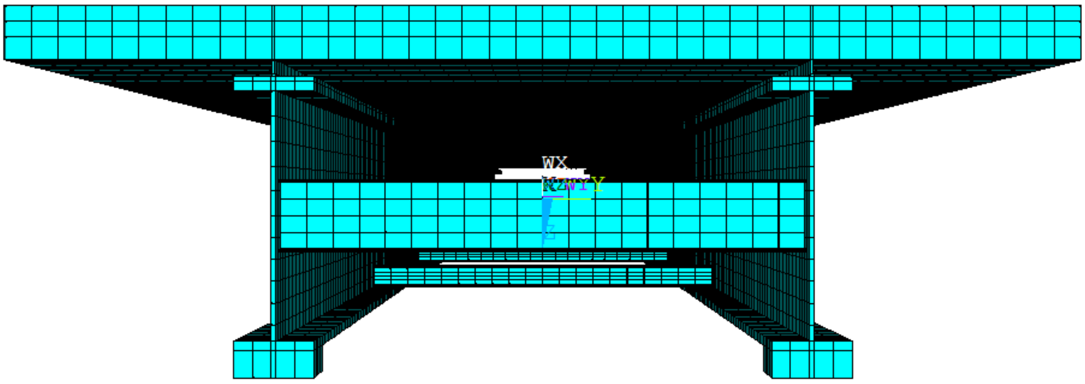
ZfP
Lehrstuhl für
Zerstörungsfreie Prüfung

Technische Universität München

Developing a Benchmark Study for Structural Health Monitoring

Sofia Artola



Scientific work to obtain the M.Sc. degree at the faculty of mechanical engineering at the Technical University of Munich.

Topic setter Univ.-Prof. Dr.-Ing. habil. Christian Große
Chair of Non-destructive Testing

Supervisor Dr. Alexander Mendler

Submitted by Sofia Artola

Submitted in Garching in Munich

TECHNISCHE UNIVERSITÄT MÜNCHEN - Lehrstuhl für Zerstörungsfreie Prüfung
Prof. Dr.-Ing. habil. Christian Große
Franz-Langinger-Straße 10 · 81245 München· Tel.+49-89-289-27221 · www.zfp.tum.de

Declaration of independence

Artola Sofia

Elisabethstraße 4

80796 Munich

Matriculation number: 03753452

Hiermit erkläre ich, die vorliegende Masterarbeit selbstständig verfasst zu haben. Es wurden keine anderen als die in der Arbeit angegebenen Quellen und Hilfsmittel benutzt. Die wörtlichen oder sinngemäß übernommenen Zitate habe ich als solche kenntlich gemacht.

Die vorliegende Masterarbeit entstand unter wissenschaftlicher und inhaltlicher Anleitung durch meine(n) Betreuer(in). Zentrale Ideen und Konzepte wurden gemeinschaftlich mit ihm/ihr erarbeitet.

München, den 29/09/2022

Unterschrift

ARTOLA SOFIA

Kurzfassung

Die Entwicklung einer Methodik zur genauen und zuverlässigen Zustandsbewertung von Bauwerken ist sehr wichtig geworden. Die Methode der Finite-Elemente-Modellaktualisierung bietet eine effiziente, zerstörungsfreie, globale Schadenserkennung, die auf der Tatsache beruht, dass die modalen Parameter (z. B. Eigenfrequenzen und Eigenformen) der Struktur durch strukturelle Schäden beeinflusst werden. Im FE-Modell wird die Schädigung durch eine Änderung der strukturellen Eigenschaften der Elemente dargestellt und kann durch Aktualisierung des FE-Modells anhand der gemessenen modalen Parameter identifiziert werden. In dieser Arbeit wird eine iterative, auf Sensitivität basierende Methode zur Aktualisierung des FE-Modells beschrieben, bei der die Diskrepanzen zwischen den Eigenfrequenzen des numerischen Modells und der realen Struktur minimiert werden. Weiterhin wird das Aktualisierungsverfahren auf das Modell der Universität der Bundeswehr (UniBw) in München angewendet. Dazu war es notwendig, das Modell zu erstellen und eine Schnittstelle zwischen der Finite-Elemente-Software (ANSYS) und einer Rechensoftware (MATLAB) zu entwickeln, die das Modellaktualisierungsverfahren auf das Finite-Elemente-Modell anwenden kann.

Schlagwörter: Überwachung des strukturellen Zustands, Modell-Kalibrierung, Model Updating, Schadenserkennung, sensitivitätsbasierte Model Updating

Abstract

The development of a methodology for accurate and reliable condition assessment of civil structures has become very important. The finite element (FE) model updating method provides an efficient, non-destructive, global damage identification technique, which is based on the fact that the modal parameters (e.g. natural frequencies and mode shapes) of the structure are affected by structural damage. In the FE model, the damage is represented by a change of the structural parameters and can be identified by updating the FE model to the measured modal parameters. This thesis describes an iterative sensitivity-based FE model updating method in which the discrepancies between the natural frequencies of the numerical model and the real structure are minimized. Furthermore, the updating procedure is applied to the model of the University of the Federal Armed Forces (UniBw) in Munich. For this purpose, it was necessary to design the model and develop an interface between the finite element software (ANSYS) and a computing software (MATLAB), which can apply the model updating technique to the finite element model.

Keywords: Structural Health Monitoring, Model Calibration, Model updating, Damage detection, Sensitivity-based model updating

Table of contents

List of figures

List of tables

List of abbreviations

APDL: ANSYS Parametric Design Language

FE: Finite Element

FEMU: Finite Element Model Updating

MAC: Modal Assurance Criterion

SHM: Structural Health Monitoring

TUM: Technical University of Munich

UniBW: University of the Federal Armed Forces

1 Introduction

The deterioration of the existing infrastructure is a fundamental concern for bridge operators. It has been estimated that a 12.4 [%] of the bridges in Germany are in poor conditions and there is a constant need of maintenance and rehabilitation of damaged structures (Krieger, 2019). Managing bridges is currently a challenging task due to the amount of gradually ageing/deteriorating bridges and the constrained maintenance budgets. In this context, information regarding damage detection and localization is crucial for the development of maintenance strategies that can prolong the service life of bridges and reduce the general maintenance costs.

The researchers at the Chair of Non-destructive Testing at the Technical University of Munich (TUM) have developed methods to analyse the performance of damage diagnosis methods based on vibration data of undamaged structures. The analysis requires the careful design of digital twins (i.e. finite element models) with the same dynamic behaviour as the real structure. This is achieved through model updating techniques, that improve finite element models based on the results obtained from field tests and continuous monitoring.

In this initial chapter, the main topics of the thesis are introduced. First of all, the importance of model updating is expressed in the context of structural health monitoring (SHM). Next the central case study of the thesis is briefly presented and finally the main objectives of the entire thesis are described.

1.1 SHM

SHM is a subclass of non-destructive testing that helps to understand the global status of a structure. Information about changes within the dynamic characteristics of structures is detected by sensors that remain permanently on the structure. The collected data from the sensors is related to the structural properties of the structure (e.g. weight, material, stress, strain and geometry). As a result, this data can be used to derive additional information about the capacity and condition of a structure (Neitzel et al., 2011).

SHM is becoming an increasingly popular topic of discussion in the bridge engineering community. Ongoing developments in sensor and data acquisition technologies have made it possible to install extensive monitoring systems on many structures. The hope is that by obtaining quantitative data it will be possible to develop “smart” structures, with monitoring systems able to supplement the largely subjective visual inspection practices, which are currently employed as the primary means of evaluating structural integrity and condition (Vardanega et al., 2016).

The increasing age of our existing infrastructure makes the cost of maintenance and repairs a growing concern. SHM may alleviate this by replacing scheduled maintenance with as-needed maintenance, saving the cost of unnecessary maintenance, on one hand, and preventing unscheduled maintenance, on the other hand. For new structures, the inclusion of structural health monitoring sensors and systems from the design stage is likely to greatly reduce the life-cycle cost. SHM systems could ensure increased safety and reliability while reducing maintenance costs (Giurgiutiu, 2014).

In Figure 1, it can be seen how a bridge is monitored. Sensors have been applied on the deck of the midspan of the bridge. These will continuously collect and evaluate vibration data according to the digital undamaged model. An interval of the features is set to represent that the structure is considered safe. If the features are outside of this interval, the structure is considered unsafe and a bridge operator has to evaluate it.

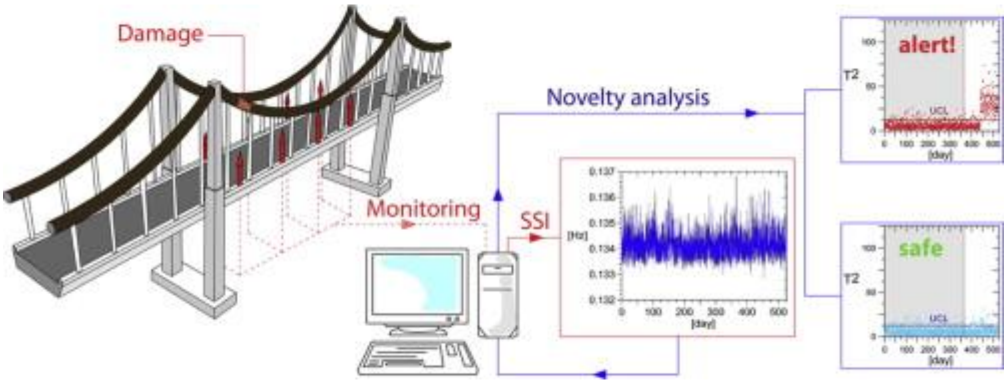


Figure 1. Structural Health Monitoring. (Yepes, 2020)

This type of damage diagnosis is called novelty detection and is based on the recognition of statistical pattern. Through this method it is possible to detect damage. According to the Rytter scheme, only the first stage of damage diagnosis is reached through this method. This scheme (Rytter, 1993) divides the depth of damage diagnosis into four different stages that vary in complexity:

- *Stage 1*: Detection
- *Stage 2*: Localization
- *Stage 3*: Quantification or assessment
- *Stage 4*: Prediction:

Apart from novelty detection, there are several other methods to diagnose damage: e.g. finite element model updating (FEMU) and parametric change detection (Mendler, 2020). This thesis focuses on the finite element model updating method. With this method, it is possible to reach the third stage of the Rytter scheme, damage quantification.

In FEMU, the information from data-driven features is used to update numerical models (Friswell & Mottorshead, 1996). The objective is to set up the equation of motion in modal coordinates, using the model-based mass, stiffness, and damping matrices, and the measured frequencies and mode shapes. If the model-based matrices do not reproduce the data-driven features, the system matrices or the underlying parameters are modified accordingly.

It is important to consider that it is not possible to obtain a numerical model with the exact same static and dynamic behaviour as the real structure through finite element modelling alone. There are many ways a model deviates from the real structure due to the large uncertainty level of infrastructure systems. Because of this, model updating techniques are necessary to reproduce the dynamic behaviour of the real structure in the model. Increasing the accuracy of the model depends not only on the skills of the designer but also on the quality of the information extracted from the real structure.

1.2 Case study

Unfortunately, there are not many prototypes to evaluate model updating methods as damaging real structures for research purposes is normally not possible. In this context, the development of benchmark models such as the bridge in University of the Federal Armed Forces (UniBw) in Munich is a unique opportunity to study the model updating methods. In Figure 2 is a picture of the real bridge.

Figure 2. UniBw bridge (Benndorf et al., 2016)

This thesis will provide the design of the ANSYS model of the UniBw bridge and it will be the main case study for the application of model updating methods. Aside from this thesis, this bridge has also been the focus of several other studies (Baumhauer, 2010 ; Benndorf et al., 2016 ; Janßen, 2022).

In order to understand the importance of the development of prototypes, a literature review of the studies focused on the UniBw bridge is provided. In the doctoral thesis (Baumhauer, 2010), a procedure for the damage assessment of the bridge according to uncertain parameters was made. This contributes to the prognosis of damage based on uncertainties, a major topic in SHM which will be later discussed in this thesis. The report (Benndorf et al., 2016) studies the feasibility of recording natural frequencies of structures using smartphones. The advances in sensor equipment directly affect the progress in SHM. One of the main limitations of SHM is the costly instrumentation and implementation difficulties. These type of studies open a new frontier in SHM in terms of engagement of citizens into the civil infrastructure sensing process and is being seen in a large number of studies (Ozer & Feng, 2020). Finally, the master thesis (Janßen, 2022) designed the Autodesk Revit model of the UniBw bridge. This can be seen in Figure 3. With the BIM software it is possible to create a digital twin of the real structure based on real measured data from sensors. The main idea is as this thesis, implement SHM methodologies to the bridge. The variability of the research studies conducted on the UniBw bridge, including this thesis, demonstrate the importance that benchmark studies have in the development of SHM techniques.

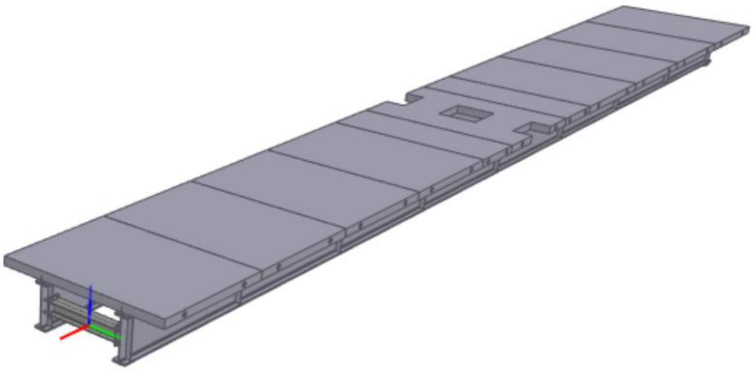


Figure 3. Autodesk Revit model of UniBw bridge (Janßen, 2022).

1.3 Objectives

There are three main objectives of this thesis:

- **Literature review:**

  The first goal is to review the existing literature on model updating methods and to select the most appropriate method for this study. For this purpose, it is necessary to develop the theoretical background on model updating, with a special emphasis on the parametrization of damage, as it defines the quality of the model updating.

- **Matlab implementation:**

  The second goal is to implement the selected method in MATLAB. A proof of concept study with a numerical model in MATLAB is provided in order to verify the validity of the method.

- **Application to UniBw bridge:**

  The third goal is to model the UniBw test bridge in ANSYS. A series of justified approximations have to be made in the design of the model. Apart from this, it is necessary to develop an interface between MATLAB and ANSYS in order to apply the selected model updating method to the model.

2 Literature review on model updating

In this chapter, a review on the existing literature on model updating is given. First of all, an overview of model updating is introduced. Then the most important steps in model updating are explained in detail. The first one is the selection of structural monitoring parameters which are employed to update the model. Secondly, the selection of features that are used to compare the dynamic response of the numerical model with the real data of the structure is explained. Finally, a number of model updating methods are introduced and one is selected for further demonstrations.

Overview on model updating

Model updating starts with the comparison between the initial uncalibrated model and the real structure. This comparison defines the quality of the numerical model and helps to identify differences and errors between the model and the real structure (Khodarapast, 2010). There are many reasons why these errors appear. Errors can appear due to incorrect assumptions of the model parameters such as material properties or section properties, due to the idealization or simplification of the structure, boundary conditions or mass distribution. Incorrect assumptions of loads, geometric shape and structural behaviour (linear/nonlinear) also lead to errors (Ereiz et al., 2022).

As a consequence of these errors, uncalibrated numerical models do not accurately reflect the actual state of real structures and need to be improved based on the results obtained from field tests and continuous monitoring. The basic idea of model updating is to use the recorded structural response to update some selected structural parameters of the numerical model (such as stiffness and internal forces) until an adequate agreement between numerical and experimental results is achieved (Friswell & Mottorshead, 1996).

In Figure 4, a scheme is presented that describes the general procedure of model updating. First of all, the real dynamic system (real structure) that will be monitored is selected. On the one hand, the dynamic response of the real structure is measured and the features of the real system are extracted (e.g. natural frequencies and mode shapes). On the other hand, the numerical model of the system is created based on selected parameters that represent the unknown structural properties of the model. These structural parameters affect the features of the model. The next step is to compare the features from the real measured data and numerical model. The structural parameters are updated until a good correlation between the measured data and the numerical output is achieved (Khodarapast, 2010).

Model updating is not only employed in model calibration but also in damage detection. Since damage is a change to the material and geometric properties of a given system which affects the system performance, the parameter distributions obtained as outcomes can provide useful information about the possible structural damage. In this effort, the FEMU is a very powerful tool for SHM (Mordini et al., 2007).

