Inset from here--                                                                  '
'                                                                                            '
'           SendKeys "^o"       CTRL+O                                                       '
'           SendKeys "%"        ALT                                                          '
'           SendKeys "~"        ENTER                                                        '
'                                                                                            '
'       -- To here--                                                                         '
'                                                                                            '
'                                                                                            '
'                                                                                            '
'   LOOP                                                                                     '
'                                                                                            '
'       --Inset from here--                                                                  '
'                                                                                            '
'           For i = 1 To 6                                                                   '
'                                                                                            '
'               i                                                                            '
'                                                                                            '
'           Next i                                                                           '
'                                                                                            '
'       -- To here--                                                                         '
'                                                                                            '
'                                                                                            '
'                                                                                            '
''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''

```

Code 26 - SRM part 4

```
Public Function RunExcelMacro(FileName As String, Macro As String)
    Set xl = CreateObject("Excel.application")
    xl.Application.Workbooks.Open ThisWorkbook.Path & "\" & FileName
    xl.Application.Visible = True
                xl.Application.Run FileName & "!" & Macro
    Set xl = Nothing
    Application.Wait Time + TimeSerial(0, 0, 2)
    SendKeys "%{F4}"
    SendKeys "i"
    Application.Wait Time + TimeSerial(0, 0, 2)
End Function
Public Function OpenExcelFile(FileName As String, Macro As String)

    Set xl = CreateObject("Excel.application")
    xl.Application.Workbooks.Open ThisWorkbook.Path & "\" & FileName
    xl.Application.Visible = True
                xl.Application.Run FileName & "!" & Macro
    Set xl = Nothing
End Function
```

Code 27 - SRM part 5

```
' Deactivate window error messages
        Application.DisplayAlerts = False



' Set Variables As String

        Dim ExcelFile As String
        Dim ExcelFile01 As String
        Dim ExcelFile02 As String
        Dim ExcelFile03 As String
        Dim ExcelFile04 As String
        Dim ExcelFile05 As String
        Dim ExcelFile06 As String
        Dim ExcelFile07 As String
        Dim ExcelFile08 As String
        Dim ExcelFile09 As String
        Dim ExcelFile10 As String
        Dim ExcelMacro As String
        Dim ExcelMacro01 As String
        Dim ExcelMacro02 As String
        Dim ExcelMacro03 As String
        Dim ExcelMacro04 As String
        Dim ExcelMacro05 As String
        Dim ExcelMacro06 As String
        Dim ExcelMacro07 As String
        Dim ExcelMacro08 As String
        Dim ExcelMacro09 As String
        Dim ExcelMacro10 As String
```

Code 28 – SRM part 6

```
' Run   MAIF            - Matrix Analysis Input Files
'       CF_B            - Control File B
'       CF_K            - Control File K
'       CF_C            - Control File C
'       CF_M            - Control File M


        ExcelFile = "01_02_MAIF.xlsm"
        ExcelMacro01 = "CF_B"
        ExcelMacro02 = "CF_K"
        ExcelMacro03 = "CF_C"
        ExcelMacro04 = "CF_M"


            Call RunExcelMacro(ExcelFile, ExcelMacro01)
            Call RunExcelMacro(ExcelFile, ExcelMacro02)
            Call RunExcelMacro(ExcelFile, ExcelMacro03)
            Call RunExcelMacro(ExcelFile, ExcelMacro04)
```

Code 29 - SRM part 7

```
' Run   ML              - Matrix Load
'       ML_FG_B         - Matrix Load File Generator B
'       ML_FG_K         - Matrix Load File Generator K
'       ML_FG_C         - Matrix Load File Generator C
'       ML_FG_M         - Matrix Load File Generator M


        ExcelFile = "01_03_ML.xlsm"
        ExcelMacro01 = "ML_FG_B"
        ExcelMacro02 = "ML_FG_K"
        ExcelMacro03 = "ML_FG_C"
        ExcelMacro04 = "ML_FG_M"


            Call RunExcelMacro(ExcelFile, ExcelMacro01)
            Call RunExcelMacro(ExcelFile, ExcelMacro02)
            Call RunExcelMacro(ExcelFile, ExcelMacro03)
            Call RunExcelMacro(ExcelFile, ExcelMacro04)
```

Code 30 - SRM part 8

```
' Run   RABG            - Run Analyses Bat Generator
'       RABG            - Run Analyses Bat Generator

        ExcelFile = "01_04_RABG.xlsm"
        ExcelMacro = "RABG"


            Call RunExcelMacro(ExcelFile, ExcelMacro)
```

Code 31 - SRM part 9

```
' Run   MEABG           - Matrix Extract Analysis Bat Generator
'       MEABGU          - Matrix Extract Analysis Bat Generator
Usfos


        ExcelFile = "01_05_MEABG.xlsm"
        ExcelMacro01 = "MEABGU_B"
        ExcelMacro02 = "MEABGU_K"
        ExcelMacro03 = "MEABGU_C"
        ExcelMacro04 = "MEABGU_M"


            Call RunExcelMacro(ExcelFile, ExcelMacro01)
            Call RunExcelMacro(ExcelFile, ExcelMacro02)
            Call RunExcelMacro(ExcelFile, ExcelMacro03)
            Call RunExcelMacro(ExcelFile, ExcelMacro04)
```

Code 32 - SRM part 10

```
' Run Usfos Analysis:

        SoftwareDir = ThisWorkbook.Path & "\" &
"_02_01_RunUsfosAnalysis.bat"

                Dim wsh As Object
                Set wsh = VBA.CreateObject("WScript.Shell")
                Dim waitOnReturn As Boolean: waitOnReturn = True
                Dim windowStyle As Integer: windowStyle = 1
                Dim errorCode As Integer

                wsh.Run (SoftwareDir), windowStyle, waitOnReturn
                Application.Wait Time + TimeSerial(0, 0, 2)
```

Code 33 - SRM part 11

