Polytechnic of Turin Technical University of Darmstadt

Model Up Date for the Reconstruction of Dynamic Base Excitation of an Experimental Set-Up Based on Dynamic Measurements.

Paz Levi

In a partial fulfillment of the requirements for the degree of

MASTER OF SCIENCE IN CIVIL ENGINEERING.

Model up Date for the Reconstruction of dynamic base excitation of an Experimental set-up based on dynamic Measurements.

Paz Levi

Jury:

Prof. Dr.-Ing. Rosario Ceravolo	Polytechnic of Turin
Prof. Dr.-Ing. Jens Schneider	Technical University of Darmstadt
M.Eng. Andrei Firus	
M.Sc . Hagen Berthold	

November 2019

Preface

The present thesis deals mostly with the dynamic analysis of civil structures, as the last step of my master's degree in civil engineering at the Polytechnic of Turin, Italy. As I am very enthusiastic about the earthquake phenomenon, I have chosen to deal with model updating for the reconstruction of dynamic base excitation of an experimental set-up based on dynamic measurements.

The work was carried out at the institute of structural mechanics and design in civil engineering department of the technical university of Darmstadt in Germany, and under the daily supervision of M.Eng. Andrei Firus and his colleague M.Sc. Hagen Berthold. The collaboration between both universities is a part of ERASMUS+ students exchange program I have participated in. The supreme supervision was conducted by Prof. Dr.-Ing. Jens Schneider from the Technical University of Darmstadt and Prof. Dr.-Ing. Rosario Ceravolo from the Polytechnic of Turin.

As someone that grew up in the state of Israel, and afterwards lived in Italy, both countries with high seismic risk hazard, the studies of earthquakes are of great importance to me. In my opinion, modern structural designs that take dynamic aspects into account may decrease the number of victims in the case of application of a base excitation.

I expect that the work done in this master thesis can demonstrate and shed light on the knowledge acquired by myself during the years at the university. The proof of my professional capability in applying this knowledge into realistic engineering problems will be represented in this work, with the goal to contribute to the world of research.

I would like to thank to Andrei Firus and Hagen Berthold for their ongoing help and support during these months. A good word deserve also to Marcel Hörbert for his technical assistance .Generally, a great thank deserve to the institute of structural mechanics and design in TU Darmstadt ,for gave me the all tools needed to perform this thesis.

This master thesis work is dedicating to my family in Israel, who never stopped believing and encouraging me.

Task description

Dynamic measurements of the motion quantities of a structure involve the acquisition of time-dependent functions of physical quantities. The measurement data allows the direct determination of modal parameters of the structure, such as natural frequencies and damping ratios. However, the discrete measured values can also be used for other purposes, such as inverse calculation of the applied load. This is an interesting problem, especially for structures where a direct measurement of the acting dynamic forces is not possible at all or only with great effort, but which allows an easy recording of the structural response, such as wind turbines or pedestrian and railway bridges.

Within some preliminary works, the Institute of Structural Mechanics and Design at the TU Darmstadt investigated different methods for the inverse reconstruction of the force-time histories acting on systems with known parameters. They are based on dynamic deformation or acceleration measurements under stationary or moving, time-varying loads with known or unknown locations. So far, the investigations exclusively concerned the bridge constructions with force load cases.

The main aim of this work is to apply the previous findings regarding the force reconstruction to the inverse calculation of a dynamic base excitation (displacement or acceleration time history) of a frame structure. If necessary, the existing inverse computational methods have to be extended and adapted or further methods from the literature have to be implemented. The work comprises numerical investigations as well as their validation based on measurements on an experimental set up.

The experimental setup represents a scaled frame structure model. A validation of the inverse determined time histories of the base excitation is easily possible since the investigation on an experimental model allows the direct measurement of the motion quantities at the base. In order to be able to check the general validity of the methods, the test rig was designed in a way that different structural configurations are possible. Thus, the number of stories, column stiffness and height as well as the story mass can be varied. In particular, within this work the measurement noise has to be considered and its influence on the accuracy of the solution has to be quantified. If necessary, different methods for the reduction of the noise influence are to be used in the solution methods. Moreover, the sensitivities of the solution algorithms to modeling uncertainties have to be addressed.

Contents

Abbreviations

BIM	Building Information Modeling
DOF	Degrees of Freedom
FDM	Frequency Domain Method
FEM	Finite Element Method
MDOF	Multi Degrees of Freedom
MPA	Mean Phase Angle
PSD	Power Spectral Densities
SDOF	Single Degrees of Freedom
TDM	Time Domain Method
[2D]	Two Dimensional
[3D]	Three Dimensional
[12D]	Twelve Dimensional

1. Introduction

1.1. Aim of thesis

In my work I will try to mark out possible techniques for the construction of an update dynamic model, for a further computational of reverse exaction applying on a physical real model. This kind of research will be executed in different stages. All the steps required a deep understanding of dynamics with proper application of technical and analytical skills alongside a creative mind.