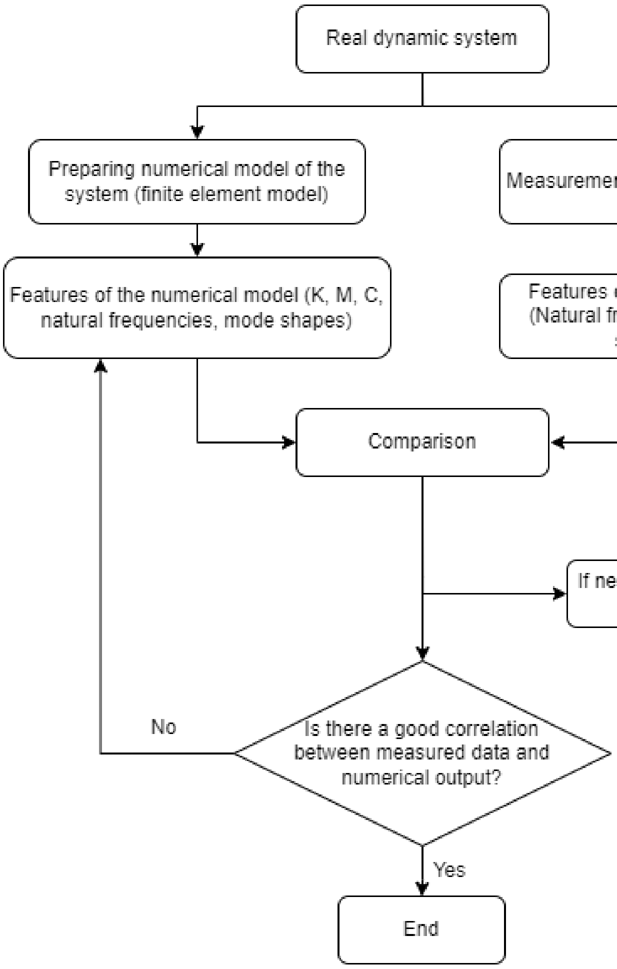


Figure 4. Scheme of model updating (Khodarapast, 2010).

As a result, the success of the procedure will strongly depend on the selection of design parameters that will be updated and the accuracy of the features extracted from the real structure (Friswell et al., 2001). Because of this, in the following sections these two fields will be explained in detail.

2.1 Parametrization

The objective of parameter updating techniques is to fit the parameters of a given initial analytical model in such a way that the model behaviour corresponds as closely as possible to the measured behaviour. As a result, the selection of the updating parameters will define the quality of the model. According to (Friswell et al., 2001) in order to meet the requirements for the accuracy and reliability of the numerical model and the performance of the model updating procedure, the parameterization procedure should meet the following criteria:

First of all, the outputs of the numerical model must be sensitive to selected updating parameters. It is necessary to choose the updating parameters that are most effective in reducing the differences between the numerical and experimental vibration data. Therefore, the chosen parameters have to directly affect the dynamic response of the model (Friswell et al., 2001). As a result, it is necessary to analyse the global and local behaviour of the structure in the selection process. The best way to perform model parametrization is sensitivity analysis, which results in suppressing the problem of inadequacy. Based on this, the parameters that do not affect the output results are excluded from the model updating process (Ereiz et al., 2022).

Secondly, the number of updating parameters must be limited in order to avoid convergence difficulties and ill-conditioned problems. It is possible to update numerous structural parameters, but then, special regularization techniques have to be applied. In general, the number of updating parameters should be kept to a minimum or advanced approaches have to be applied to reduce the number of parameters or improve the ill-conditioning of the updating problem (Ereiz et al., 2022).

Thirdly, the parameters have to be consistent with the real source. The model loses its physical meaning if the updated structural parameters have illogical values. Inconsistent models may be able to reproduce the test data, but they will naturally not be useful to predict the behaviour of the system (Mottershead, 2011).

Finally, the selected parameters should represent the unknown structural properties of the model. If model updating is applied for model calibration, the selected parameters would clarify the ambiguity of the model. Alternatively, if the purpose of model updating is to localize and quantify damage, the selected parameters have to represent the structural properties that are expected to change due to damage. There are many structural parameters that are generally considered in model updating. In Figure 5 is an overview of the updating parameters that were considered in this thesis.

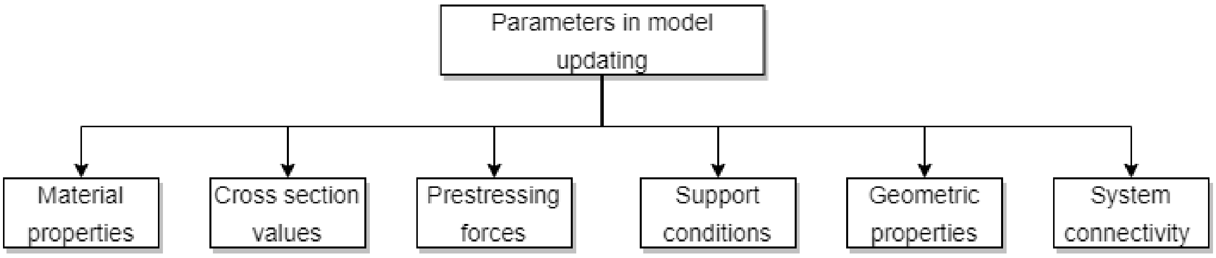


Figure 5. Structural parameters for model updating.

Physical properties such as Young's modulus and mass density are employed in model updating in many studies (Mordini et al., 2007; Mottershead, 2011; Teughels & Roeck, 2004). Geometric properties such as cross-sectional values or thicknesses are also very common. The material parameters, thicknesses and cross-sectional dimensions tend to be powerful updating parameters because they often apply throughout a finite element mesh affecting a large number of elements (Mottershead, 2011).

Prestressing values also tend to be good updating parameters in cable-supported bridges, especially for damage quantification. The presence of damage leads to a loss of axial stiffness. In a statically indeterminate structure, like a cable-stayed bridge, this causes a new internal force distribution with a lower load in the damaged elements and higher load in the others. FE model updating allows the evaluation of this reduction. Hence, the damage in a cable can be detected by calculating the axial force and comparing it with the design value or with a previous measurement. The parametrization of change in support conditions is similar to prestressing values, as changes support settlements lead to changes in the distribution of the axial stiffness of the structure. Support settlement are also very common in damage quantification (Teughels & Roeck, 2004). How the finite element model is affected by support settlements is explained in following sections.

System connectivity is generally an issue in finite element modelling. Joints and constraints tend to be less accurate and need the model updating as they are often very difficult to model. There are normally approximations that need to be made such as having to model rigid joints instead of flexible ones. System connectivity can be parametrised through physical and geometric changes or through geometric offset nodes. The first option can give good results. However, the model error is not localized in the joints but spread through the updated elements. It has been shown that adding flexibility to the structure in order to localize the stiffness in the joints can cause great problems for the updating algorithms. One approach, useful in many applications, is to make use of offset finite element nodes and to use the offset dimensions to correct the model . Lengthening or shortening an offset dimension usually corresponds to making the joint more flexible or to stiffening it, and in this way it is possible to reconcile the modification with engineering understanding of the structure (Friswell et al., 2001).

With some structural parameters such as the Young’s modulus, it is necessary to make further considerations. The strong correlation of the elastic modulus with temperature may lead to some inaccuracies in the final updated model. Several studies have shown how environmental impact such as a severe decrease in temperature can cause considerable changes on structural modal properties. These changes are even comparable to those induced by damage. Therefore, the environmental variability should be considered when using stiffness as model updating parameter (Zhou & Song, 2016).

It is important to consider that the resulting parameters from the model updating represent estimated values rather than true values. This can be due to the unavoidable random and systematic errors of the test data (Mottershead, 2011). Apart from this, it also has to be noted that in practice, the discrepancies between the experimental and analytical models are due to many parameters and uncertainties and not just the selected updating parameters.

In general, only sensitive parameters are selected, otherwise the updating process may be ill-conditioned since insufficient information is available to estimate the parameters accurately. As a result, the parameter selection requires a considerable physical insight into the target structure, and trial-and-error approaches are often used with different set of selected parameters for more complicated structures (Jaishi & Ren, 2007).

2.2 Features

The next step in model updating is to determine the features that will be used. Residuals or features represent the differences between the numerically and experimentally obtained data sets. These data sets include structural dynamic parameters, static data sets or combinations thereof (Ereiz et al., 2022). The most common features are natural frequencies and mode shapes. Changes to the structure lead to changes in the structural stiffness which in turn affect the structural dynamic parameters. However these changes are small and as a result, it is important to obtain high accuracy during the experimental tests in the field. In addition to natural frequencies and mode shapes, there are other structural dynamic properties that are frequently employed in model updating such as formulation of the residual function using frequency response function, modal flexibility or modal strain energy. This thesis will focus only on natural frequencies and mode shapes.

Aside from the structural dynamic properties, which are suitable for modelling complex structures, displacements and strains obtained from in-situ static tests have also been successfully used for FEMU. Nevertheless, the main problem with static measurements is the placement of the sensors and the errors they are subject to (size, position, orientation, thermal expansion, reading technique, or measurement accuracy) (Ereiz et al., 2022). In order to avoid errors, the best solution is to use these features in combination with the structural dynamic parameters.

In this article (Jaishi & Ren, 2005), the objective function that considers frequency residual only, mode shape (MAC) related function only, modal flexibility residual only, and their full combination are studied independently. The authors conclude that the objective function that considers all the combination of different residuals shows the best performance in model updating. However, this approach requires the proper definition of the weighted factor values in the residual function.

Natural frequencies

Natural frequencies are considered indispensable magnitudes in the model updating process (Jiménez-Alonso & Sáez, 2018). They are very sensitive to the stiffness of a structure and can be accurately obtained experimentally through a vibration test. However, a drawback to natural frequencies is that they are global quantities and are less sensitive to changes in local properties.

In model updating the features based on natural frequencies are simply obtained the difference between the natural frequencies of the real structure and the numerical model as shown in Equation (2.1):

$$r_i = \omega_i - \omega_i^{fea} \tag{2.1}$$

This represents a column vector where i refers to the mode number, r_i is the residual parameter or feature of mode i, ω_i is the natural frequency of the real structure, ω_i^{fea} is the natural frequency of the numerical model.

Mode shapes

Mode shapes are less sensitive as natural frequencies with respect to the material properties but provide spatial information of the structure, which makes them more sensitive to local damages and enables them to detect multiple damages or nonlinearities. Aside from this, mode shapes are less sensitive than natural frequencies to environmental effects such as temperature (Gorgin & Rahim, 2020). However, their experimental definition is more affected by noise and it is more difficult to identify accurately mode shapes in the real structure, also leading to inaccuracies in the model (Jiménez-Alonso & Sáez, 2018).

The implementation of mode shapes is not as forward as natural frequencies because additional considerations regarding the mode shapes scaling have to be made. To analyse the discrepancies between mode shapes, there are various residual functions that are applied in literature. Firstly, the direct difference between mode shapes:

$$r_{ij} = \phi_{ij} - \phi_{ij}^{fea} \tag{2.2}$$

This represents an $i\ x\ j$ matrix where i refers to the mode number and j the number of the sensor on the structure, r_{ij} is the residual parameter or feature of mode i at location j, ϕ_{ij} is the mode shape of the real structure, ω_i^{fea} is the mode shape of the numerical model.

Another common objective function to represent mode shapes is the modal assurance criterion (MAC) between the experimental and the analytical mode shapes. It is presented in the Equation (2.3):

$$MAC\ (r,q) = \frac{\left|\{\phi_A\}_r^T\{\phi_X\}_q\right|^2}{(\{\phi_A\}_r^T\{\phi_A\}_r)\{\phi_X\}_q^T\{\phi_X\}_q} \tag{2.3}$$

The MAC is the normalized scalar product of the two sets of vectors. It quantifies the accuracy of the identified mode shapes. It is a real quantity even with complex model shape data. It takes value from 0 (representing no consistent correspondence) to 1 (consistent correspondence) (Pastor et al., 2012).

2.3 Method selection

There are various finite model updating methods that through the comparison of the numerical and real feature calibrate the numerical model. They used for many applications, not only in bridges but also industrial-scale structures. In this thesis, it was important to choose the most optimal updating method for the case studies. Because of this, the most common updating methods are introduced. As a first classification, the updating methods are deterministic or non-deterministic.

The deterministic methods can be broadly classified into two categories, namely the direct methods, the iterative methods. Direct model updating methods are the oldest methods used to update numerical models. They are used to directly update the structure FEM by changing the structural stiffness matrix and the mass matrix. They substitute the numerical system matrices and the experimentally obtained frequencies and mode shapes to solve for the residual vector without iteration.

The direct methods include the direct solution and the optimal matrix update. The modal force residual (r_i) in the direct solution can be obtained from the equations of a finite element model of a vibrating structure (Carvalho et al., 2006). It is shown in the Equation (2.5) and could be used to detect and localize damages.

$$r_i = (-\omega_i^2\mathbf{M} + i\omega_i\mathbf{D} + \mathbf{K})\ \boldsymbol{\phi}_i \tag{2.5}$$

where ω_i is the natural frequency of mode i, $\boldsymbol{\phi}_i$ is the mode shape of mode i, $\mathbf{M}$ is the mass matrix, $\mathbf{D}$ is the damping matrix and $\mathbf{K}$ is the stiffnes matrix.

The optimal matrix updates the entries es of the mass, damping and stiffness matrices in order to minimize the residual vector. The optimization problem is generally formulated as a Lagrange multiplier or penalty-based optimization as shown in Equation (2.6).

$$\min_{\Delta \mathrm{M},\Delta \mathrm{D},\Delta \mathrm{K}} r(\Delta\mathbf{M},\Delta\mathbf{D},\Delta\mathbf{K}) + \lambda R(\Delta\mathbf{M},\Delta\mathbf{D},\Delta\mathbf{K}) \tag{2.6}$$

where r is the objective function, λ is the Lagrange multiplier and R is the constraint function. In comparison to the direct solution, the optimal matrix update diminishes the data-driven smearing effect due to the constraint function.

Without using iterative methods, direct methods can reproduce accurate experimental data and are computationally efficient. The main disadvantage of direct methods is that the updating process is performed without involving the physical knowledge of the problem. This drawback caused the later appearance of indirect methods, where the model updating arises from the changes applied on some well-defined structural physical parameters selected by the users. In this case, the modified parameters are not linearly related to the modal parameters so the adjustment process requires the use of iterative techniques (Mottershead, 2011)

In the iterative methods, an iterative process based on sensitivity analysis is required in order to minimise the residual function. This is achieved through the careful selection of updating parameters that represent the uncertainties of the model. These parameters will iteratively change until the residual function is a minimum. However, issues of convergence and ill-conditioning of the matrices may appear (Khodarapast, 2010). In the Equation (2.7) the residual function r of the sensitivity-based model updating is shown for the case where the natural frequencies (ω) are used as features.

$$r_i = f_i(\theta) = \omega_i - \omega_i^{fea} \tag{2.7}$$

where θ is the selected structural updating parameters. A drawback of sensitivity-based model updating is that it does not consider the variability of the features, bias to the collected data is easily introduced due to measurement noise, the use of sensors that affect the measurement or signal processing might. It is possible to decrease the variability by increased information (Khodarapast, 2010) but in some cases, it may be necessary to consider these uncertainties during the model updating. For these cases, the non-deterministic or stochastic model updating techniques are introduced. The most common non-deterministic methods are the Bayesian model updating, the interval methods and fuzzy updating. Firstly, the Bayesian model updating describes variabilities of the model parameters through probability functions. The formulation is based on the Bayes' theorem which is showed below (Mares et al., 2006).

$$p(\theta|r) = \frac{p(r|\theta)\, p(\theta)}{p(r)} \tag{2.8}$$

The terms of the Bayes' theorem can be seen in Figure 6, where p(θ) is the prior distribution before updating, p(θ|r) is the posterior distribution after evaluating the data-driven residuals, p(r| θ) is the likelihood function that describes the probability of observing a residual r for a given parameter and p(r) is the evidence, a normalization constant. (Mares et al., 2006)

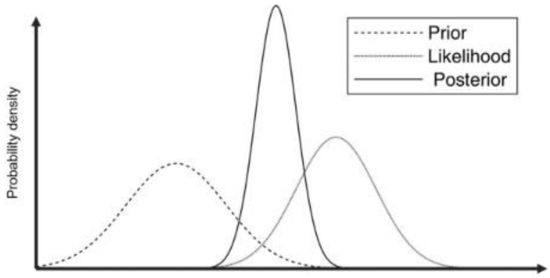


Figure 6. Bayes updating from its prior to its posterior distribution (Etz, 2017).

Bayesian model updating is commonly used to consider the variability of the model parameters. In order to parametrise uncertainty, however, the interval and fuzzy updating techniques are more common (Fang et al., 2013). In interval model updating, the uncertain inputs are defined through intervals as shown in the Equation (2.8).

$$\widetilde{\boldsymbol{X}_\iota} \,\epsilon\, \left(\left[\underline{x_1},\, \overline{x_1}\right] \cup \ldots \cup \left[\underline{x_p},\, \overline{x_p}\right] \qquad i = 1 \cdot p \tag{2.8}$$

Where p is the number of input parameters, then a function $y = f(x)$ that maps inputs to outputs is obtained. The goal is to solve the equation by considering the intervals as follows (Fang, Zhang, & Ren, 2013) .

$$\overline{y_i} = \max\big(y_i(x)\big) \qquad \underline{y_i} = \min\big(y_i(x)\big) \tag{2.9}$$

The last method is the fuzzy model updating, where membership functions are used to describe uncertainties. The membership function is depicted in the Figure 7, and is obtained through interval method instead of the probability function as the Bayesian model updating. In the worst cases of uncertainties, it is the most useful technique (Boulkaibet et al., 2017).