```
' Run   MEA_XS          - Matrix Extraction Analysis Xact Script
'       MEA_XS          - Matrix Extraction Analysis Xact Script

        ExcelFile = "02_01_MEA_XS.xlsm"
        ExcelMacro01 = "MEA_XS_B"
        ExcelMacro02 = "MEA_XS_K"
        ExcelMacro03 = "MEA_XS_C"
        ExcelMacro04 = "MEA_XS_M"


            Call RunExcelMacro(ExcelFile, ExcelMacro01)
            Call RunExcelMacro(ExcelFile, ExcelMacro02)
            Call RunExcelMacro(ExcelFile, ExcelMacro03)
            Call RunExcelMacro(ExcelFile, ExcelMacro04)
```

Code 34 - SRM part 12

```
' Run   XME             - Xact Matrix Extraction
'       XME_B           - Xact Matrix Extraction B
'       XME_K           - Xact Matrix Extraction K
'       XME_C           - Xact Matrix Extraction C
'       XME_M           - Xact Matrix Extraction M

        ExcelFile = "02_02_XME.xlsm"
        ExcelMacro01 = "XME_B"
        ExcelMacro02 = "XME_K"
        ExcelMacro03 = "XME_C"
        ExcelMacro04 = "XME_M"


            Call RunExcelMacro(ExcelFile, ExcelMacro01)
            Call RunExcelMacro(ExcelFile, ExcelMacro02)
            Call RunExcelMacro(ExcelFile, ExcelMacro03)
            Call RunExcelMacro(ExcelFile, ExcelMacro04)
```

Code 35 - SRM part 13

```
' Run   MEA_XTP_B       - Matrix Extration Analysis Xact Table Processing B
'       MEA_XTP_B       - Matrix Extration Analysis Xact Table Processing B

        ExcelFile = "03_00_MEA_XTP_B.xlsm"
        ExcelMacro = "MEA_XTP_B_BL_TG_BL"
        ExcelMacro01 = "MEA_XTP_B"
        ExcelMacro02 = "MEA_XTP_BL"
        ExcelMacro03 = "MEA_TG_B"
        ExcelMacro04 = "MEA_TG_BL"
        ExcelMacro05 = "MEA_ME_B"
        ExcelMacro06 = "MEA_ME_BL"


            Call OpenExcelFile(ExcelFile, ExcelMacro)
'            Call OpenExcelFile(ExcelFile, ExcelMacro01)
'            Call OpenExcelFile(ExcelFile, ExcelMacro02)
'            Call OpenExcelFile(ExcelFile, ExcelMacro03)
'            Call OpenExcelFile(ExcelFile, ExcelMacro04)
'            Call OpenExcelFile(ExcelFile, ExcelMacro05)
'            Call OpenExcelFile(ExcelFile, ExcelMacro06)
```

Code 36 - SRM part 14

```
' Run   MEA_XTP_K       - Matrix Extration Analysis Xact Table Processing K
'       MEA_XTP_K       - Matrix Extration Analysis Xact Table Processing K

        ExcelFile = "03_01_MEA_XTP_K.xlsm"
        ExcelMacro = "MEA_XTP_K_KL_TG_KL"
        ExcelMacro01 = "MEA_XTP_K"
        ExcelMacro02 = "MEA_XTP_KL"
        ExcelMacro03 = "MEA_TG_K"
        ExcelMacro04 = "MEA_TG_KL"
        ExcelMacro05 = "MEA_ME_K"
        ExcelMacro06 = "MEA_ME_KL"


            Call OpenExcelFile(ExcelFile, ExcelMacro)
'            Call OpenExcelFile(ExcelFile, ExcelMacro01)
'            Call OpenExcelFile(ExcelFile, ExcelMacro02)
'            Call OpenExcelFile(ExcelFile, ExcelMacro03)
'            Call OpenExcelFile(ExcelFile, ExcelMacro04)
'            Call OpenExcelFile(ExcelFile, ExcelMacro05)
'            Call OpenExcelFile(ExcelFile, ExcelMacro06)
```

Code 37 - SRM part 15

```
' Run   MEA_XTP_C       - Matrix Extration Analysis Xact Table Processing C
'       MEA_XTP_C       - Matrix Extration Analysis Xact Table Processing C

        ExcelFile = "03_02_MEA_XTP_C.xlsm"
        ExcelMacro = "MEA_XTP_C_CL_TG_CL"
        ExcelMacro01 = "MEA_XTP_C"
        ExcelMacro02 = "MEA_XTP_CL"
        ExcelMacro03 = "MEA_TG_C"
        ExcelMacro04 = "MEA_TG_CL"
        ExcelMacro05 = "MEA_ME_C"
        ExcelMacro06 = "MEA_ME_CL"


            Call OpenExcelFile(ExcelFile, ExcelMacro)
'            Call OpenExcelFile(ExcelFile, ExcelMacro01)
'            Call OpenExcelFile(ExcelFile, ExcelMacro02)
'            Call OpenExcelFile(ExcelFile, ExcelMacro03)
'            Call OpenExcelFile(ExcelFile, ExcelMacro04)
'            Call OpenExcelFile(ExcelFile, ExcelMacro05)
'            Call OpenExcelFile(ExcelFile, ExcelMacro06)
```

Code 38 - Matrices extraction tool system run VBS part 16

```
' Run   MEA_XTP_M       - Matrix Extration Analysis Xact Table Processing M
'       MEA_XTP_M       - Matrix Extration Analysis Xact Table Processing M