In the first step a physical 3 story scaled framed structure model made with steel and aluminum materials was constructed. Every single element was measured to understand its volume and weight. This kind of information is useful for the computation of the mass matrix and the FEM model.

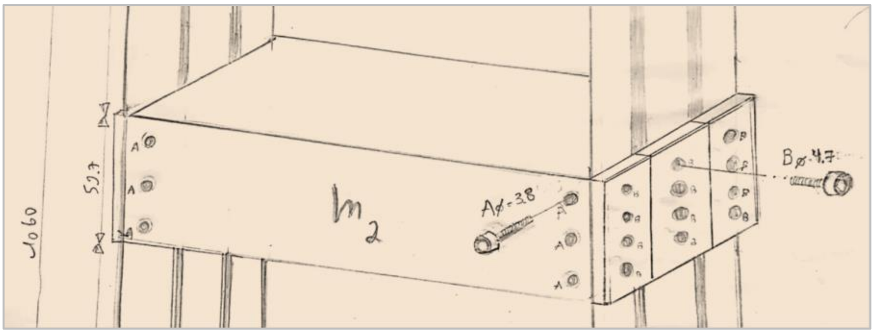


Figure 1 -Hand sketch of the second mass,conclude bolts types and plates

As a second step, we performed an experimental investigation to understand the real motion quantities at the base: modal shapes and natural frequencies. Scientific Instruments and tools such as impulse hammer and accelerometers were also used for that matter. Afterwards the analysis of the statistical measurements were done through numerical soft wares with special considerations about the noise error influence.

In the subsequent phase a numerical FEM model was performed by using a commercial software. Validation of mass property was done by preforming static analysis and calculation of the reactions on the base. Afterwards a [2D] framed modal analysis was executed.

Iteration procedure was conducted in order to understand the precise stiffness value of the joints, represented by not perfect hinges.

As an ultimate step, a literature research on the subject of inverse base excitation computation was done. Suitable methods and formulation were taken into account for the identification of numerical investigation procedure. For this procedure, a suitable measurement concept was developed.

1.2. Background

Structural dynamics over the last centuries were developed and raised by theoretical investigations and computational methods, especially in the fields of mechanical, aeronautics and civil engineering (Massimo Coradi).

In this thesis, I would like to deal with the most intense force that structure can undergo in case of an earthquake. An earthquake is an elastic vibration that is transmitted to the ground, from an inner zone called hypocenter to the surface. Earthquakes can be classified as a single impulse, or of a long duration with rich frequencies (Chopra 2012). Their particular analysis in terms of response, evolution in time and design are of a prior importance, to guarantee sufficient resistance for civil structures.

1.3. Preliminary knowledge

1.3.1. General definitions (Chopra 2012; Alonso e Finn 1992; Clough e Penzien 2003)

In order to understand the dynamics of structures, first we must define basic physical quantities used for future steps.

The Force F is a physical vector quantity that is defined as the result of a measurement performed through a dynamometer measured in [N].

Density ρ of a material defines as its mass in [Kg] divide by its volume [m^3].

Stiffness k is the resistance for deformation where the force is being applied, measured in [N*m/rad].

c is the viscous damping coefficient, that is difficult to measure, and can be approximated through an experimental test (for example the logarithmic decrement test) or assumed as a common value in terms of regular structure made by homogenous materials such as steel or concrete .

Displacement is defining as $\vec{r}(t) \equiv x(t)\vec{i} + y(t)\vec{j} + z(t)\vec{k}$ and measured in [m].

Velocity is defining as $\vec{v} \equiv \frac{d\vec{r}(t)}{dt} = \frac{dx(t)}{dt}\vec{i} + \frac{dy(t)}{dt}\vec{j} + \frac{dz(t)}{dt}\vec{k} \equiv \dot{\vec{r}}(t) = \dot{x}(t)\vec{i} + \dot{y}(t)\vec{j} + \dot{z}(t)\vec{k}$ and measured in [m/s].

Acceleration defines as $\vec{a} \equiv \frac{d\vec{v}(t)}{dt} = \frac{d^2\vec{r}(t)}{dt^2} = \frac{d^2x(t)}{dt^2}\vec{i} + \frac{d^2y(t)}{dt^2}\vec{j} + \frac{d^2z(t)}{dt^2}\vec{k} \equiv \ddot{\vec{r}}(t) = \ddot{x}(t)\vec{i} + \ddot{y}(t)\vec{j} + \ddot{z}(t)\vec{k}$ and measured in [m/s^2].