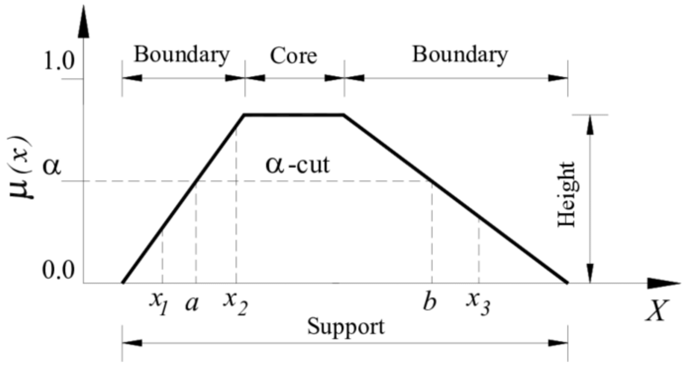


Figure 7- Membership function of fuzzy logic (Kishk & Assem, 2000).

For this thesis, it was necessary to choose one for its implementation to the case studies. As a first approach, it was decided not to consider direct methods as they do not consider the physical meaning of updating the mass and stiffness matrices. In sensitivity-based model updating and non-deterministic methods, however, it is possible to select structural parameters that represent uncertainties in order to update the model. The main difference between these methods relies on the consideration of the variability and uncertainties of the structural parameters. Sensitivity-based methods do not consider these, which leads to inaccuracies in the numerical model.

However, they are more straight-forward to apply than non-deterministic methods. The latter require an in-depth background on higher statistics where sensitivity-based method are closer to the structural dynamics domain.

As a result, it was decided to implement the sensitivity-based model updating method in the this thesis. Aside from this, at the Chair of Non-destructive testing at TUM, sensitivity-based statistical tests are used for damage diagnosis, which has a similar background to the model updating technique. The sensitivity vector could therefore first be used for model updating and then for the damage diagnosis based on different techniques.

3 Sensitivity-based model updating

Sensitivity-based model updating is the selected method in this thesis. Therefore, a complete theoretical background on the technique is provided. First of all, the employed data-driven residuals are presented. Then how the method is applied is explained, with a step by step procedure. Finally, a concept study demonstrating the theory is introduced.

3.1 Feature selection

As explained previously, the selection of the data-driven residuals is a defining step in model updating. There are several different types of features that can be used for model updating techniques. Some of them have been presented in previous sections. In this thesis, it has been decided to solely use the natural frequencies as data-driven features due to the simplicity of its implementation.

By selecting uniquely one feature, it is not necessary to introduce the weigh the influence of the residuals in the updating parameters. Apart from this, the issue of shape scaling of the mode shapes is avoided. As the final intention of this thesis is to demonstrate the sensitivity-based model updating approach, it was deemed unnecessary to select different residuals. The residual function based on natural frequencies has been used in many studies and is even considered an indispensable magnitude in model updating.

3.2 Method

The easiest way to explain the sensitivity-based model updating method is by setting a simple example with only one parameter to be updated. As stated in the previous section, the residual is a function of the structural updating parameters. This function is represented for this simple case in Figure 8, f(x) represents the residual and x is the parameter to be updated.

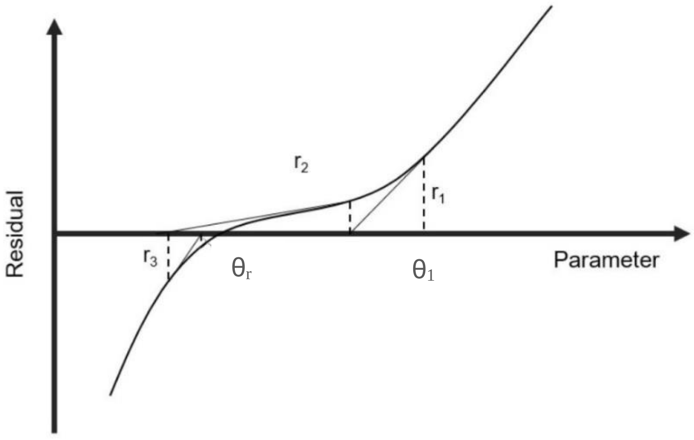


Figure 8. Newton-Raphson method (Mendler, 2021).

The objective is to minimise the value of the residual, it has to be as close to 0 as possible. This is achieved through the Newton Raphson iteration. This method begins with an initial estimate of the updating parameter for which the residual is not 0 ($\theta_1 \neq \theta_r$). Then, the first derivative of the function, also called sensitivity, at the point θ_1 is determined.

$$m = \frac{\partial}{\partial \theta_1} r(\theta) \tag{3.1}$$

The tangent of the function is obtained through the first-order Taylor expansion. The interception of the tangent with the horizontal axis ($\Delta r = 0$) is easily solved as both the value of the residual and the first derivative is known, the $\Delta\theta$ is solved.

$$\Delta r \approx r + m\,\Delta\theta \qquad \Delta\theta \approx \frac{-r}{m} \tag{3.2}$$

Lastly, the value of the residual is re-estimated with the updated $\Delta\theta$. The whole process is repeated until the error function reaches a minimum (Garret, 2015).

$$\theta_2 = \theta_1 + \Delta\theta \qquad r_2 = f(\theta_2) \tag{3.3}$$

This example was just for one parameter and one feature. Normally the parameters to be updated and features are more than one. As a result, sensitivity vectors and matrices have to be set up. For the case studies in this thesis, for example, it was necessary to compute a sensitivity matrix. This matrix has n rows and m columns, where the number of rows n is the same as the number of features and the number of columns m is the number of parameters. Each entry of the sensitivity matrix J_{ij} will contain the first-order derivative of the residual feature i with respect to the model parameter to be updated j. The sensitivity matrix contains the first derivatives and blows up to a Jacobian matrix. Below is the example of a 2 x 2 Jacobian matrix shown, this belongs to the case where there are two features to analyse and two parameters to update.

$$\boldsymbol{J} = \begin{bmatrix} \frac{\partial}{\partial \theta_1} r_1(\theta) & \frac{\partial}{\partial \theta_2} r_1(\theta) \\ \frac{\partial}{\partial \theta_1} r_2(\theta) & \frac{\partial}{\partial \theta_2} r_2(\theta) \end{bmatrix} \tag{3.4}$$

For some residuals, it is not possible or very difficult to obtain analytical derivatives for the computation of the sensitivity matrix. Other methods have to be used such as the finite difference method. It consists in evaluating the feature r before and after a small perturbation $\Delta\theta_j$ is applied to the structural parameter: $\theta_j^* = \theta_j + \Delta\theta_j$. The corresponding sensitivity entry is calculated as in the Equation (3.5):

$$J_{ij} = \frac{\Delta r_i}{\Delta\theta_j} = \frac{r_i^* - r_i}{\theta_j^* - \theta_j} \tag{3.5}$$

where i and j are the row and the column of the sensitivity matrix. In each column j, is the result of the small perturbation $\Delta\theta_j$ for each feature i. Consequently, the accuracy of the obtained derivative depends on the step size $\Delta\theta_j$. It is important to note that for big step sizes, it is not possible to represent the tangent through the finite difference method.

With the computation of the sensitivity entries, the complete Jacobian matrix can be computed. Its purpose is to map changes in features onto changes in structural parameters and to create the link between data-driven domain and the model-based domain. The Jacobian matrix makes it possible to apply the complete sensitivity-based model updating procedure to a complex system with i features and j parameters.

$$\begin{bmatrix} r_1 \\ r_2 \end{bmatrix} = \boldsymbol{J} \begin{bmatrix} \theta_1 \\ \theta_2 \end{bmatrix} \tag{3.6}$$

The final objective of sensitivity-based model updating is to find the updated structural parameter θ vector that corresponds to a minimum deviation of the residual function. This is achieved through iterative methods such as the Newton-Raphson method. This method is applied in the following three steps:

Firstly, the sensitivity matrix is computed by applying the finite difference method as explained previously. Secondly, with the inverse of the sensitivity matrix, the parameter vector is updated as shown below.

$$\begin{bmatrix} \Delta\theta_1 \\ \Delta\theta_2 \end{bmatrix} = \boldsymbol{J}^{-\boldsymbol{1}} \begin{bmatrix} \Delta r_1 \\ \Delta r_2 \end{bmatrix} \tag{3.7}$$

Thirdly the delta residual vector Δr is re-evaluated. If it is equal to 0, the model and the real structure have the same features, and the model has been updated. Alternatively, it is necessary to return to the first step with the new updated parameter θ until the residual vector reaches a minimum.

Two types of iterative methods have been considered in this thesis. On the one hand, the above explained, Newton Raphson method where the sensitivity matrix is calculated in each iteration and on the other, the modified Newton-Raphson where the sensitivity matrix is calculated once at the beginning. Computing the sensitivity matrix in each iteration requires several executions (one per each considered parameter) of the modelling programm. As a result, when modelling complex structures, the modified Newton-Raphson method is preferred as it is generally more time efficient in spite of the lower number of iterations in the Newton-Raphson method. Below is a figure that compares both iteration methods according to the number of iterations needed to obtain the same result.

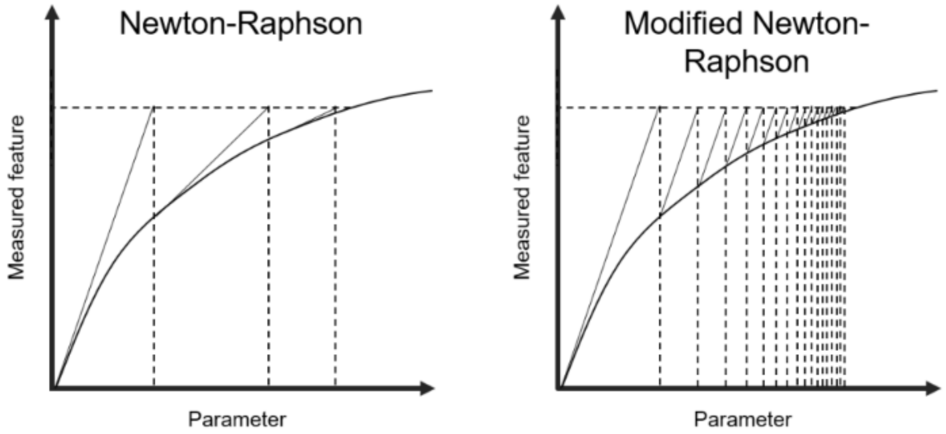


Figure 9. Newton Raphson vs modified Newton-Raphson (Mendler, 2021).

3.3 Step-by-step procedure

In this section, the complete step-by-step procedure of sensitivity-based model updating is described. As the main model of this thesis is the UniBw bridge, which can be considered a complex structure due to its size and its computation time, it has been decided to apply the modified Newton-Raphson method.

Naturally, the first step is to select the updating parameters and the features to be compared. As previously explained, this step defines the final quality of the model. Therefore, it is necessary to make the proper considerations and analysis as explained in the previous sections.

Then an initial assumption of the updating parameters has to be introduced. This assumption has to be consistent with the real structure as it may decrease the number of iterations to obtain the final updated model. Apart from this, a tolerance has to be set to end the iteration. With this initial data, it is possible to assemble the Jacobian matrix of the model. In the modified Newton-Raphson method, this is done before the iteration. The entries of the Jacobian matrix are obtained through the finite difference method, so a small perturbation to each θ parameter has to be applied to the structure. This means that the model has to be executed the same number of times as number of θ parameters considered. After obtaining the Jacobian matrix, it is possible to perform its validation. The validation consists in comparing the residual function with the tangent obtained through finite difference method.

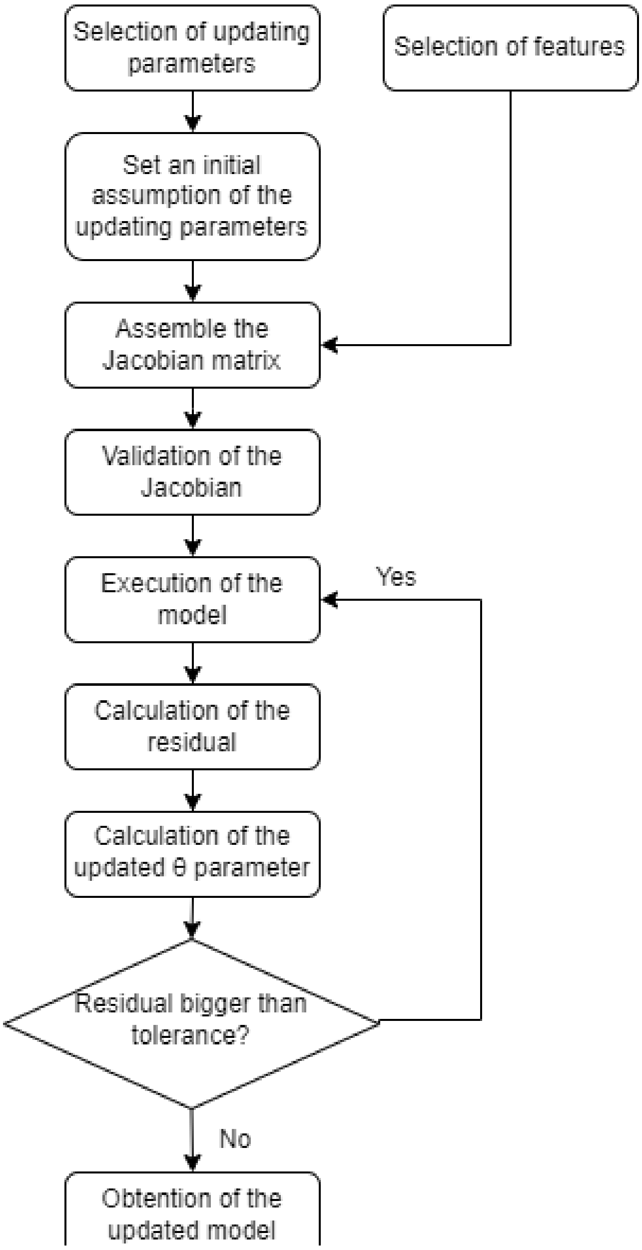


Figure 10. Step-by-step procedure of sensitivity-based model updating.

Next, the iterative process starts. In each iteration, the first step is to obtain the modal data of the model with the introduced θ parameter and to calculate the corresponding residual vector. With the inverse of the Jacobian matrix, $\Delta\theta$ is obtained and the θ can be updated. Finally, the residual vector is evaluated. In the case of this thesis, this was done by calculating the Euclidean norm of the residual vector. If this value is smaller than the tolerance, the model has been updated with the resulting θ parameter. On the contrary, if the norm of the residual vector is larger than the tolerance, it is necessary to start the iterative process again with the last obtained θ parameter. The complete process is represented in Figure 10.

3.4 Proof of concept study

In this section, the sensitivity-based model updating method is applied to the following model in order to proof the presented theory. It is a six floor building that is modelled as a mass-and-spring system with six degrees of freedom (6-DOF) as shown in Figure 11.

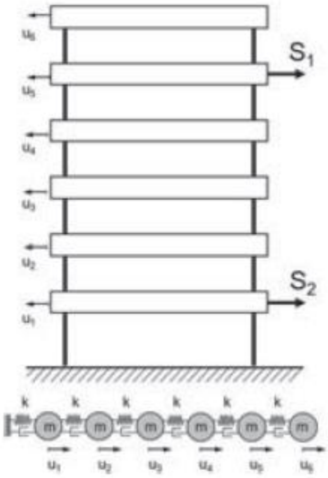


Figure 11. 6-DOF mass and spring system (Mendler et al., 2022).

The modal frequencies of the damaged and undamaged structure were numerically calculated. In the healthy state, the first six frequencies are evaluated one-hundred times. In the damaged state, the modal frequencies are available for six modes. Each mode was evaluated one hundred times (as the healthy state) and there are two different damage scenarios represented. The evaluated features for the comparison between the data-driven and numerical domain are the six natural frequencies. Due to the amount of data provided and to increase the accuracy, the selected features will each be the average of the one hundred data segments.

In the first damage scenario, the spring stiffness of the first floor is reduced by a 25 [%] and in the second damage scenario, the second spring stiffness will be reduced by the same amount. Because of this, it has been decided that the structural monitoring parameter $\boldsymbol{\theta}$ contains the stiffness of each spring (shown below). In the healthy state the stiffness of each spring is 1000 [N/m].

$$\boldsymbol{\theta} = \boldsymbol{k} = [k_1 \quad k_2 \quad k_3 \quad k_4 \quad k_5 \quad k_6\,]^T \tag{3.9}$$

The objective of this concept study, is to demonstrate that it is possible to quantify damage through model updating methods. Therefore, the initial assumption of the monitoring parameter has to be the healthy state, that is, the spring stiffness of the 6 masses are 1000 [N/m]. This way, it is possible to directly compare the healthy state with the damaged state. Both damage scenarios will be presented in this thesis. It is important to note that as this numerical model has a very low computation time, the most suitable iterative method is the Newton-Raphson.

First damage scenario

In this first damage scenario, the first spring stiffness decreases by a 25 [%]. In Figure 12, it can be seen how the features are affected by this damage scenario. In black, are the data segments of each feature in the healthy state and in red, the damaged state. The modal frequencies decrease in the damaged state, it is worth mentioning this result is consistent with real structures. If the stiffness of a spring decreases, the natural frequencies have to decrease as well.