        ExcelFile = "03_03_MEA_XTP_M.xlsm"
        ExcelMacro = "MEA_XTP_M_ML_TG_ML"
        ExcelMacro01 = "MEA_XTP_M"
        ExcelMacro02 = "MEA_XTP_ML"
        ExcelMacro03 = "MEA_TG_M"
        ExcelMacro04 = "MEA_TG_ML"
        ExcelMacro05 = "MEA_ME_M"
        ExcelMacro06 = "MEA_ME_ML"


            Call OpenExcelFile(ExcelFile, ExcelMacro)
'            Call OpenExcelFile(ExcelFile, ExcelMacro01)
'            Call OpenExcelFile(ExcelFile, ExcelMacro02)
'            Call OpenExcelFile(ExcelFile, ExcelMacro03)
'            Call OpenExcelFile(ExcelFile, ExcelMacro04)
'            Call OpenExcelFile(ExcelFile, ExcelMacro05)
'            Call OpenExcelFile(ExcelFile, ExcelMacro06)
End Sub
```

11.30.2 01_02_MAIF

This workbook writes all the control files for all matrices (B, C, K and M) for the analysis to work folder.

Code 39 - CF_B

```
Sub CF_B()


''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''

' For all matrix dofs
For i = 1 To 6

    Sheets(1).Select
    FName = "CF_B"

        ActiveSheet.SaveAs Filename:= _
        ThisWorkbook.Path & "\" & FName & i & ".fem", _
        FileFormat:=xlTextPrinter, _
        CreateBackup:=False

Next i

''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
End Sub
```

Code 40 - CF_K

```
Sub CF_K()

''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''


' For all matrix dofs
For i = 1 To 6

    Sheets(2).Select
    FName = "CF_K"

        ActiveSheet.SaveAs Filename:= _
        ThisWorkbook.Path & "\" & FName & i & ".fem", _
        FileFormat:=xlTextPrinter, _
        CreateBackup:=False

Next i



''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
End Sub
```

Code 41 - CF_C

```
Sub CF_C()


'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''


' For all matrix dofs
For i = 1 To 6

    Sheets(3).Select
    FName = "CF_C"
    CRange = "M"

    Range(CRange & 42 + i).Select
    ActiveCell.FormulaR1C1 = "Activate"

        ActiveSheet.SaveAs Filename:= _
        ThisWorkbook.Path & "\" & FName & i & ".fem", _
        FileFormat:=xlTextPrinter, _
        CreateBackup:=False

    Range(CRange & 42 + i).Select
    ActiveCell.FormulaR1C1 = ""

Next i


'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''


End Sub
```

Code 42 - CF_M

```
Sub CF_M()



''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''


' For all matrix dofs
For i = 1 To 6

    Sheets(4).Select
    FName = "CF_M"

        ActiveSheet.SaveAs Filename:= _
        ThisWorkbook.Path & "\" & FName & i & ".fem", _
        FileFormat:=xlTextPrinter, _
        CreateBackup:=False

Next i



''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''



End Sub
```

11.30.3 01_03_ML

This workbook writes all matrix load files to work folder for all matrices.

Code 43 - ML_FG_B

```vb
Sub ML_FG_B()
''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''

' For all matrix dofs
For i = 1 To 6

    Sheets(1).Select
    FName = "ML_B"
    CRange = "N"

    Range(CRange & 1 + i).Select
    ActiveCell.FormulaR1C1 = "Activate"

        ActiveSheet.SaveAs Filename:= _
        ThisWorkbook.Path & "\" & FName & i & ".fem", _
        FileFormat:=xlTextPrinter, _
        CreateBackup:=False

    Range(CRange & 1 + i).Select
    ActiveCell.FormulaR1C1 = ""

Next i


''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
End Sub

```

Code 44 - ML_FG_K

```
Sub ML_FG_K()


''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''

' For all matrix dofs
For i = 1 To 6

    Sheets(2).Select
    FName = "ML_K"
    CRange = "N"

    Range(CRange & 1 + i).Select
    ActiveCell.FormulaR1C1 = "Activate"

        ActiveSheet.SaveAs Filename:= _
        ThisWorkbook.Path & "\" & FName & i & ".fem", _
        FileFormat:=xlTextPrinter, _
        CreateBackup:=False

    Range(CRange & 1 + i).Select
    ActiveCell.FormulaR1C1 = ""

Next i


''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''


End Sub
```

Code 45 - ML_FG_C

```
Sub ML_FG_C()



''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''


' For all matrix dofs
For i = 1 To 6

    Sheets(3).Select
    FName = "ML_C"
    CRange = "N"

    Range(CRange & 1 + i).Select
    ActiveCell.FormulaR1C1 = "Activate"

        ActiveSheet.SaveAs Filename:= _
        ThisWorkbook.Path & "\" & FName & i & ".fem", _
        FileFormat:=xlTextPrinter, _
        CreateBackup:=False

    Range(CRange & 1 + i).Select
    ActiveCell.FormulaR1C1 = ""

Next i


''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''



End Sub
```

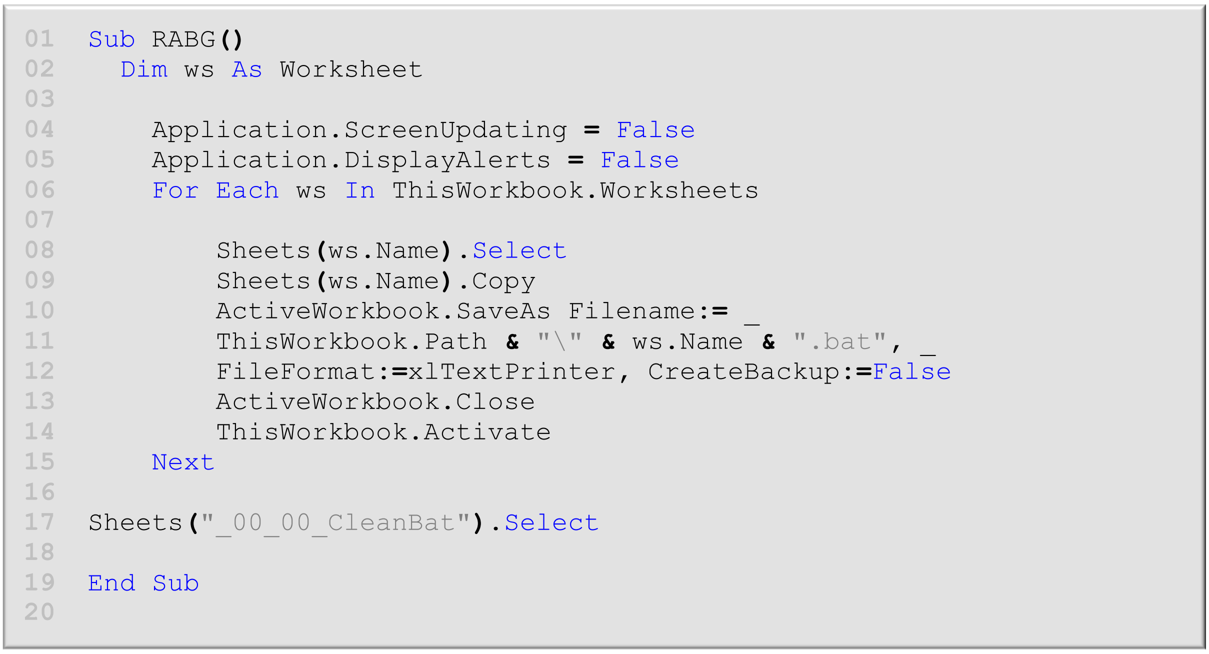
Code 46 - ML_FG_M

```
Sub ML_FG_M()


'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''


' For all matrix dofs
For i = 1 To 6

    Sheets(4).Select
    FName = "ML_M"
    CRange = "N"

    Range(CRange & 1 + i).Select
    ActiveCell.FormulaR1C1 = "Activate"

        ActiveSheet.SaveAs Filename:= _
        ThisWorkbook.Path & "\" & FName & i & ".fem", _
        FileFormat:=xlTextPrinter, _
        CreateBackup:=False

    Range(CRange & 1 + i).Select
    ActiveCell.FormulaR1C1 = ""

Next i


'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''


End Sub
```

11.30.4 01_04_RABG

This workbook writes the Run Analyse Bat Generator files to working folder

Code 47 - RABG



```
Sub RABG()
  Dim ws As Worksheet

    Application.ScreenUpdating = False
    Application.DisplayAlerts = False
    For Each ws In ThisWorkbook.Worksheets

        Sheets(ws.Name).Select
        Sheets(ws.Name).Copy
        ActiveWorkbook.SaveAs Filename:= _
        ThisWorkbook.Path & "\" & ws.Name & ".bat", _
        FileFormat:=xlTextPrinter, CreateBackup:=False
        ActiveWorkbook.Close
        ThisWorkbook.Activate
    Next

Sheets("_00_00_CleanBat").Select

End Sub

```

11.30.5 01_05_MEABG

This workbook writes all analyse run batch script for the matrices to working folder.

Code 48 - MEABGU_B

```
Sub MEABGU_B()


'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''

' Buckling matrice B

'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''

AnalyseName = "MA_B"

HeadFile = "CF_B"
ModelFile = "ModF_B"
ExFile = "ML_B"
OutputFile = "MEA_B"

DynamicWind = "n"
Windfile = ""

    For i = 1 To 6

        Range("E10").Select
        ActiveCell.FormulaR1C1 = AnalyseName & i

        Range("E12").Select
        ActiveCell.FormulaR1C1 = HeadFile & i

        Range("E13").Select
        ActiveCell.FormulaR1C1 = ModelFile

        Range("E14").Select
        ActiveCell.FormulaR1C1 = ExFile & i

        Range("E15").Select
        ActiveCell.FormulaR1C1 = OutputFile & i

        Range("E17").Select
        ActiveCell.FormulaR1C1 = DynamicWind

        Range("E18").Select
        ActiveCell.FormulaR1C1 = Windfile

        Call UsfosToFile

    Next i

'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''

End Sub
```

Code 49 - MEABGU_K

```
Sub MEABGU_K()

'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''

' Stiffness matrice K

'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''

AnalyseName = "MA_K"

HeadFile = "CF_K"
ModelFile = "ModF_K"
ExFile = "ML_K"
OutputFile = "MEA_K"

DynamicWind = "n"
Windfile = ""

    For i = 1 To 6

        Range("E10").Select
        ActiveCell.FormulaR1C1 = AnalyseName & i

        Range("E12").Select
        ActiveCell.FormulaR1C1 = HeadFile & i

        Range("E13").Select
        ActiveCell.FormulaR1C1 = ModelFile

        Range("E14").Select
        ActiveCell.FormulaR1C1 = ExFile & i

        Range("E15").Select
        ActiveCell.FormulaR1C1 = OutputFile & i

        Range("E17").Select
        ActiveCell.FormulaR1C1 = DynamicWind

        Range("E18").Select
        ActiveCell.FormulaR1C1 = Windfile

        Call UsfosToFile

    Next i

'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''

End Sub
```

Code 50 - MEABGU_C

```
Sub MEABGU_C()


''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''

' Damping matrice C

''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''

AnalyseName = "MA_C"