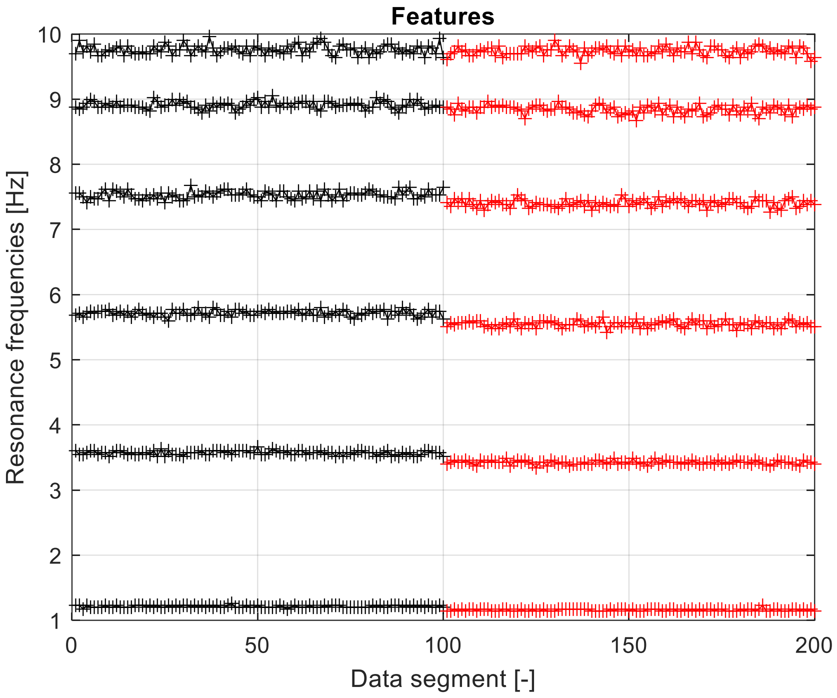


Figure 12. Frequencies of the undamaged and first damaged system (6-DOF).

Once the features and the initial monitoring parameter are known, the next step is to start the Newton Raphson iterative method. For each iteration, the Jacobian matrix is assembled through the finite difference method as previously explained. This model has a very low computation time and because of this, it is possible to estimate the updated parameter with a low tolerance. In this case a tolerance of 10^{-12}[-]. After 8 iterations, the following results were obtained as seen in Figure 13.

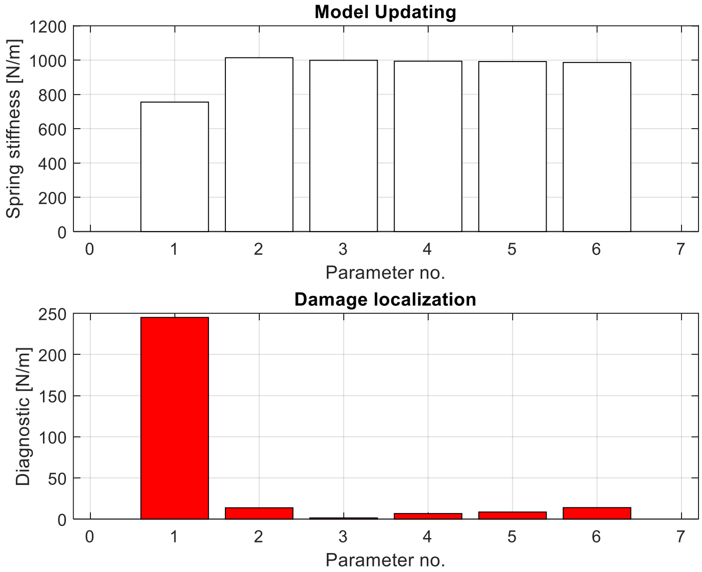


Figure 13. Changes structural monitoring parameter, first damage scenario (6 DOF).

As expected, there is a significant decrease in the stiffness of the first spring/floor. Specifically, it has decreased from 1000 N/m to 755 N/m, which corresponds to approximately 25[%] of the initial spring stiffness in the healthy scenario. This coincides with the numerical results, meaning that the model updating was successful. It was possible to identify which spring/floor was failing and how severe the damage is, that is to localize and quantify damage.

Second damage scenario

The same analysis was done to the second damage scenario. In this case, the second spring stiffness was decreased by a 25 [%]. The final result of the sensitivity-based model updating can be seen in the next figure. As in the previous case, it was also possible to localize the damage in the correct spring/floor and to quantify it. The stiffness of the second spring was reduced by a 25 [%] as the Figure 14 shows.

With this numerical example, it was possible to demonstrate the quantification of damage for two damage scenarios simply by using sensitivity-based numerical model updating. In the next sections of the thesis, more complex models are evaluated and the same verifications will be made.

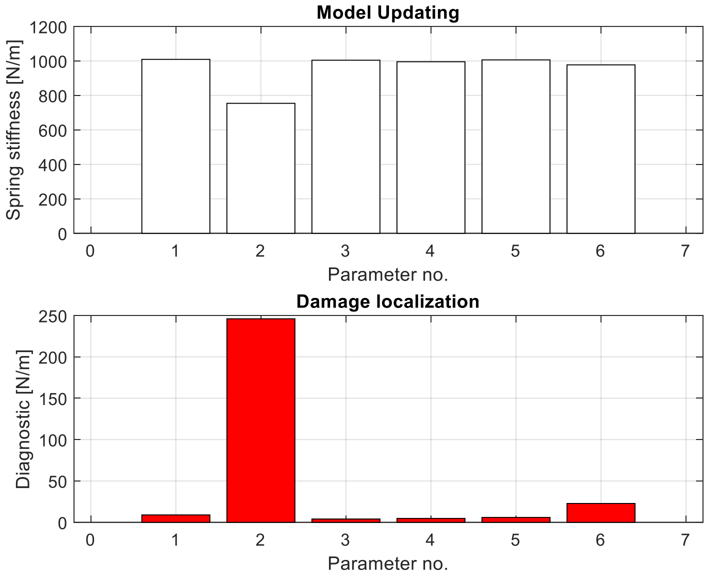


Figure 14. Changes structural monitoring parameters, second damage scenario (6 DOF)

4 Finite element modelling in ANSYS

In engineering design, the most reliable and widespread technique for numerical modelling is the finite element method. It consists of dividing a complicated structure into discrete areas or volumes known as ‘elements’ with simple and standard geometrical shapes such as small tetrahedral or hexahedral volumes. The standard formulation for the finite element method is the displacement method. This method consists in the the idealization of general structure as an assemblage of beams and truss elements interconnected at nodes whose modal and static solution can be obtained through the static and dynamic balance equations (Bathe, 2014).

Finite element analysis allows to analyse the dynamic behaviour of numerical models under different types of loadings or support settlements. With accurate models that have been updated, it is possible to detect any deficiency from the early stages of the design process or to predict damage in structures that are being used at the moment of analysis (Khodarapast, 2010). The more accurate the initial model, the more accurate the model updating process and the more accurate the damage information. As a result, it is crucial to model the finite element modelling as accurate as possible.

The chosen software for the modelling of the bridge is ANSYS Parametric Design Language (APDL). This software is suitable for model updating purposes as it allows the parametrization of the structural properties of the model. Furthermore, it is possible to interface ANSYS APDL with MATLAB. Through this interface, iterative methods can be applied to the finite element model.

In this thesis, the sensitivity-based model updating is applied to two different models of varying complexity. The first model is a simple bi-supported beam with an intermediate support and the second study consists of the UniBw bridge model, the main case study of the thesis. The first model is only introduced to verify the interface between ANSYS and MATLAB and the code on sensitivity-based model updating. The case study of the UniBw bridge, however, has to be designed more carefully. This chapter introduces some considerations that were made for the design of the finite element models. Firstly, the main approximations that are generally made in finite element modelling are described. Then how to model changes in support conditions is explained. This section is important for the development of the case studies. Finally it is described in a step-by-step procedure how the interface between ANSYS and MATLAB is computed.

4.1 Assumptions in FE modelling

Linearity is one of the main approximations that are generally employed in finite element modelling. In a linear static analysis a linear relation holds between applied forces and displacements as shown in Figure 15. In practice, this is applicable to structural problems where stresses remain in the linear elastic range of the used material. In a linear static analysis the model's stiffness matrix is constant, and the solving process is relatively short compared to a nonlinear analysis on the same model. Therefore, for a first estimate, the linear static analysis is often used prior to performing a full nonlinear analysis (Bathe, 2014).

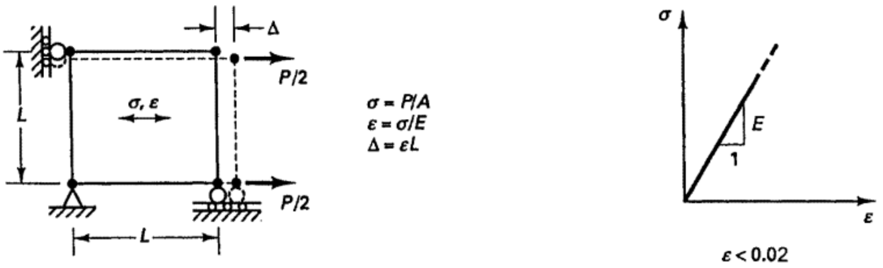


Figure 15: Linear elastic FEM (Bathe, 2014)

A nonlinear analysis is an analysis where a nonlinear relation holds between applied forces and displacements. Nonlinear effects can originate from geometrical nonlinearity's (i.e. large deformations), material nonlinearity's (i.e. elasto-plastic material), and contact. These effects result in a stiffness matrix which is not constant during the load application. This is opposed to the linear static analysis, where the stiffness matrix remained constant. As a result, a different solving strategy is required for the nonlinear analysis and therefore a different solver (Femto Engineering , 2017).

Geometrical nonlinearities

Geometric nonlinearities consider the changes of the geometry as the structure deforms. There are different degrees of nonlinearities. The first one considers large displacements, large rotations but small strain whereas the second considers large displacements, rotations and large strain. In Figure 16, both degrees o geometric nonlinearities can be seen and compared.

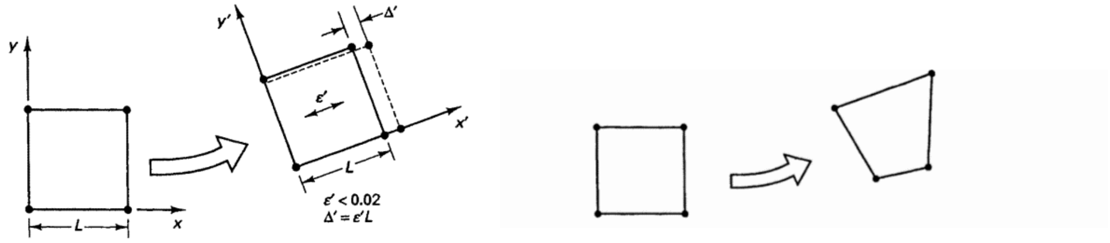


Figure 16. Geom. non-linearities, small strain (left), large strain (right) (Bathe, 2014)

The main difference between the small strain and large strain consolidation or any deformation analysis is that in small strain approach there is no update in geometry with respect to time which means the shear stress is constant with increasing strains. On the other hand, in large-strain analysis, the geometry is updated with respect to time which means that the shear stresses are decreasing with increasing strains (or displacement). Generally, the small strain approach will result in larger deformation as compared to the large-strain approach. However, large-strain approach is more realistic (Bathe, 2014).

Material nonlinearities

Material nonlinearities involves the nonlinear behaviour of a material- nonlinear displacement and stress–strain response. For the case studies of this thesis, steel has been one of the considered materials. As it can be seen in Figure 17, it has a nonlinear behaviour. If it was decided to model the steel with a linear behaviour, any deformation that occurs as a result of a stress higher than the Yield strength, would be inaccurately represented.

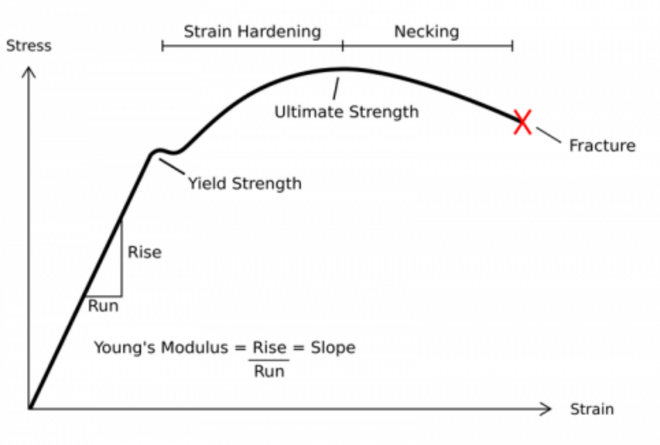


Figure 17. Stress strain relation of steel (Autodesk , 2018).

In actual finite element analysis, it is necessary to decide whether a problem falls into one or other category of nonlinearity as it dictates which formulation will be used to describe the physical situation. It is true that the use of the most general large strain formulation "will always be correct"; however, the use of a more restrictive formulation may be computationally more effective and may also provide more insight into the response prediction (Bathe, 2014). As a result, it has been decided to consider that the finite element models introduced in this thesis have a linear behaviour.

Apart from assuming the linear behaviour of the FE model, there are several other approximations or assumptions that are commonly introduced to FE models. For example, the variability of material properties is in some cases not properly considered. Aside from this, there are also usually approximations introduced in the modelling of system connectivity as it is usually difficult to model (Friswell et al., 2001). Nevertheless, it is important to always assess the error introduced to the model due to approximations.

4.2 Modelling changes in support conditions

In some case studies of this thesis, damage will be represented as the change in boundary conditions. Therefore, it is important to explain how the imposition of displacement boundary conditions alters the initial stress state in structures and the resulting modal parameters and how it can be implemented in finite element software.

First of all, a structural analysis of the finite element model has to be executed to obtain the global stress vector. Next, the force vector is transformed onto element level in order to evaluate the geometric stiffness and then back-transformed to element level. Ultimately, the modal parameters are solved through a dynamic analysis which considers the new terms of the geometric stiffness (Mendler et al., 2022). To determine the global force vector, the equilibrium equations are derived, neglecting damping, as follows:

$$\begin{bmatrix} M_{aa} & M_{ab} \\ M_{ba} & M_{bb} \end{bmatrix} \begin{bmatrix} \ddot{U}_a \\ \ddot{U}_b \end{bmatrix} + \begin{bmatrix} K_{aa} & K_{ab} \\ K_{ba} & K_{bb} \end{bmatrix} \begin{bmatrix} U_a \\ U_b \end{bmatrix} = \begin{bmatrix} R_a \\ R_b \end{bmatrix} \tag{4.1}$$

where U_a are the unknown displacements and U_b are the known/fixed displacements. Considering that accelerations are zero at the fixed supports, for a static support displacement, the U_a can be solved as:

$$M_{aa}\ddot{U}_a + K_{aa}U_{aa} = R_a - K_{ab}U_b \tag{4.2}$$

In other words, the permanent support displacement is modelled through an equivalent force to static displacements at the unknown displacements. With this, we demonstrate how the change in the support settlements affect the final solution of the structure. This is an important aspect of the thesis because for the final case studies, the change in support settlement will be representing damage (Bathe, 2014).

For this thesis, the code presented in the Appendix A had to be included in the ANSYS command window, in order to correctly model the change in support settlements. A static analysis is performed prior the modal analysis. In this static analysis firstly the nodes that will be displaced are selected and then the support settlements are applied with the command D. As seen in this section, the stress state of the model changes with support settlements, therefore it is necessary to save its new distribution for the modal analysis. This is implemented in ANSYS through the PSTRES, ON command as seen in the appendix. This is how support settlements are implemented in ANSYS.

4.3 Interface between ANSYS and MATLAB

In this section, how MATLAB and ANSYS are interfaced is explained. Both software are necessary for the application of model updating techniques. On the one hand, with ANSYS, models of real structures are designed and built. On the other hand, MATLAB is needed in order to apply model updating techniques to these models and to obtain the results of model calibration and damage diagnosis.

As explained previously, sensitivity-based model updating consists in changing selected parameters iteratively until the residual vector between the model and real structure is zero. This means, that in order to apply this method, it is necessary to run the ANSYS model from MATLAB iteratively for different parameter vectors and the corresponding modal solution has to be saved in MATLAB for its evaluation. In Figure 18, how both software have to be combined in each iteration of the sensitivity-based model updating is shown.

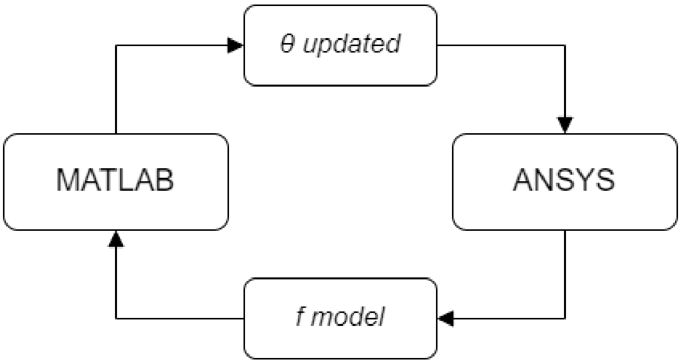


Figure 18. Interface between ANSYS and MATLAB.

First of all, a θ parameter is introduced in MATLAB. ANSYS builds the model and obtains the modal solution, i.e. natural frequencies and mode shapes. This solution is saved in MATLAB so that the corresponding residual vector can be calculated. If the residual vector is non-zero, the θ parameter has to be updated and a new iteration takes place. On the contrary, if the residual vector reaches a minimum, the model is updated with the last introduced θ parameter.