HeadFile = "CF_C"
ModelFile = "ModF_C"
ExFile = "ML_C"
OutputFile = "MEA_C"

DynamicWind = "n"
Windfile = ""




    For i = 1 To 6

        Range("E10").Select
        ActiveCell.FormulaR1C1 = AnalyseName & i

        Range("E12").Select
        ActiveCell.FormulaR1C1 = HeadFile & i

        Range("E13").Select
        ActiveCell.FormulaR1C1 = ModelFile

        Range("E14").Select
        ActiveCell.FormulaR1C1 = ExFile & i

        Range("E15").Select
        ActiveCell.FormulaR1C1 = OutputFile & i

        Range("E17").Select
        ActiveCell.FormulaR1C1 = DynamicWind

        Range("E18").Select
        ActiveCell.FormulaR1C1 = Windfile

        Call UsfosToFile

    Next i

''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''

End Sub
```

Code 51 - MEABGU_M

```
Sub MEABGU_M()
'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''

' Mass matrice M

'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''

AnalyseName = "MA_M"

HeadFile = "CF_M"
ModelFile = "ModF_M"
ExFile = "ML_M"
OutputFile = "MEA_M"

DynamicWind = "n"
Windfile = ""



    For i = 1 To 6

        Range("E10").Select
        ActiveCell.FormulaR1C1 = AnalyseName & i

        Range("E12").Select
        ActiveCell.FormulaR1C1 = HeadFile & i

        Range("E13").Select
        ActiveCell.FormulaR1C1 = ModelFile

        Range("E14").Select
        ActiveCell.FormulaR1C1 = ExFile & i

        Range("E15").Select
        ActiveCell.FormulaR1C1 = OutputFile & i

        Range("E17").Select
        ActiveCell.FormulaR1C1 = DynamicWind

        Range("E18").Select
        ActiveCell.FormulaR1C1 = Windfile

        Call UsfosToFile

    Next i

'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
End Sub
```

Code 52 - UsfosToFile

```
Sub UsfosToFile()

FName01 = Sheets("Usfos_Analysis").Range("C2").Text
FName02 = Sheets("Usfos_Analysis").Range("D2").Text

    Sheets("Usfos_Analysis").Select
    ActiveWorkbook.SaveAs Filename:= _
    ThisWorkbook.Path & "\" & FName01 & ".bat", _
    FileFormat:=xlText, CreateBackup:=False

    Sheets("Usfos_Analysis_Redir").Select
    ActiveWorkbook.SaveAs Filename:= _
    ThisWorkbook.Path & "\" & FName02 & ".txt", _
    FileFormat:=xlText, CreateBackup:=False

ThisWorkbook.Sheets(1).Name = "Usfos_Analysis"
ThisWorkbook.Sheets(2).Name = "Usfos_Analysis_Redir"

Sheets("Analyse_Setup").Select

End Sub
```

11.30.6 02_01_MEA_XS

This workbook writes the Xact result extraction scripts for all matrix analyses to working folder.

Code 53 - MEA_XS_B

```
Sub MEA_XS_B()


'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''' B-Matrix

FName = "MEA_XS_B"
A1Value = "plot history Time,-1,-1,-1,Nodal Displacement,1,-1,"
A2Value = "plot savedata MEA_XT_B"

For j = 1 To 6
    For i = 1 To 6
        Cells((i * 2) - 1, 1).Select
        ActiveCell.FormulaR1C1 = A1Value & i
        Cells((i * 2), 1).Select
        ActiveCell.FormulaR1C1 = A2Value & i & j

    Next i

        B1Value = "plot materialmodel Material Model,1,1,1,1"
        B2Value = "plot savedata flag"
        Cells((i * 2) - 1, 1).Select
        ActiveCell.FormulaR1C1 = B1Value
        Cells((i * 2), 1).Select
        ActiveCell.FormulaR1C1 = B2Value

        Application.ScreenUpdating = False
        Application.DisplayAlerts = False
            Sheets(1).Select
            Sheets(1).Copy
            ActiveWorkbook.SaveAs Filename:= _
            ThisWorkbook.Path & "\" & FName & j & ".usf", _
            FileFormat:=xlTextPrinter, CreateBackup:=False
            ActiveWorkbook.Close
            ThisWorkbook.Activate

Next j


End Sub
```

Code 54 - MEA_XS_K

```
Sub MEA_XS_K()


'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''' K-Matrix

FName = "MEA_XS_K"
A1Value = "plot history Time,-1,-1,-1,Nodal Force,1,-1,"
A2Value = "plot savedata MEA_XT_K"

For j = 1 To 6
    For i = 1 To 6
        Cells((i * 2) - 1, 1).Select
        ActiveCell.FormulaR1C1 = A1Value & i
        Cells((i * 2), 1).Select
        ActiveCell.FormulaR1C1 = A2Value & i & j

    Next i

        B1Value = "plot materialmodel Material Model,1,1,1,1"
        B2Value = "plot savedata flag"
        Cells((i * 2) - 1, 1).Select
        ActiveCell.FormulaR1C1 = B1Value
        Cells((i * 2), 1).Select
        ActiveCell.FormulaR1C1 = B2Value

        Application.ScreenUpdating = False
        Application.DisplayAlerts = False
            Sheets(1).Select
            Sheets(1).Copy
            ActiveWorkbook.SaveAs Filename:= _
            ThisWorkbook.Path & "\" & FName & j & ".usf", _
            FileFormat:=xlTextPrinter, CreateBackup:=False
            ActiveWorkbook.Close
            ThisWorkbook.Activate

Next j


End Sub
```

Code 55 - - MEA_XS_C

```
Sub MEA_XS_C()


''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''' C-Matrix

FName = "MEA_XS_C"
A1Value = "plot history Time,-1,-1,-1,Nodal Force,1,-1,"
A2Value = "plot savedata MEA_XT_C"

For j = 1 To 6
    For i = 1 To 6
        Cells((i * 2) - 1, 1).Select
        ActiveCell.FormulaR1C1 = A1Value & i
        Cells((i * 2), 1).Select
        ActiveCell.FormulaR1C1 = A2Value & i & j

    Next i

        B1Value = "plot materialmodel Material Model,1,1,1,1"
        B2Value = "plot savedata flag"
        Cells((i * 2) - 1, 1).Select
        ActiveCell.FormulaR1C1 = B1Value
        Cells((i * 2), 1).Select
        ActiveCell.FormulaR1C1 = B2Value

        Application.ScreenUpdating = False
        Application.DisplayAlerts = False
            Sheets(1).Select
            Sheets(1).Copy
            ActiveWorkbook.SaveAs Filename:= _
            ThisWorkbook.Path & "\" & FName & j & ".usf", _
            FileFormat:=xlTextPrinter, CreateBackup:=False
            ActiveWorkbook.Close
            ThisWorkbook.Activate

Next j

End Sub

```

Code 56 - MEA_XS_M

```
Sub MEA_XS_M()


'''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''
''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''''' M-Matrix


FName = "MEA_XS_M"
A1Value = "plot history Time,-1,-1,-1,Nodal Force,1,-1,"
A2Value = "plot savedata MEA_XT_M"




For j = 1 To 6
    For i = 1 To 6
        Cells((i * 2) - 1, 1).Select
        ActiveCell.FormulaR1C1 = A1Value & i
        Cells((i * 2), 1).Select
        ActiveCell.FormulaR1C1 = A2Value & i & j

    Next i

        B1Value = "plot materialmodel Material Model,1,1,1,1"
        B2Value = "plot savedata flag"
        Cells((i * 2) - 1, 1).Select
        ActiveCell.FormulaR1C1 = B1Value
        Cells((i * 2), 1).Select
        ActiveCell.FormulaR1C1 = B2Value

        Application.ScreenUpdating = False
        Application.DisplayAlerts = False
            Sheets(1).Select
            Sheets(1).Copy
            ActiveWorkbook.SaveAs Filename:= _
            ThisWorkbook.Path & "\" & FName & j & ".usf", _
            FileFormat:=xlTextPrinter, CreateBackup:=False
            ActiveWorkbook.Close
            ThisWorkbook.Activate
Next j



End Sub
```

11.30.7 02_02_XME

This workbook run all Xact result extraction scripts in Xact for all analyses of the matrices B, K, C and M.

Code 57 - XME_Flag 1

```
Public Sub FlagDelete(FlagName As String)

      FindIt = Dir(ThisWorkbook.Path & "\" & FlagName)

      If Not Len(FindIt) = 0 Then
         Kill ThisWorkbook.Path & "\" & FlagName
      End If

End Sub

```

Code 58 - XME_Flag 2

```
Public Sub FlagWait(FlagName As String)
      FindIt = Dir(ThisWorkbook.Path & "\" & FlagName)

      While Len(FindIt) = 0

      FindIt = Dir(ThisWorkbook.Path & "\" & FlagName)

      Wend

End Sub
```

Code 59 - XME_B

```
Sub XME_B()

' Deactivate window error messages
  Application.DisplayAlerts = False

' Delete flag from last sequence
  FlagDelete ("flag")

' Open Xact software:
  Shell (ThisWorkbook.Path & "\" & "_01_04_RunXact.bat")
'  ("C:\USFOS\bin\Xact.exe")

' Waits for (5) seconds
  Application.Wait Time + TimeSerial(0, 0, 5)

 For i = 1 To 6

  ' Opens analyse file
    SendKeys "^o"
    Application.Wait Time + TimeSerial(0, 0, 2)

  ' Analyse file name:
    SendKeys "MEA_B" & i & ".raf"
    SendKeys "~"
    Application.Wait Time + TimeSerial(0, 0, 10)



  ' Open Script file
    SendKeys "%"
    SendKeys "r"
    SendKeys "n"
    Application.Wait Time + TimeSerial(0, 0, 2)

  ' Script file name:
    SendKeys "MEA_XS_B" & i & ".usf"
    SendKeys "~"


  ' Wait for flag
    FlagWait ("flag")

  ' Delete flag from last sequence
    Application.Wait Time + TimeSerial(0, 0, 2)
    FlagDelete ("flag")

Next i


' Close Xact
  SendKeys "%"
  SendKeys "f"
  SendKeys "x"
  Application.Wait Time + TimeSerial(0, 0, 2)

End Sub
```

Code 60 - XME_K

```
Sub XME_K()

' Deactivate window error messages
    Application.DisplayAlerts = False

' Delete flag from last sequence
    FlagDelete ("flag")

' Open Xact software:
    Shell (ThisWorkbook.Path & "\" & "_01_04_RunXact.bat")
'    ("C:\USFOS\bin\Xact.exe")

' Waits for (5) seconds
    Application.Wait Time + TimeSerial(0, 0, 5)


 For i = 1 To 6

    ' Opens analyse file
        SendKeys "^o"
        Application.Wait Time + TimeSerial(0, 0, 2)

    ' Analyse file name:
        SendKeys "MEA_K" & i & ".raf"
        SendKeys "~"
        Application.Wait Time + TimeSerial(0, 0, 10)



    ' Open Script file
        SendKeys "%"
        SendKeys "r"
        SendKeys "n"
        Application.Wait Time + TimeSerial(0, 0, 2)

    ' Script file name:
        SendKeys "MEA_XS_K" & i & ".usf"
        SendKeys "~"


    ' Wait for flag
        FlagWait ("flag")