The interface between the two software requires a toolbox that has to be downloaded in MATLAB and the 2021 R1 license of ANSYS. ANSYS has to be run in the ANSYS as a server (aas) mode. This way, it is possible to input and issue ANSYS commands directly from MATLAB. Because of this, it was necessary to build and analyse the ANSYS model through text files. These text files contain all the commands that are used in the building and analysis of the model. Apart from this, there are also commands that can create result data files which MATLAB can open. This is how the natural frequencies and mode shapes of the model can be used in MATLAB. The step-by-step procedure of the MATLAB code (see appendix B) that obtains the natural frequencies and mode shapes from the ANSYS model is shown in Fig. 19:

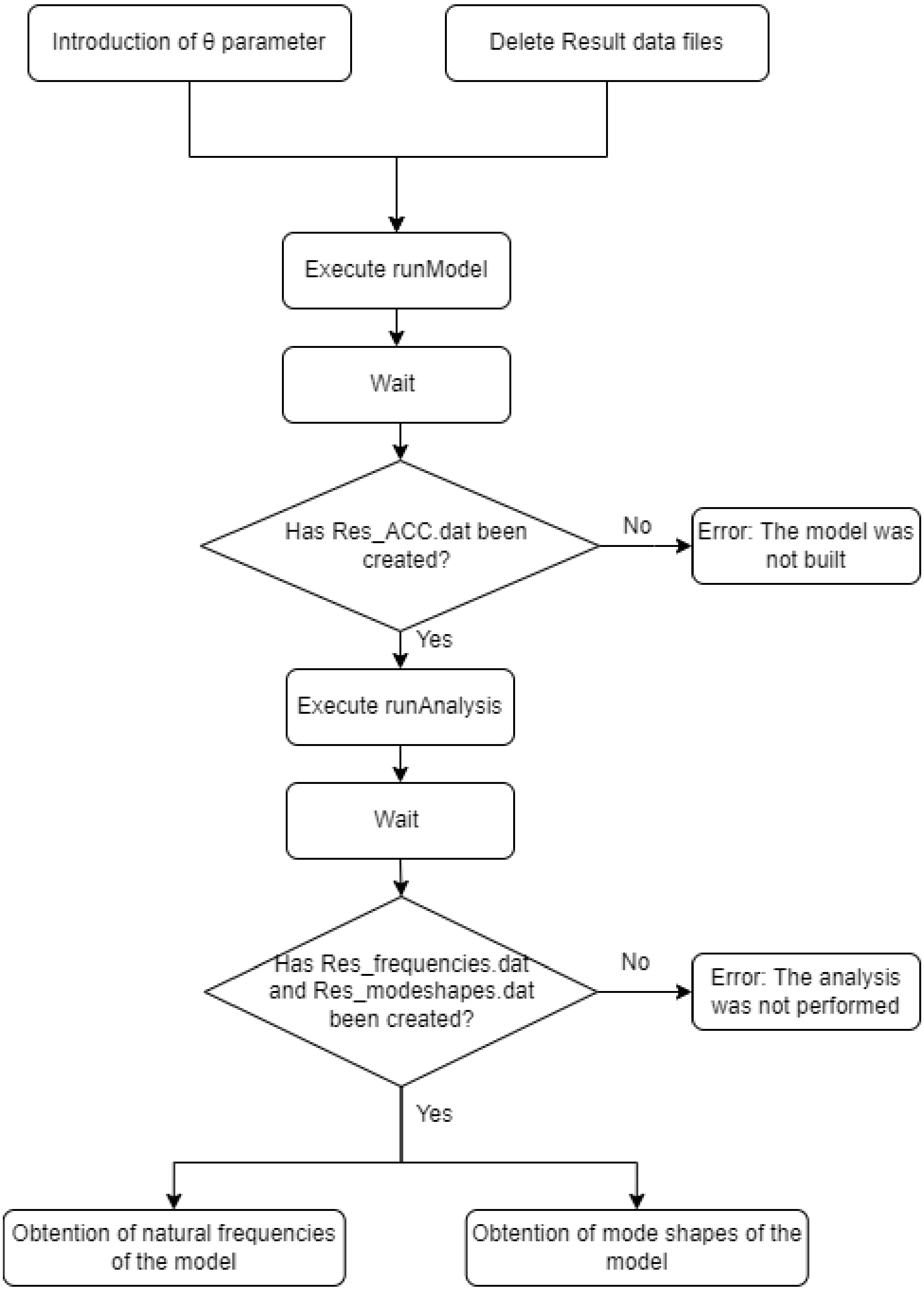


Figure 19. Scheme of the interface between Matlab and Ansys.

As it can be seen, firstly the values of the θ parameter have to be introduced by the user. Apart from this, all the result data files are deleted. Naturally, the computation time of the model in ANSYS has to be considered in the MATLAB code. If not, the model will not be built and it will not be possible to obtain the natural frequencies. Because of this, it is necessary to issue a while loop until the result data file is created. For the text file that builds the finite element model this data is going to be Res_ACC.dat which contains the locations of the sensors in the model. After this, the runAnalysis text file (see appendix A) is input. This text file as previously explained, saves the natural frequencies and mode shapes of the model. This is done through the Res_frequencies.dat and Res_modeshapes.dat data files. These results are then loaded in MATLAB and can be used to apply the model updating techniques.

5 Case studies

In this section, the sensitivity-based model updating method is applied to two different case studies, each varying in complexity. First of all, the HSS beam is introduced. For this case, the calibration of the Young's modulus is performed and then a support displacement is updated. With this case study, it is possible to verify the interface between ANSYS and MATLAB and the correct application of the sensitivity-based model updating.

Aside from this, a second case study is introduced with a more detailed finite element model, the UniBw model. One of the main tasks of this thesis was to design the ANSYS model of this bridge. In this section, a description of the bridge is firstly provided. Aside from this, the ANSYS model is presented, detailing the main approximations that were made during the design process. Finally, sensitivity-based model updating is applied to the model. Firstly for its calibration, and then to diagnose damage.

5.1 HSS beam

In this section, the HSS beam is presented. The purpose of this case study is to verify that the interface between ANSYS and MATLAB together with the code on sensitivity-based model updating were correctly applied. Due to the low computation time of this model, it was possible to edit and correct errors until the code could be correctly applied to the next case study, the UniBw bridge.

5.1.1 Description of the HSS beam model

First of all, a case study of a simple HSS beam has been modelled by ANSYS APDL. It can be seen in Figure 20 below. The particularity of this beam is that it is composed of two different materials which have a different Young's modulus. In blue is the material 1 and in yellow the material 2. Apart from this, the beam is supported on its ends and also in the middle.

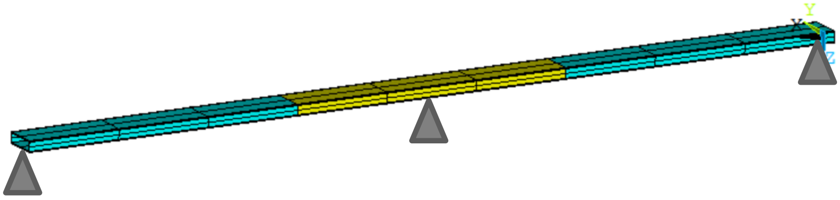


Figure 20. HSS beam, in blue material 1 and in yellow material 2

Both a “healthy” and a “damaged” scenario of the HSS beam is introduced. The main objective is to firstly, with the healthy state, calibrate the model using the sensitivity-based model updating . Secondly, by comparing the healthy and damaged state, damage is detected.

5.1.2 Calibration of the Young’s modulus in two segments.

In general, for the comparison between the numerical model and the real structure the two first natural frequencies have been selected. For the calibration of the model, two scenarios are needed: an initial assumption of the structural updating parameters and the “healthy” scenario of the HSS beam. It is important to consider that this study does not evaluate experimentally-obtained data. Simply two numerical scenarios have been set up. As structural monitoring parameter, it has been chosen to use the Young’s modulus (in [GPa]) of material 1 and 2. In the initial assumption, the values of both Young’s moduli are contained in the $\boldsymbol{\theta_0}$ parameter shown in Equation (5.1). Apart from this, the natural frequencies obtained from the modal analysis of the initial assumption are contained in $\boldsymbol{f_0}$ [Hz] in the Equation (5.2).

$$\boldsymbol{\theta_0} = [\,210 \quad 240]^T \tag{5.1}$$

$$\boldsymbol{f_0} = [\,12.302 \quad 13.2564]^T \tag{5.2}$$

For the “healthy” case scenario, the real Young’s modulus of the beam is assumed to be higher, for the material 1, it is going to be 225 [GPa] and for the material 2, [250] GPa. The “real” structural parameter will have each Young’s modulus and will be contained in $\boldsymbol{\theta_d}$. The corresponding natural frequencies are contained in $\boldsymbol{f_d}$ [Hz] .

$$\boldsymbol{\theta_d} = [\,225 \quad 250]^T \tag{5.3}$$

$$\boldsymbol{f_d} = [\,12.6743 \quad 13.6530]^T \tag{5.4}$$

Through sensitivity-based model updating, the $\boldsymbol{\theta_0}$ is updated and made equal to the $\boldsymbol{\theta_d}$. The Jacobian matrix is calculated with the $\boldsymbol{\theta_0}$ by the finite difference method. Afterwards, a validation of the Jacobian matrix is performed in order to ensure that the finite difference method was correctly used. This is a visual validation and it can be seen in Figure 21. In this case, for the Young’s modulus of material 1, in red are the values of the residual function and in black, the obtained Jacobian entry through the finite difference method. If it is tangent to the residual function, the Jacobian entry can be validated. In the Figure 21, it can be seen how the Jacobian entries with respect to the Young's modulus of the material 1 are valid.

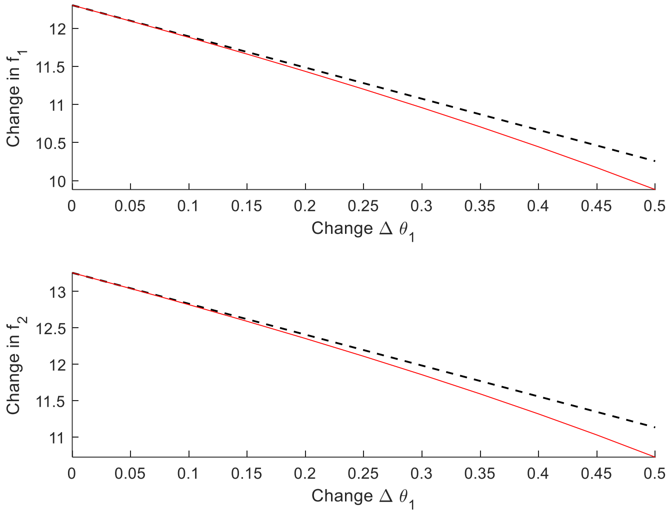


Figure 21. Validation of the Jacobian vector for parameter 1.

Once the Jacobian matrix is obtained, the model can be updated through the modified Newton-Raphson method as explained in previous sections. After seven iterations of the modified Newton Raphson method, the model is updated. In Figure 22, the changes in the Young's modulus can be appreciated. In red is the initial assumption and in blue, the updated structural monitoring parameter through sensitivity-based model updating.

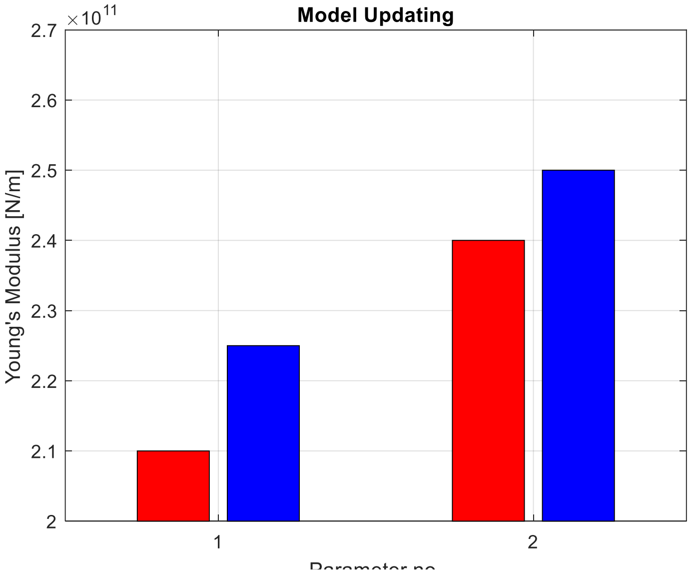


Figure 22. Calibration of the HSS beam.

In order to evaluate the accuracy of the obtained results, the relative error is calculated. The formula of the relative error is shown in Equation (5.5):

$$Error = \left|\theta_i^{result} - \theta_i^{expected}\right| / \theta_i^{expected} \tag{5.4}$$

The results of the model updating are shown in Table 1. As it can be seen, the relative error is quite low for both monitoring parameters. Therefore, it can be concluded, that the model updating was successful in the calibration of the HSS beam.

Table 1. Relative errors of the monitoring parameter.

$\theta_i^{expected}$	θ_i^{result}	$Error$
225,000,000,000	224,999,999,771	$1.01\cdot10^{-9}$
250,000,000,000	250,000,000,523	$2.09\cdot10^{-9}$

5.1.3 Updating the support displacement

In this section, a "damage" scenario is added. The idea is to transform the "healthy" scenario into the "damaged" through sensitivity-based model updating. The initial assumption is therefore, in this case, the "healthy" scenario. However, an extra structural monitoring parameter has to be included to represent damage. In previous sections of this thesis, it has been explained how support settlements are modelled in FE analysis. For this case study, damage will be represented as the vertical displacement of the middle support. For the initial assumption, the healthy state is defined, that is a zero support settlement. In this case, the structural monitoring vector $\boldsymbol{\theta_0}$ has parameters with different units, the first two correspond to Young's modulus and are in [GPa] whereas the last parameter refers to the support settlement and is in [m], these parameters are contained. The corresponding natural frequencies are contained in $\boldsymbol{f_0}$ [Hz] and have naturally the same values as the previous case.

$$\boldsymbol{\theta_0} = [\,225 \quad 250 \quad 0]^{\boldsymbol{T}} \tag{5.5}$$

$$\boldsymbol{f_0} = [\,12.6743 \quad 13.6530]^T \tag{5.6}$$

For the “damaged” case scenario, a support settlement of 0.02 [m] is implemented and the updated structural monitoring parameter will have the values presented in Equation (5.7). As a result of the support settlements, the natural frequencies will be affected as expected. The “damaged” structural parameter $\boldsymbol{\theta_d}$ will contain each Young’s modulus in [GPa] and the implemented support settlement in [m]. The corresponding natural frequencies are contained in $\boldsymbol{f_d}$ [Hz] .

$$\boldsymbol{\theta_d} = [\,225 \quad 250 \quad 0.02]^{\boldsymbol{T}} \tag{5.7}$$

$$\boldsymbol{f_d} = [\,12.6581 \quad 13.6322]^{T} \tag{5.8}$$

The result of the model updating can be interpreted in Figure 23, which shows the changes in the modal parameter. As it can be seen, the Young’s modulus of material 1 and 2 are not affected by the model updating, whereas the support settlement was updated to 0.02 [m]. This was possible after two iterations of model updating.

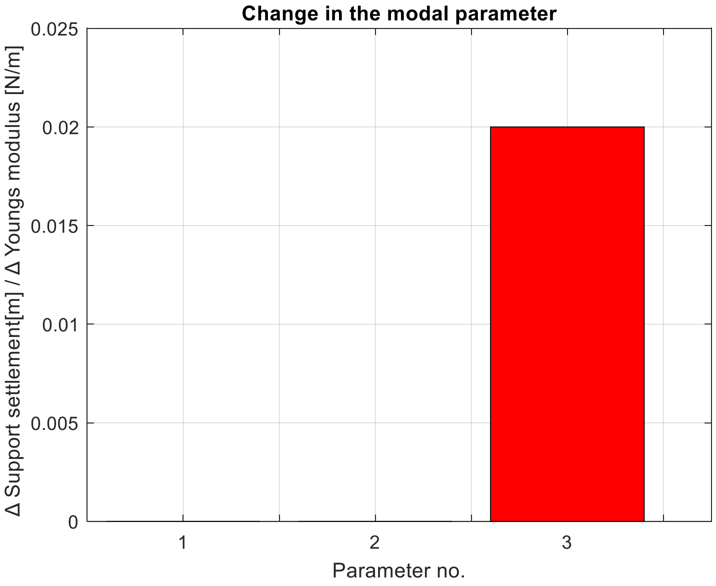


Figure 23. Change in the model updating parameters for support settlement conditions

These two studies verify the sensitivity-based model updating is useful for calibration and damage quantification. It has also been demonstrated that both the interface between MATLAB and

ANSYS and the sensitivity-based model updating have been correctly applied. It is now possible to continue the studies with a more complex model, the UniBw model.

5.2 UniBw Bridge

One of the main tasks of this thesis is to design the model of the UniBw bridge. In Figure 24, the complete model can be seen. In this section, first of all the description of the bridge is provided. Secondly, the main considerations made for its modelling are explained. Lastly, the same study as the HSS beam is performed: calibration and damage detection. The idea is to apply the sensitivity-based model updating methods to this model.