    ' Delete flag from last sequence
        Application.Wait Time + TimeSerial(0, 0, 2)
        FlagDelete ("flag")

Next

' Close Xact
    SendKeys "%"
    SendKeys "f"
    SendKeys "x"
    Application.Wait Time + TimeSerial(0, 0, 2)

End Sub
```

Code 61 - XME_C

```
Sub XME_C()

' Deactivate window error messages
    Application.DisplayAlerts = False

' Delete flag from last sequence
    FlagDelete ("flag")


' Open Xact software:
    Shell (ThisWorkbook.Path & "\" & "_01_04_RunXact.bat")
'    ("C:\USFOS\bin\Xact.exe")

' Waits for (5) seconds
    Application.Wait Time + TimeSerial(0, 0, 5)

For i = 1 To 6

    ' Opens analyse file
        SendKeys "^o"
        Application.Wait Time + TimeSerial(0, 0, 2)

    ' Analyse file name:
        SendKeys "MEA_C" & i & ".raf"
        SendKeys "~"
        Application.Wait Time + TimeSerial(0, 0, 10)

    ' Open Script file
        SendKeys "%"
        SendKeys "r"
        SendKeys "n"
        Application.Wait Time + TimeSerial(0, 0, 2)

    ' Script file name:
        SendKeys "MEA_XS_C" & i & ".usf"
        SendKeys "~"


    ' Wait for flag
        FlagWait ("flag")

    ' Delete flag from last sequence
        Application.Wait Time + TimeSerial(0, 0, 2)
        FlagDelete ("flag")

Next i


' Close Xact
    SendKeys "%"
    SendKeys "f"
    SendKeys "x"
    Application.Wait Time + TimeSerial(0, 0, 2)

End Sub
```

Code 62 - XME_M

```
Sub XME_M()

' Deactivate window error messages
    Application.DisplayAlerts = False

' Delete flag from last sequence
    FlagDelete ("flag")


' Open Xact software:
    Shell (ThisWorkbook.Path & "\" & "_01_04_RunXact.bat")
'    ("C:\USFOS\bin\Xact.exe")

' Waits for (5) seconds
    Application.Wait Time + TimeSerial(0, 0, 5)


For i = 1 To 6

    ' Opens analyse file
        SendKeys "^o"
        Application.Wait Time + TimeSerial(0, 0, 2)

    ' Analyse file name:
        SendKeys "MEA_M" & i & ".raf"
        SendKeys "~"
        Application.Wait Time + TimeSerial(0, 0, 10)


    ' Open Script file
        SendKeys "%"
        SendKeys "r"
        SendKeys "n"
        Application.Wait Time + TimeSerial(0, 0, 2)

    ' Script file name:
        SendKeys "MEA_XS_M" & i & ".usf"
        SendKeys "~"


    ' Wait for flag
        FlagWait ("flag")

    ' Delete flag from last sequence
        Application.Wait Time + TimeSerial(0, 0, 2)
        FlagDelete ("flag")

Next i


' Close Xact
    SendKeys "%"
    SendKeys "f"
    SendKeys "x"
    Application.Wait Time + TimeSerial(0, 0, 2)

End Sub
```

11.30.8 03_01_MEA_XTP_B

This workbook import and postprocess data for matrix B.

Code 63 - MEA_XTP_B

```
Sub MEA_XTP_B()

'Activate Sheet 1
Sheets(1).Select

' Deletion of Sheet content
ActiveWorkbook.Sheets(1).Range("G:ZZ").ClearContents

' Specifies number of file extraction
NumberOfFiles = Sheets(1).Range("C2").Text

' Number of extraction repetitions
For i = 1 To NumberOfFiles
    ' Name of import file
    FName = Sheets(1).Cells(i + 3, 3).Text

            ' Import file
            With Sheets(1).QueryTables.Add(Connection:= _
                "TEXT;" & ThisWorkbook.Path & "\" & FName, _
                Destination:=Cells(2, 2 + (i * 5)))
                .Name = "FName"
                .FieldNames = True
                .FillAdjacentFormulas = False
                .PreserveFormatting = True
                .RefreshOnFileOpen = False
                .RefreshStyle = xlInsertDeleteCells
                .SavePassword = False
                .SaveData = True
                .AdjustColumnWidth = True
                .RefreshPeriod = 0
                .TextFilePromptOnRefresh = False
                .TextFilePlatform = 850
                .TextFileStartRow = 1
                .TextFileParseType = xlDelimited
                .TextFileTextQualifier = xlTextQualifierDoubleQuote
                .TextFileConsecutiveDelimiter = False
                .TextFileTabDelimiter = True
                .TextFileSemicolonDelimiter = False
                .TextFileCommaDelimiter = False
                .TextFileSpaceDelimiter = False
                .TextFileColumnDataTypes = Array(1, 1)
                .TextFileTrailingMinusNumbers = True
                .Refresh BackgroundQuery:=False
            End With

    ' Insert file name heading
    Cells(1, 2 + (i * 5)).Select
        ActiveCell.FormulaR1C1 = FName
Next i
End Sub
```

Code 64 - MEA_XTP_BL Part 1

```
Sub MEA_XTP_BL()

'Activate Sheet 1
Sheets(1).Select

' Specifies number of file extraction
NumberOfFiles = Sheets(1).Range("C2").Text

' Deletion of Sheet content
For i = 1 To NumberOfFiles
    ActiveWorkbook.Sheets(1).Range(Columns(4 + (i * 5)), Columns(5 + (i * 5))).ClearContents
Next i

' Insert file name heading

    For i = 1 To NumberOfFiles

    ' Name of import file
        FName = Sheets(1).Cells(i + 3, 3).Text

    ' Filename
        Cells(1, 4 + (i * 5)).Select
            ActiveCell.FormulaR1C1 = FName & "L"
    Next i