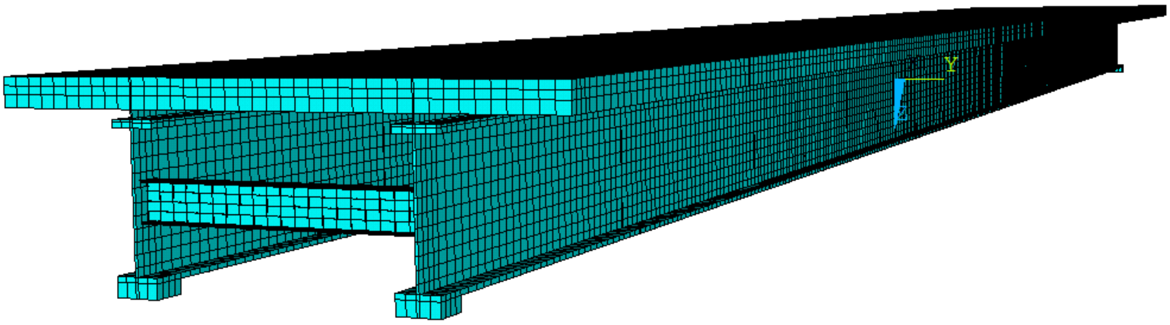


Figure 24. ANSYS model of the UniBw bridge.

5.2.1 Bridge description

In the figure below, real pictures of the bridge are shown. In Figure 25.a, a general view of the bridge is given. The bridge is 29.9 [m] long and 4 [m] wide. It is supported by abutment walls at each end and by pier situated in the middle, with two spans of 14.95 [m] each. In Figure 25.b, the view from the inside of the bridge is seen. As the picture depicts, the bridge consists of two steel HEB1000 beams which are connected on the top by cement boxes and on the bottom by steel braces. Apart from this, there are stiffeners on the HEB1000 beams every 3.6 [m].

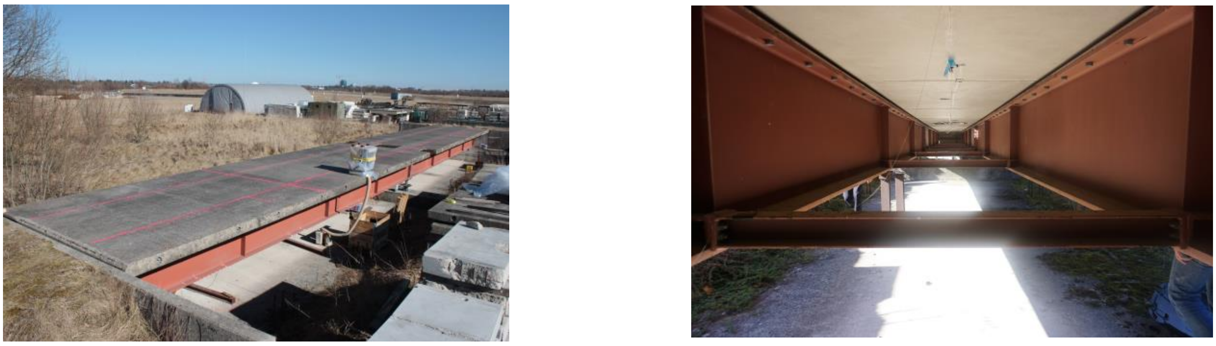


Figure 25. UniBw bridge (left), between HEB1000 profiles (right) (Jaelani et al., 2022).

On the deck, there are ten cement boxes that are 4 [m] wide and 2.98 [m] long. In Figure 26 the reinforcement of the boxes can be seen. It consists of half of a HEA340 steel profile. These profiles are then fixed to the HEB1000 beams on its bottom. In the picture, the complete front view of the cement boxes is not shown but in reality there are two reinforcement beams pro cement box. Apart from this, it is important to note that the concrete of the boxes is C30/37 as this study (Baumhauer, 2010) on the same bridge shows.

Figure 26. Cement blocks in the UniBw M bridge (Jaelani et al., 2022)

The HEB1000 beams are connected through nine braces that are placed between them. These braces can be seen in Figure 25. B and in the Figure 27. Two types of braces were used. At the ends of the bridge there are two IPE270 braces situated in the middle of the HEB1000 beams and along the bridge there are seven HEA120 situated in the lower part of the HEB1000 as seen in the figures.

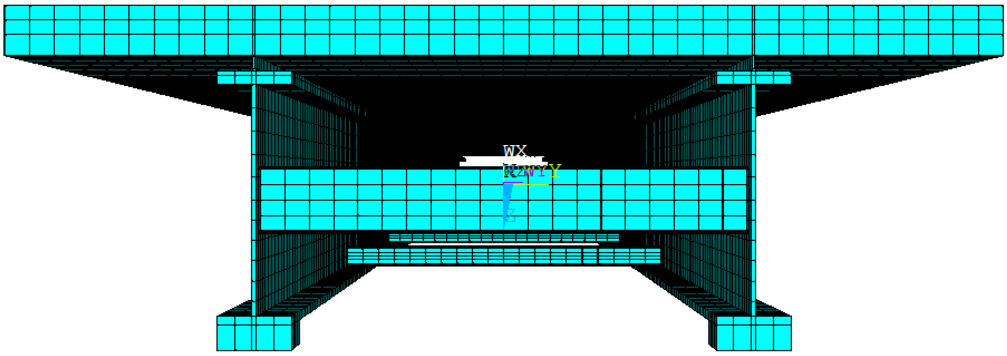


Figure 27. ANSYS model of the UniBw bridge (front view)

5.2.2 Ansys model

In this section, the main aspects of the process of the design of the UniBw model is explained. These are on the one hand, the main approximations applied to the model and on the other, how to properly mesh the UniBw model is decided.

Approximations

In this section, the most important approximations that had to be made for the UniBw bridge model are explained. As a first consideration, it was decided to maintain the whole model linear. In previous sections, it has been seen how the linearity can affect the accuracy of the numerical model. Nevertheless, considering the non-linear behaviour increases the computation time of the model considerably. As the nonlinearities are not the main interest of this thesis and in order to increase time-efficiency of the model, the model is maintained linear.

Secondly, some approximations were made to model the contact regions between the volume surfaces. In reality as it can be seen in Figure 28, there are stiffeners and screws that were not included in the ANSYS model. On the one hand, modelling the screws in the model is very time consuming and increases considerably the computation time of the model. Because of this, it was decided not to model the screws and directly consider that the contact regions are completely glued together.

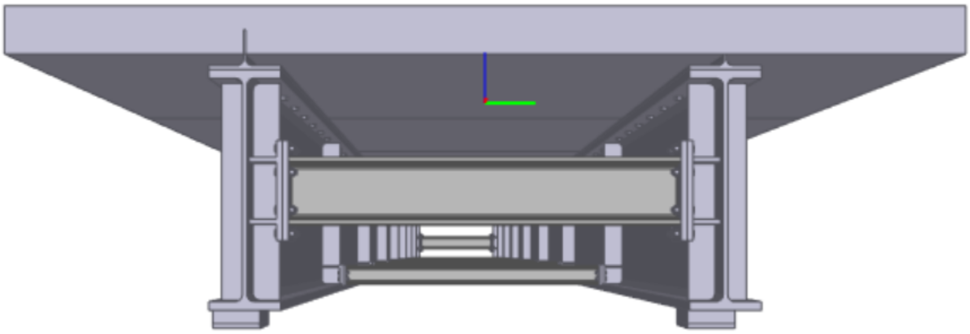


Figure 28. Front of the Autodesk Revit model of the UniBw bridge (Janßen, 2022)

On the other hand, the stiffeners were not included as they increase the number of volume cutting in the ANSYS model for its proper meshing. It has been observed that adding number of cuts to the model increases significantly the computation time.

Another contact region which was not modelled correctly is the surface between the braces and the HEB1000 profiles. In Figure 28, it is clear that there is a rigid union between them. Nevertheless, there were many problems encountered when modelling this in ANSYS. Because of this, instead of a rigid union an articulate was modelled. In Figure 29, the consequence of this implementation can be interpreted. The rigid union produces joint bending of both elements against gravitational and horizontal loads, increasing stiffness and reducing deformation. Not considering this union correctly, introduces a big error to the model.

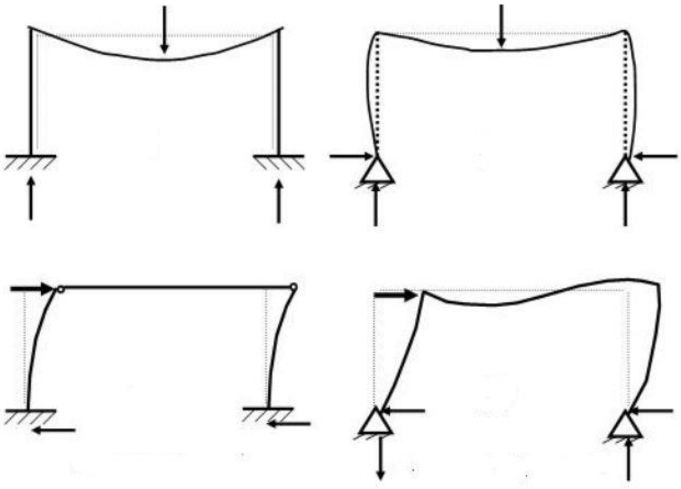


Figure 29. Deformation of articulated beams (right) vs rigid (left) (Barluenga, 2018).

Finally, the material properties of the model were assumed according to the provided literature. The materials considered in the model are steel and cement. For steel in the first run, a Young's modulus of 210 [GPa] is going to be assumed and for the cement as it is a C30/37 (Baumhauer, 2010), a Young's modulus of 33GPa . The density of the steel is 7850kg/m^3 and the concrete 2500 kg/m^3. It is true that some properties are going to be updated with real measuring data, but he uncertainties of these properties will inevitably introduce errors to the model. In conclusion, there were several approximations applied to the model and as a result, it is necessary to calibrate it for it to be useful in damage detection.

Meshing considerations

Once the geometry of the model has been assigned, the model has to be meshed. Meshing consists in dividing the structure into smaller 'elements' as previously explained. To apply a meshing to a model, it is necessary to define the shape of the 'elements' and their size. How the structure is meshed, affects the general accuracy and the computational efficiency of the solution. Because of this, it is important to make the proper considerations in the type of meshing and the size of the elements.

a) Hexahedral vs tetrahedral meshing

The most common type of meshes are tetrahedral or hexahedral. In Figure 30, both types of meshing are depicted. The type of meshing affects the solution of the model. The goal in choosing a particular type of mesh, whether it is hexahedral or tetrahedral, is to find the best balance between simulation accuracy, computational time, convergence rate, and difficulty in generating the numerical model (CFD, 2022). Because of this, both types will be compared and the option that best suits the model of the bridge will be chosen.

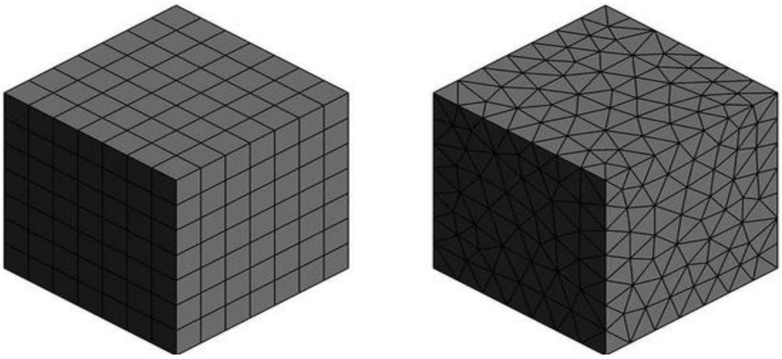


Figure 30. Hexahedral (right) and tetrahedral (left) meshing (Sosnowski et al., 2018)

The generation of hex-dominant meshes often require time-consuming user interaction. This is the case in ANSYS APDL. When choosing the hexahedral meshing, it is necessary to make extra cuts on the model as depicted in Figure 31. That means that when there is an L or T shaped volume, it will have to be cut in all its corners. In the Figure 31, the cuts are shown in red for an L-shaped figure.

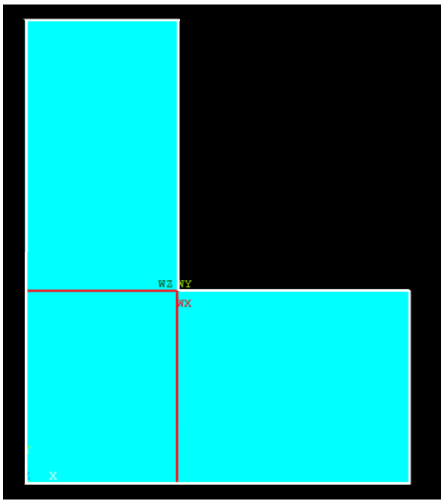


Figure 31. Needed cuts (red) in hexahedral meshing.

Apart from this, hexahedral meshes cannot be adapted to complicated geometry, such as spheres. It is important to note that in our case, the UniBw bridge is mainly composed of rectangular prisms, which have accurate results with hexahedral meshes.

There are also some disadvantages in choosing tetrahedral meshing. First of all, there is a higher number of elements created, so the runtime will be less efficient. Secondly, tetrahedral meshes can lead to some issues as inaccuracy or locking problems. Locking problems is an error that occurs in finite element analysis due to the linear nature of quadrilateral elements. Linear models can not accurately model the curvature present in the actual material bending and a shear stress is introduced. In Figure 32, the shear-locking problem is represented in the example of pure bending. In the linear element the corner nodes at horizontally shifted, resulting in a shear strain in horizontal plane. The shear stress contributes to the equilibrium of forces and thereby disturbs the deformation of the bending beam. To avoid this problem and get more accurate results, the area of interest has to be as rectangular as possible (preferably square). This is achieved with hexahedral mesh instead of tetrahedral.

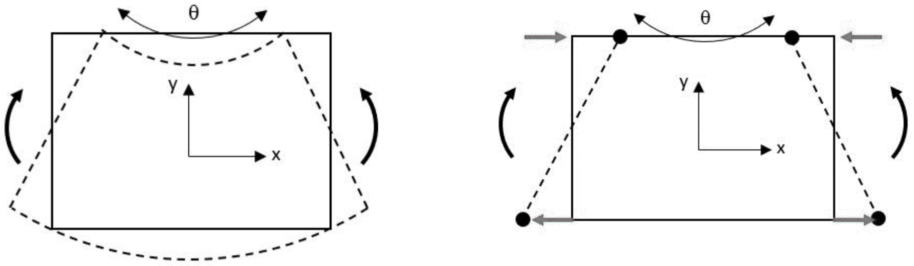


Figure 32. Real solution (left), linear solution (right) (FEM data Streamlinear, 2021)

Given the geometry of the structure of the UniBw bridge and in order to avoid locking problems, it has been decided to mesh the whole bridge with hexahedrons. Even though it was necessary to make several cuts to the volume, it is expected to improve the accuracy of the analysis.

b) Size of the elements

Another important aspect to consider in the mesh is the size of the elements of the model. In finite element analysis, the accuracy of the FEA results and requested computing time are determined by the finite element size (mesh density). According to FEA theory, the FE models with fine mesh (small element size) yield to highly accurate results but may take longer computing time. On the contrary, those FE models with coarse mesh (large element size) may lead to less accurate results but save computing time. Additionally, small element size will increase the FE model's complexity which is only necessary when high accuracy is required. Large element size, however, will reduce the FE model's size and is extensively used in simplified models in order to provide a quick and rough estimation of designs. Due to its importance, in generating FEA models, the foremost problem is to choose appropriate elements size so that the created models will yield accurate FEA results while save as much computing time as possible (Liu & Glass, 2013). A good technique is to reduce element sizes in places where big deformations/stresses/instabilities take place. This allows for greatly increased accuracy without great expense at computing time (Skotny, 2017). In the areas that are more stable, coarser meshes can be used.

For the UniBw model, the size of the elements of the model was carefully selected. Different element sizes were applied to the model and their result was checked. For “poorly” shaped elements, ANSYS APDL shows shape warnings. These are identified by the program through an element shape checking which is done after meshing. In a good model, no shape warnings should occur (ANSYS I. , 2021). After analysing the results of the model for different element sizes, it can be seen through the summary of the shape check that these warnings appear due to its aspect ratio, which is a measure of the stretching of a cell. This means that the elements are not cubes with equal side lengths, which leads to errors in the results. To avoid the warnings and the possible errors, the size of the elements has to take into account the size of the geometry of the structure. The smallest measurements on the structure are clearly the thicknesses of the profiles which are around 0.01-0.2 [m] thick. Because of this, for element sizes smaller than 0.1 [m] no warning was given by the program.

After applying several different element sizes to the model, it was determined that in order to obtain zero warnings from the program, the element size has to be equal to or less than 0.1 [m] as established before. It is also possible to make it coarser. However, for an element size of 0.07, the running time of the modal analysis was considerably larger. As a result, by keeping the element size as 0.1 [m], we obtain zero warnings from the program and a low computation times with respect to finer meshes. It was decided to maintain the 0.1 [m] element size for the whole bridge.