' Insert StartTime and RowNumber

    For i = 1 To NumberOfFiles

    ' StartTime
        Cells(2, 4 + (i * 5)).Select
            ActiveCell.FormulaR1C1 = "=" & "R" & (i + 3) & "C" & "4"
    ' StartRow
        Cells(2, 5 + (i * 5)).Select
           ActiveCell.FormulaR1C1 = "=MATCH(" & "R" & "2" & "C" & (4 + (i * 5)) & "," & "R1C" & (2 + (i * 5)) & ":R999999C" & (2 + (i * 5)) & ",0)"
    Next i

' Insert EndTime and RowNumber

    For i = 1 To NumberOfFiles

    ' EndTime
        Cells(3, 4 + (i * 5)).Select
            ActiveCell.FormulaR1C1 = "=" & "R" & (i + 3) & "C" & "5"
    ' EndRow
        Cells(3, 5 + (i * 5)).Select
           ActiveCell.FormulaR1C1 = "=MATCH(" & "R" & "3" & "C" & (4 + (i * 5)) & "," & "R1C" & (2 + (i * 5)) & ":R999999C" & (2 + (i * 5)) & ",0)"
    Next i
```

Code 65 - MEA_XTP_BL Part 2

```
' Insert TimeTable

    For i = 1 To NumberOfFiles

        FNameStart = Sheets(1).Cells(2, 5 + (i * 5)).Text
        FNameEnd = Sheets(1).Cells(3, 5 + (i * 5)).Text

        Cells(FNameStart, 4 + (i * 5)).Select
        ActiveCell.FormulaR1C1 = "=RC[-2]"
        Selection.AutoFill Destination:=Range(Cells(FNameStart, 4 + (i * 5)), Cells(FNameEnd, 4 + (i * 5))), Type:=xlFillDefault

    Next i

' Insert TrendTable

    For i = 1 To NumberOfFiles

        FNameStart = Sheets(1).Cells(2, 5 + (i * 5)).Text
        FNameEnd = Sheets(1).Cells(3, 5 + (i * 5)).Text

        Cells(FNameStart, 5 + (i * 5)).Select
        ActiveCell.FormulaR1C1 = "=TREND(" & "R" & FNameStart & "C" & (3 + (i * 5)) & ":" & "R" & FNameEnd & "C" & (3 + (i * 5)) & "," & "R" & FNameStart & "C" & (2 + (i * 5)) & ":" & "R" & FNameEnd & "C" & (2 + (i * 5)) & "," & "RC[-3]" & ")"

        Cells(FNameStart, 5 + (i * 5)).Select
        Selection.AutoFill Destination:=Range(Cells(FNameStart, 5 + (i * 5)), Cells(FNameEnd, 5 + (i * 5))), Type:=xlFillDefault
    Next i
End Sub
```

Code 66 - MEA_ME_B

```
Sub MEA_ME_B()


'Activate Sheet 4

    Sheets(4).Select

'Save Sheet 4 as txt

    ChDir ThisWorkbook.Path
    ActiveWorkbook.SaveAs Filename:= _
        ThisWorkbook.Path & "\" & "K_Matrix.txt" _
        , FileFormat:=xlText, CreateBackup:=False

End Sub
```

Code 67 - MEA_TG_B

```
Sub MEA_TG_B()
'Activate Sheet 1
Sheets(1).Cells(1, 1).Select


' Delete graphs in active sheet

    ' Dim wsItem As Worksheet
        Dim chtObj As ChartObject

    ' For Each chtObj In wsItem.ChartObjects
        For Each chtObj In ThisWorkbook.ActiveSheet.ChartObjects
            chtObj.Delete
        Next

' Specifies number of file extraction
    NumberOfFiles = Sheets(1).Range("C2").Text

' Specifies number of lines
    NumberOfLines = Sheets(1).Range("C3").Text

' Number of extraction repetitions
    For i = 1 To NumberOfFiles

    'Start-End
            FNameStart = Sheets(1).Cells(2, 5 + (i * 5)).Text
            FNameEnd = Sheets(1).Cells(3, 5 + (i * 5)).Text

    ' GenerateGraphs

        ' Insert Chart

            ActiveSheet.Shapes.AddChart.Select
            ActiveChart.ChartType = xlXYScatterSmoothNoMarkers

            ActiveChart.SeriesCollection(1).Delete
            ActiveChart.SeriesCollection(2).Delete

            ActiveChart.SeriesCollection.NewSeries
            ActiveChart.SeriesCollection(1).Name = Cells(1, 2 + (i *
5))
            ActiveChart.SeriesCollection(1).XValues = Range(Cells(5,
2 + (i * 5)), Cells(NumberOfLines, 2 + (i * 5)))
            ActiveChart.SeriesCollection(1).Values = Range(Cells(5,
3 + (i * 5)), Cells(NumberOfLines, 3 + (i * 5)))

            ActiveChart.SeriesCollection(2).Delete

        With ActiveChart
        '*** Change top left cell location to suit ***
            .Parent.Top = Cells(20, 2 + (i * 5)).Top
            .Parent.Left = Cells(20, 2 + (i * 5)).Left
        End With

    Next i

End Sub
```

Code 68 - DelGraphsSheet1

```
Sub DelGraphsSheet1()
'Activate Sheet 1
Sheets(1).Select


' Delete graphs in active sheet

    ' Dim wsItem As Worksheet
        Dim chtObj As ChartObject

    ' For Each chtObj In wsItem.ChartObjects
        For Each chtObj In ThisWorkbook.ActiveSheet.ChartObjects
            chtObj.Delete
        Next
End Sub
```