5.2.3 Calibration of the Young’s modulus

In this section, sensitivity-based model updating is applied to the UniBw model. As in the previous case study, the HSS beam, this section willl start with the comparison of the structural health monitoring vector and the corresponding natural frequencies in both its initial and “healthy” state. It was decided to use the Young’s modulus of the steel and the Young’s modulus of the concrete of the structure as structural monitoring parameters. Even though the literature review (Baumhauer, 2010), states their values, Young’s modulus are very common structural updating parameters and have been used in many cases as explained in previous sections. The residuals in this case, are going to be the first three natural frequencies. In the initial assumption, the values of the Young’s modulus of the steel and the Young’s modulus of the concrete are contained in the $\boldsymbol{\theta_0}$ parameter shown in Equation (5.8). Apart from this, the natural frequencies obtained from the modal analysis of the initial assumption are contained in $\boldsymbol{f_0}$ [Hz] in the Equation (5.2).

$$\boldsymbol{\theta_0} = [\,210 \quad 33]^{\boldsymbol{T}} \tag{5.9}$$

$$\boldsymbol{f_0} = [\,9.2281 \quad 14.4173 \quad 29.5711]^T \tag{5.10}$$

For the “healthy” case scenario, the real Young’s modulus of the bridge is assumed to be higher, for the steel, it is going to be 220 [GPa] and for the concrete, 34 [GPa]. The “real” structural parameter will have each Young’s modulus and will be contained in $\boldsymbol{\theta_d}$. The corresponding natural frequencies are contained in $\boldsymbol{f_d}$ [Hz] .

$$\boldsymbol{\theta_d} = [\,220 \quad 34]^{\boldsymbol{T}} \tag{5.11}$$

$$\boldsymbol{f_d} = [\,9.4387 \quad 14.7444 \quad 30.2440]^T \tag{5.12}$$

With the initial structural monitoring parameter and the final residual vector, sensitivity-based model updating is applied to the model. The results can be seen in Figure 33. In red are the initial assumptions and in blue, the final values.

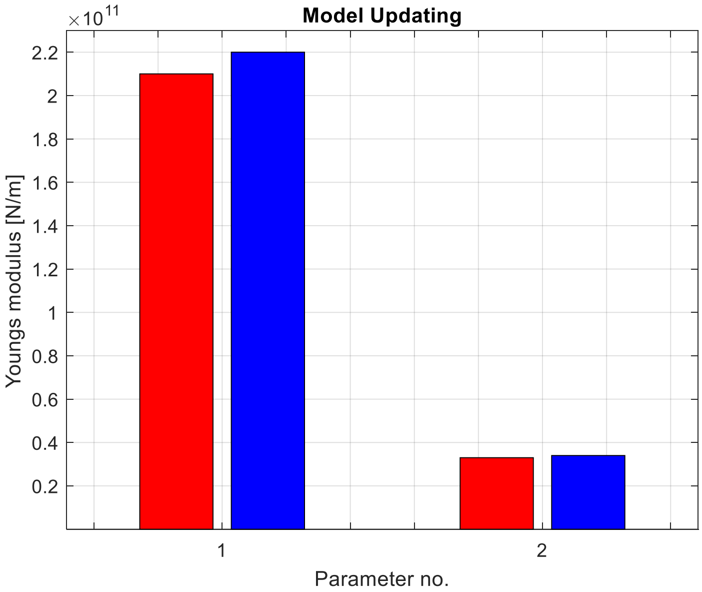


Figure 33. Model updating of UniBw model. Calibration.

The results of the model updating are shown in Table 2. It is important to note that a total of four iterations were needed to converge the solution. As it can be seen, the relative error is quite low for both monitoring parameters. Therefore, it can be concluded, that the model updating was successful in the calibration of the UniBw model.

Table 2. Relative errors of the calibration of the UniBw model.

$\theta_i^{expected}$	θ_i^{result}	$Error$
220,000,000,000	219,999,999,302.	3.17 10^{-9}
34,000,000,000	33,999,999,911	2.61 10^{-9}

5.2.4 Damage detection

In this section, a “damage” scenario is added. Damage will be represented as the vertical displacement of the middle support as in the HSS case study. The idea is to transform the “healthy” scenario into the “damaged” through sensitivity-based model updating. The initial assumption is therefore, in this case, the “healthy” scenario. However, the support settlement of the middle support has to be included in the structural health monitoring parameter for its assessment. For the initial assumption, the “healthy” state is defined, that is, a zero support settlement. In this case, the structural monitoring vector $\boldsymbol{\theta_0}$ has parameters with different units, the first two correspond to Young’s modulus and are in [GPa] whereas the last parameter refers to the support settlement and is in [m], these parameters are contained. The corresponding natural frequencies are contained in $\boldsymbol{f_0}$ [Hz] and have naturally the same values as the previous case.

$$\boldsymbol{\theta_0} = [\,220 \quad 34 \quad 0]^T \tag{5.13}$$

$$\boldsymbol{f_0} = [\,9.4387 \quad 14.7444 \quad 30.2440]^T \tag{5.14}$$

If a support settlement of 0.03 m is applied to the model, the updated structural health monitoring parameter $\boldsymbol{\theta_d}$ would have the values presented in Equation (5.15). As a result of the support settlement, the natural frequencies $\boldsymbol{f_d}$ [Hz] would change to the values presented in (5.16).

$$\boldsymbol{\theta_d} = [\,220 \quad 34 \quad 0.03]^T \tag{5.15}$$

$$\boldsymbol{f_d} = [\,9.4001 \quad 14.7168 \quad 30.2121]^T \tag{5.16}$$

Through model updating, the parameters are updated as shown in the Figure 34. The result was successfully converged in two iterations. In the figure, Young's moduli are shown on the left side in blue and the support settlement on the right side in red. Each with its own units. For the Young's moduli, the lighter blue are the initial assumptions and the darker blue the final result of the model updating. It can be seen that through the model updating, the Young's modulus of both the steel and concrete remain constant.

On the right side in red, is the result of the support settlement in [m]. Initially it is 0 as stated in the beginning of the section. This is the reason why it cannot be seen on the graph. After the model updating, the support settlement is approximately 0.03 [m], as expected. This means that the model updating was successful.

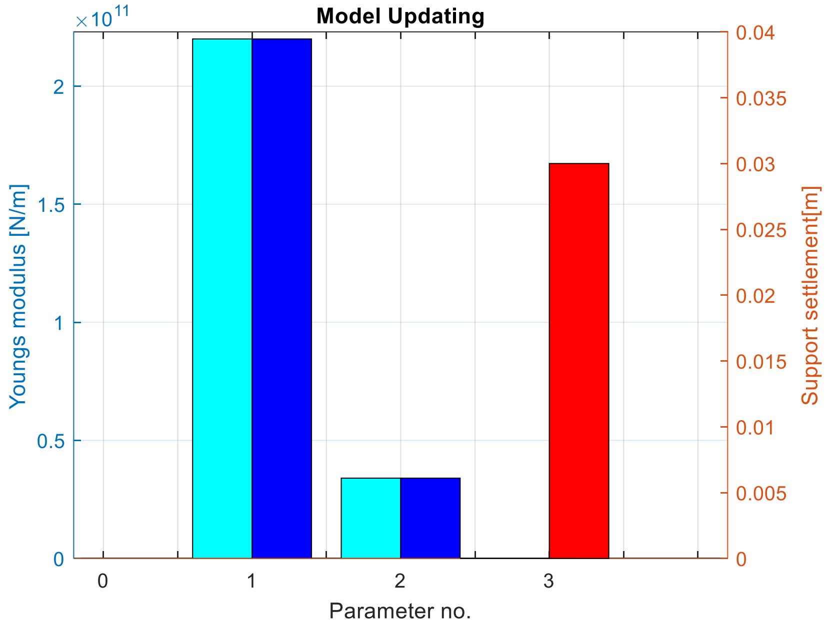


Figure 34. Model updating of the UniBw bridge damage detection

In Table 3, the relative errors and the comparison between the initial and updated monitoring parameters are shown. They relative errors of each parameter is very low and therefore, the model was successfully updated.

Table 3. Relative errors of damage detection in UniBw model.

$\theta_i^{expected}$	θ_i^{result}	$Error$
220,000,000,000	219,999,978,166.	$9.92\ 10^{-8}$
34,000,000,000	34,000,004,121	$1.21\ 10^{-7}$
0.03	0.02999	0.003

Finally, as a last visual representation of the success of the model updating method, the Figure 35 is shown. This figure represents the deviations of the structural monitoring parameters. In blue are the changes in the Young's modulus of the concrete and steel and in red, the changes of the support settlement are represented.

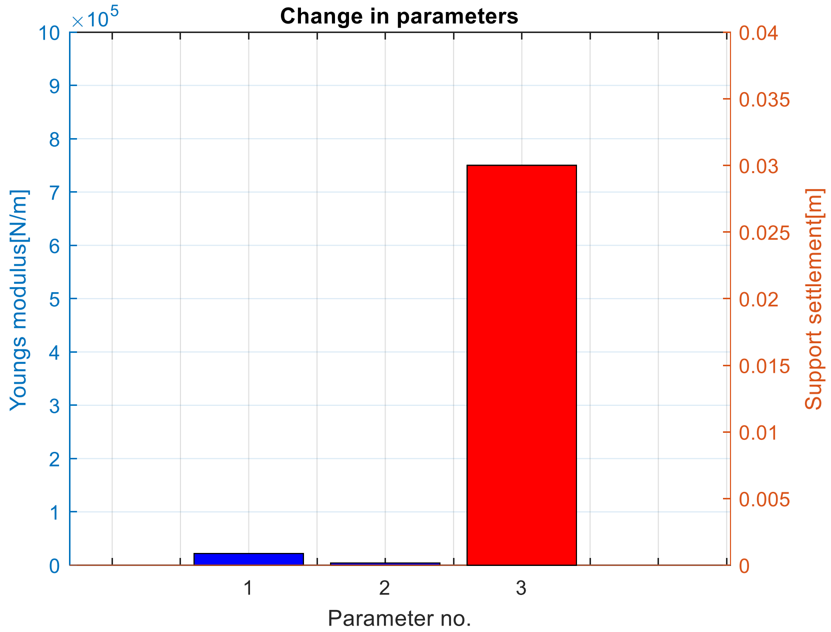


Figure 35. Change in parameters in damage detection.

With this case study, it is represented how sensitivity-based model updating techniques can be implemented to complex models through the interface of MATLAB and ANSYS. Once again, it is necessary to highlight the benefits to sensitivity-based model updating. As the different case studies presented in this thesis have concluded, sensitivity-based model updating allows the damage localization and quantification, two very sought-out characteristics for SHM methodologies. Finally, appendix C contains figures of the ANSYS model in the damaged scenario for the three studied modes in order to visually verify that the obtained modes in the analysis correspond to the eigenfrequencies.

6 Conclusions

In this last section, a summary and an analysis of the future research in the field are given. Firstly, the main ideas and conclusions of the thesis are presented. For this purpose, it is important to note that the thesis is divided into two main parts: on one hand, the literature review on model updating and on the other, the application of model updating to the UniBw model. Furthermore, the conclusions of this thesis also offer a reflection of possible future research.

6.1 Summary

Due to the continuous advances in information technology and sensors, SHM is becoming a cost-efficient and reliable solution to ensure safety and reliability of structures. For the prediction or assessment of damage based on SHM, a digital model of the real structure is required. To improve the correlation between the numerical model and the real structure and make its predictions as credible as possible, it is increasingly common to combine numerical modelling with the results of experimental investigation of the structure. This thesis provides an overview of the process of updating finite element models based on the results of experimental investigations and the most used methods. Several relevant conclusions can be drawn from the literature review:

1. Considering the actual behaviour of the structure and the parameters that most credibly represent it, one can conclude that the numerical model of the structure can be improved based on the experimentally determined dynamic properties (e.g. natural frequencies and mode shapes). The accuracy of the experimentally obtained data will influence the quality of the updated numerical model.
2. Selecting the appropriate design variables has a significant influence on reducing errors and simplifying the finite element model. It is very important that the selected design variables represent the real structural behaviour as well as possible. The change in structural dynamic parameters is most associated in the changes of structural stiffness in form of the damage.
3. There are several finite element model updating methods represented in the literature review of this thesis. For the application of the FEMU to the UniBw model, the sensitivity-based model updating method was selected. The existing research has highlighted the advantage of the stochastic method (e.g. Bayesian), which provides the overall probability of the distribution of the physical parameters under consideration. Nevertheless, it was chosen to use the sensitivity-based model updating for the case studies as they have a similar background to sensitivity-based statistical tests, which is currently used at the Chair of Non-destructive Testing at TUM to perform the damage diagnosis.

Therefore, for this thesis it was more consistent to develop the sensitivity-based model updating technique.

Apart from this, the sensitivity-based model updating method was successfully applied to the numerical model of the UniBw bridge. For this purpose, the numerical model had to be firstly designed. Then it was possible to apply FEMU for its calibration and damage detection. The complete process involved several considerations:

1. First of all, there were several modelling assumptions which had to be made to simplify the numerical model. It was decided to maintain the complete structure linear. Moreover, there were some complications encountered when modelling the system connectivity of the braces with the main profiles of the bridge. Additionally, the material constants of the bridge were maintained constant throughout the whole bridge. These approximations introduce errors to the model and as a result, it is necessary to improve it through FEMU methods.
2. Secondly the updating structural parameter had to be selected for model updating. It was decided that due to the uncertainties presented in the Young's modulus of both materials of the bridge (steel and concrete), these would be the structural health monitoring parameter. As an example, in order to proof the damage detection algorithm, a damage scenario was modelled that represented the change in the support settlement. Therefore, for the damage detection, the support settlement was also a structural monitoring parameter.
3. Finally, the results showed, how it is possible to quantify damage through sensitivity-based modal updating. The results were quite accurate for all the case studies and proof of concept studies presented, not only the UniBw bridge model.

6.2 Future research

The Chair of Non-destructive Testing at TUM in collaboration with other universities have already performed field work on the UniBw bridge. Real vibration data has been extracted for several scenarios. Firstly, an undamaged scenario, and then various damaged scenarios were evaluated such as a change in middle support settlement. All of these are collected in the forthcoming paper (Jaelani et al. , 2022).

With the undamaged scenario, the FE model can be calibrated. This would allow the development of a benchmark study of the UniBw bridge. Through the several damage scenarios that have been applied to the real bridge, it is possible to validate further model updating techniques and develop a benchmark study of the UniBw bridge.

Unfortunately, this has not been possible in the duration of this thesis. The dynamic response of the initially designed FE model had too many differences from the real bridge and the solution did not converge for only two structural updating parameters. Nevertheless, the provided FE model, can be used as a reference for its calibration and to perform further studies on model updating techniques at different damage scenarios.

The approximation on the system connectivity of the braces to the main profiles could probably be the leading issue in the lack of the convergence in the model updating of the FE model. Therefore, as a next step, a different solution to model this connection could be proposed. Aside from this, in this thesis it has been shown that model updating structural parameters can be used to improve system connectivity. This could be another approach to solve this issue.

In general, a sensitivity analysis should be performed on the FE model before its updating. This way the parameters that are most sensitive to the output can be chosen. Nevertheless, it should be noted that the number of parameters should be as small as possible to avoid ill-conditioning issues. Through these steps, the calibration of the FE model of the UniBw bridge could be obtained.

To conclude, the relevance of benchmark studies in FEMU is highlighted. Such case studies can verify the efficiency of different model updating techniques, develop damage detection algorithms, study the selection of structural parameters in model updating and more, contributing to the research of non-destructive SHM.

7 Appendix A: ANSYS APDL - runAnalysis

```
/SOLU ! STATIC ANALYSIS
ANTYPE, static
PSTRES,ON ! Stress stiffening
NLGEOM,OFF ! Large deflections
outres,all,last
EMATWRITE,YES
OUTPR,,1
NSEL,S,loc,x, -0.115,0.115
NSEL,R,loc,y, d - 0.115, d + 0.115
NSEL,R,LOC,z,0.5
D,all,uz, theta3
allsel,all
SOLVE
SAVE
FINISH
```

```
/SOLU ! MODAL ANALYSIS
allsel,all
NSEL,all
nModes = 15

ANTYPE, modal
MODOPT, LANB, nModes
MXPAND, nModes, ,,YES
PSTRES,ON
NLGEOM,OFF ! Large deflections

SOLVE
SAVE
FINISH
```

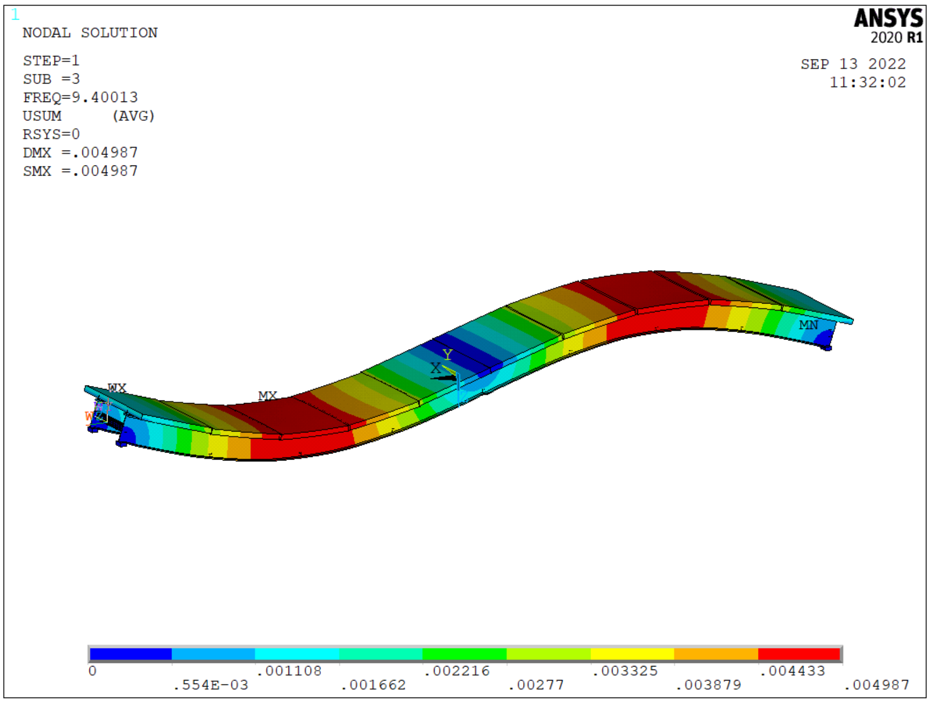
8 Appendix B: Interface between ANSYS and Matlab

```
function [f, phi] = askANSYS(theta)
%% Inferface MATLAB and ANSYS
orb = initialize_orb();
load_ansys_aas();
iCoMapdlUnit = actmapdlserver(orb,'aaS_MapdlId.txt');
char(iCoMapdlUnit.executeCommandToString('/clear'));

%% Define Monitoring Vector
char(iCoMapdlUnit.executeCommandToString(['theta1 = ',num2str(theta(1))]));
char(iCoMapdlUnit.executeCommandToString(['theta2 = ',num2str(theta(2))]));
char(iCoMapdlUnit.executeCommandToString(['theta3 = ',num2str(theta(3))]));
%% Build Model
char(iCoMapdlUnit.executeCommandToString('/DELETE, Res_ACC, dat'));
char(iCoMapdlUnit.executeCommandToString('/input,runModel1,txt'));

% Wait for ANSYS to respond
count = 0;
disp(['1. ANSYS is building the model']);
while (exist('Res_ACC.dat')==0)
    pause(0.25)
    count = count+1;
    maxCount = 500;
    if count ==maxCount
        disp('no file was generated in allowed within 180 seconds');
        break
    end
end

%% Run Modal Analysis
char(iCoMapdlUnit.executeCommandToString('/DELETE, Res_frequencies, dat'));
char(iCoMapdlUnit.executeCommandToString('/DELETE, Res_modeshapes, dat'));
char(iCoMapdlUnit.executeCommandToString('/DELETE, Res_participation, dat'));
char(iCoMapdlUnit.executeCommandToString('/DELETE, Res_K_sol, txt'));
char(iCoMapdlUnit.executeCommandToString('/DELETE, Res_M_sol, txt'));
char(iCoMapdlUnit.executeCommandToString('/input,runAnalysis1,txt'));

% Wait for ANSYS to respond
count = 0;
disp(['2. ANSYS is solving the model']);
while (exist('Res_frequencies.dat')==0 || exist('Res_modeshapes.dat')==0)
    pause(0.25)
    count = count+1;
    maxCount = 500;
    if count ==maxCount
        disp('Error: No file was generated in allowed within 180 seconds');
        beep on; beep;
        break
    end
end


%% Load Modal Parameters
f = load('Res_frequencies.dat');
phi = load('Res_modeshapes.dat');

end
```

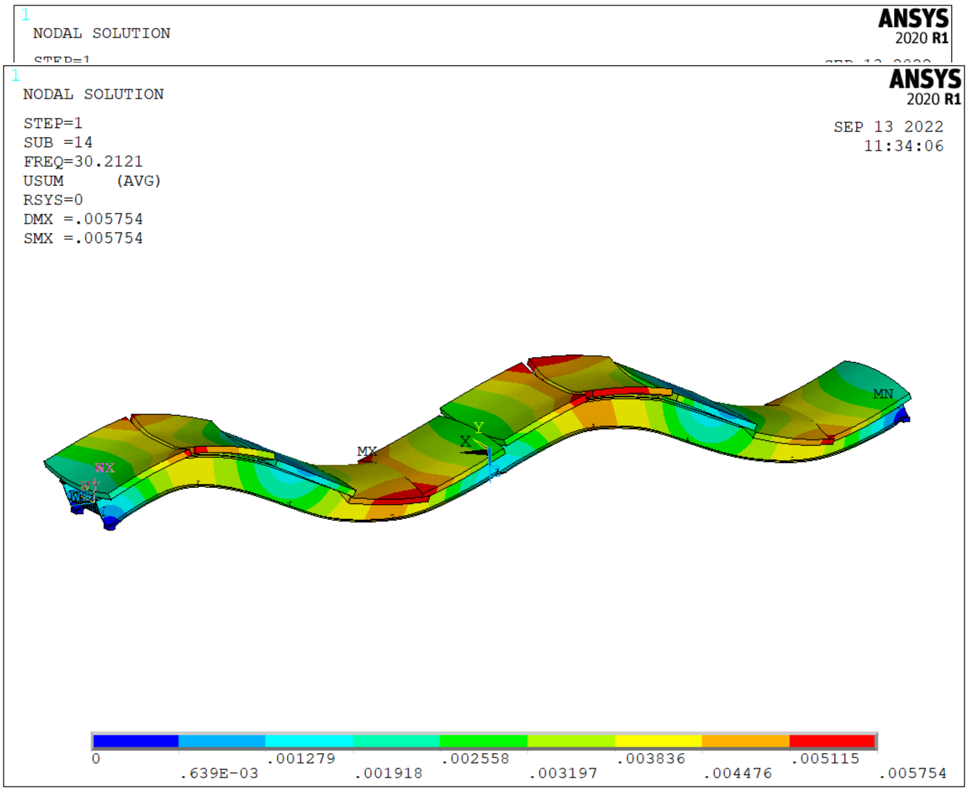
9 Appendix C: Pictures of UniBw ANSYS model

Case: Damaged scenario of the UniBw bridge, with a 0.03 [m] settlement.

First natural frequency: 9.4001 [Hz]



Second natural frequency: 14.7168[Hz]



Third natural frequency: 30.2121 [Hz]

10 Bibliography

ANSYS. (2021). *ANSYS Command Reference.* Cannonsburg: ANSYS Inc.

ANSYS. (2021). *Modeling and Meshing Guide.* Cannonsburg: ANSYS, Inc.

Asia, D. (2021, July 23). *What is a mode shape of a structure?* Keepthatshort. Retrieved August 15, 2022, from https://keepthatshort.com/what-is-a-mode-shape-of-structure

Autodesk. (2018, June 14). *Metrics and the Basics of Mechanics - Part 2*. Retrieved from Autodesk Sustainable Product Design: https://knowledge.autodesk.com/search-result/caas/simplecontent/content/metrics-and-the-basics-mechanics-part-2.html

Bandara, R. P., Chan, T. H., & Thambiratnam, D. P. (2014). Structural damage detection method using frequency response functions. *Structural Health Monitoring, 13*(4),418-429.

Barluenga, G. (2018). *Introduction to construction.* Madrid: Universidad de Alcalá.

Bathe, K. J. (2014). *Finite Element Procedures.* Massachusets: Prentice Hall, Pearson Education.

Baumhauer, A. (2010). *Beurteilung geschädigter Brückenbauwerke unter Berücksichtigung unscharfer Tragwerksparameter* [Assessment of damaged bridge structures considering sfuzzy structural parameters] [Doctoral dissertation, University of the Federal Armed Forces in Munich].

Benndorf, M., Garsch, M., Haenselmann, T., Gebbeken, N., & Videkhina, I. (2016). Mobile Bridge Integrity Assessment. *International Conference on Sensors.*

Blevins, R. D. (2001). *Formulas for natural frequency and mode shapes.* Krieger Publishing Company.

Boulkaibet, I., Marwala, T., Friswell, M., Khodaparast, H., & Adhikari, S. (2017). Fuzzy Finite Element Model Updating Using Metaheuristic Optimization Algorithms. *Society for Experimental Mechanics Series.* Frankfurt.

Butterworth, J., Lee, J. H., & Davidson, B. (2006). Experimental determination of modal damping from full scale testing. *13th World Conference on Earthquake Engineering.* Vancouver.

Carvalho, J., Datta, B. N., Gupta, A., & Lagadapati, M. (2006). A direct method for model updating with incomplete measured data and without spurious modes. *Mechanical Systems and Signal Processing, 21*(2007), 2715-2731.

CFD, C. (2022, January 15). *Hexahedral Mesh vs. Tetrahedral: Comparing High-Quality Meshing*. (Cadence Design Systems) Retrieved August 18, 2022, from https://resources.system-analysis.cadence.com/blog/msa2022-hexahedral-mesh-vs-tetrahedral-comparing-high-quality-meshing

Chegg. (2021, October 15). *The Stress-Strain Curve Of Concrete And Steel*. Retrieved from https://www.chegg.com/homework-help/questions-and-answers/-draw-stress-strain-curve-elastic-behaviour-linear-elastic-behaviour-plastic-behaviour-b-s-q66264042

Ereiz, S., Duvnjak, I., & Jiménez-Alonso, J. F. (2022). Review of finite element model updating methods for structural applications. *Structures, 41,* 684-723.

Etz, A. (2017). Introduction to the Concept of Likelihood and Its Applications. *Advances in Methods and Practices in Psychological Science, 1*(1), 1-14.

Fang, S.-E., Zhang, Q.-H., & Ren, W.-X. (2013). An interval model updating strategy using interval response. *Mechanical Systems and Signal Processing, 60-61*(2015), 909-927.

Femto Engineering . (2017, May 22). *In FEA, what is linear and nonlinear analysis?* Retrieved from https://www.femto.eu/stories/linear-non-linear-analysis-explained/

Friswell, M. I., Mottershead, J. E., & Ahmadian, H. (2001). Finite element model updating using experimental test data: parametrization and regularization. *Philosophical Transactions of the Royal Society of London, 359*(1778),169-186.

Friswell, M., & Mottorshead, J. (1996). *Finite Element Model Updating in Structural Dynamics.* Springer.

Garret, S. (2015). Introduction to Actuarial and Financial Mathematical Methods. In *Introductory Numerical Methods* (pp. 411-463). Academic Press.

Giurgiutiu, V. (2014). Introduction. In *Structural Health Monitoring with Piezoelectric Wafer Active Sensors* (pp. 1-19). Elsevier.

Gorgin, R. (2020). Damage identification technique based on mode shape analysis of beam structures. *Structures, 27*(2020*)*, 2300-2308.

Halliday, D., Resnick, R., & Walker, J. (2013). *Fundamentals of Physics.* Wiley.

Jaelani, Y., Klemm, A., Wimmer, J., Seitz, F., Köhncke, M., Marsili, F., Popp, A. (2022). *Developing a Benchmark Study for Bridge Monitoring.* Munich: University of Federal Armed Forces Hamburg.

Jaishi, B., & Ren, W.-X. (2005). Structural Finite Element Model Updating Using Ambient Vibration Test Results. *Journal of Structural Engineering, 131*(4), 618-628.

Jaishi, B., & Ren, W.-X. (2007). Finite element model updating based on eigenvalue and strain energy residuals using multiobjective optimisation technique. *Mechanical Systems and Signal Processing, 21*(5) 2295-2317.

Janßen, S. (2022). *Entwicklung von Ansätzen zur messdatenbasierten BIM-Umsetzung eines digitalen Zwillings am Beispiel einer Testbrücke.* [Development of approaches for the measurement-based BIM implementation of a digitial twin using the example of a test bridge] [Master's thesis, University of the Federal Armed Forces Hamburg].

Jiménez-Alonso, J., & Sáez, A. (2018). Assessment of the deterioration of concrete structures using a finite element model. In *Eco-Efficient Repair and Rehabilitation of Concrete Infrastructures* (pp. 235-260). Woodhead Publishing.

Khodarapast, H. H. (2010). *Stochastic Finite Element Model Updating and its Application in Aeroelasticity.* [Doctoral dissertation, University of Liverpool].

Krieger, J. (2019, February 11). European bridge maintenance: monitoring safety in Germany. *Innovations News Network*. Retrieved from https://www.innovationnewsnetwork.com/european-bridge-maintenance/568/

Liu, Y., & Glass, G. (2013). Effects of Mesh Density on Finite Element Analysis. *SAE International.* Detroit.

Lye, A., Cicirello, A., & Patelli, E. (2021). Sampling methods for solving Bayesian model updating problems: A tutorial. *Mechanical Systems and Signal Processing. 159.* 107760. .

Mares, C., Dratz, B., Mottorshead, J., & Friswell, M. (2006). Model Updating Using Bayesian Estimation. *ISMA, 5.* Leuven.

Mendler, A. (2020). *Minimum Diagnosable Damage and Optimal Sensor Placement for Structural Health Monitoring.* [Doctoral dissertation, the University of British Columbia].

Mendler, A. (2021). *Damage Diagnosis based on Structural Models. Lecture Notes.* Munich: Technical University of Munich.

Mendler, A., Döhler, M., & Hille, F. (2022). Detecting Changes in Boundary Conditions based on Sensitivity-based Statistical Tests. *The internation Symposium on Nondestructive Testing in Civil Engineering.* Zurich.

Mendler, A., Greś, S., Döhler, M., & Keßler, S. (2022). On the Probability of Localizing Damages Based on Mode Shape Changes. *European Workshop on Structural Health Monitoring* (pp. 233-243). Palermo: Springer.

Mohammad Amin Hariri-Ardebili, S. S.-K. (2018). Response Surface Method for Material Uncertainty. *Hindawi Shock and vibration,* 1-14.

Mordini, A., Savov, K., & Wenzel, H. (2007). The Finite Element Model Updating: A Powerful Tool for Structural Health. *Structural Engineering International, 17* (4), 352-358.

Mordini, A., Savov, K., & Wenzel, H. (2008). Damage Detection on Stay Cables using an Open Source-based Framework for. *Structural Health Monitoring, 7*(2), 91-102.

Mottershead, J. &. (2011). The sensitivity method in finite element model updating. A tutorial. In *Mechanical Systems and Signal Processing* (pp. 2275-2296).

Neitzel, F., Resnik, B., Weisbrich, S., & Andreas, F. (2011). Vibration monitoring of bridges. *Reports on Geodesy,1*(90*)*, 331-340.

Ozer, E., & Feng, M. Q. (2020). Structural Health Monitoring. In *Start-Up Creation* (pp. 345-367). New York: Elsevier.

Pastor, M., Binda, M., & Harcarik, T. (2012). Modal Assurance Criterion. *Procedia Engineering, 48*(2012), 543-548.

Ramle, M., & Hassan, M. (2017). *A review of experimental modal analysis and finite element analysis.* Faculty of Mechanical Engineering, Universiti Malaysia Pahang (UMP).

Rytter, A. (1993). *Vibrational based inspection of civil engineering structures.* [Doctoral dissertation, Aalborg University].

Simscale. (2022, July 13). *Linear Elastic Materials.* Retrieved August 23, 2022, from https://www.simscale.com/docs/simulation-setup/materials/linear-elastic-materials/

Skotny, L. (2017, March 17). *Correct Mesh Size*. (Enterfea) Retrieved August 18, 2022, from https://enterfea.com/correct-mesh-size-quick-guide/

Sönnerlind, H. (2015, June 29). *Comsol*. (Comsol) Retrieved August 20, 2022, from https://www.comsol.com/blogs/modeling-linear-elastic-materials-how-difficult-can-it-be/

Sosnowski, M., Gnatowska, R., Sobcyk, J., & Wodziak, W. (2018). Numerical modelling of flow field within a packed bed of granular material. *Journal of Physics Conference Series*, 1-12.

Sunar, Ö. (2021, February 28). *Difference between static and transient analysis*. (Mechead Engineering Analysis Design) Retrieved August 14, 2022, from https://www.mechead.com/difference-between-static-and-transient-analysis/

Teughels, A., & Roeck, G. D. (2004). Structural damage identification of the highway bridge Z24 by FE model updating. *Journal of Sound and Vibration, 278*(3), 589-610.

Thompson, M. K., & Thompson, J. M. (2017). *ANSYS Mechanical APDL for Finite Element Analysis.* Butterworth-Heinemann.

Vardanega, P., Webb, G., Fidler, P., & C.R., M. (2016). Bridge monitoring. *Innovative Bridge Design Handbook* (pp. 759-775). Bristol: Elsevier.

Wang, J., & Wang, C. (2016). Atructural model updating of frequency response function based on Kriging method. *Information Science and Control Engineering* (pp. 640-644). Beijing: Computer Society.

Wei, L., Cheng, H., & Li, J. (2012). Modal Analysis of a Cable-stayed Bridge. *Procedia Engineering, 31*(2012), 481-486.

Yepes, V. (2020). *The asset time bomb.* Valencia: Polytechnich University of Valencia.

Zhong Chena, U. G. (2016). Variation and Consistency of Young's Modulus in Steel. *Journal of Materials Processing Technology, 227*(2016), 227-243.

Zhou, S., & Song, W. (2016). Updating Finite Element Models Considering Environmental Impacts. *Smart Structures and Materials + Nondestructive Evaluation and Health Monitoring* . Las Vegas.