All this quantities are function of the time t in the [3D] space(Alonso e Finn 1992).The deletion of t in this formula is for simplification.

As we are dealing with a rigid body, where the relative distance between all its points remains constant during the time, we can assume linear motion representation such as: displacement u, velocity $\dot{u}$ and acceleration $\ddot{u}$ (Alonso e Finn 1992).

The oscillation of the system, that can be a natural property or a force, can be represented by its frequency (Clough e Penzien 2003).

$\omega n = \sqrt{\frac{k}{m}}$ is the natural circular frequency Measured in $\left[\frac{rad}{s}\right]$.

$fn = \frac{\omega n}{2\pi}$ is the natural cyclic frequency Measured in $[Hz]$.

$T = \frac{2\pi}{fn}$ is the inverse of the natural frequency and called natural period, mesraured in [s].

1.3.2. Equation of motion (Alonso e Finn 1992)

We considered linear elastic structure with physical properties as mass m, stiffness k and damping c. This structure undergoes a dynamic excitation. As a first simple case, we consider SDOF system, where only one possibility of motion is possible and in just one coordinate. We would like to understand the dynamic response for this kind of system, in terms of the differential equation of motion. In this case, the solution is unique and exact because just one possibility is allowed.

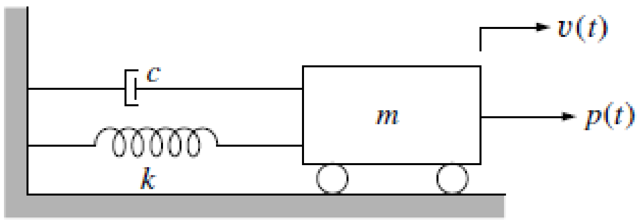


Figure 2 -Idealized SDOF system with it components (Alonso e Finn 1992)

Assuming d'Alembert's Principle of equilibrium along the direction of motion as shown in figure 2. A dynamic force p(t) applying on the direction of the degree of freedom while 3 different forces resist it in the opposite direction : the damping force $f_D(t)$, the inertial force $f_I(t)$, and the spring force $f_S(t)$.The equilibrium along the horizontal equation form the equation of motion for SFOF :

$$f_I(t) + f_D(t) + f_S(t) = p(t)$$

All resisting forces are a function of the displacement and its derivatives resulting from the application of p(t) .
Assuming the following assumption of d'Alembert's principle:
$f_I(t)=m\ddot{u}(t)$
$f_D(t)=c\dot{u}(t)$
$f_S(t)=ku(t)$

By preforming substitution, the equation of motion In terms of the physical quantities is:

$$m\ddot{u}(t) + c\dot{u}(t) + ku(t) = p(t)$$

In our case of research, the physical model represents more than one possibility of motion, therefore, the previous formulation is not sufficient for describing the dynamical behavior. We have to assume MDOF system that is an approximation of the true dynamic behavior.

A development of the SDOF motion equation should be done by using mathematical steps. The overall system consists of SDOF equations, therefore our equation may be transformed into vector and matrix representation. The equation of motion In terms of the physical quantities for a MDOF system is:

$$[M]\{\ddot{u}(t)\} + [C]\{\dot{u}(t)\} + [K]\{u(t)\} = \{p(t)\}$$

Assuming vector representation for the displacement {u(t)} , velocity $\{u(\dot{t})\}$ and acceleration $\{u(\ddot{t})\}$.

$[K]=\begin{bmatrix} k_{11} & \cdots & k_{1i} \\ \vdots & \ddots & \vdots \\ k_{j1} & \cdots & k_{ij} \end{bmatrix}$ is the stiffness matrix, with stiffness influence coefficients kij .where excitation corresponding coordinate i due to unit displacement of coordinate j.

$[M]=\begin{bmatrix} m_{11} & \cdots & m_{1i} \\ \vdots & \ddots & \vdots \\ m_{j1} & \cdots & m_{ij} \end{bmatrix}$ is the mass matrix, represented by the relationship between the accelerations of the degrees of freedom and the resulting inertial forces as coefficients mij.

$[C]=\begin{bmatrix} c_{11} & \cdots & c_{1i} \\ \vdots & \ddots & \vdots \\ c_{j1} & \cdots & c_{ij} \end{bmatrix}$ is the damping matrix, with damping influence coefficients cij .where excitation corresponding coordinate i due to unit velocity of coordinate j.

1.3.3. Modal analysis and eigenvalue problem (Chopra 2012)

In order to find the natural properties of some structures (or a structure), one may consider the case of free vibration motion-without damping and external force application. Referring to our case considering the global solution of MDOF, results that:

$$[M]\{\ddot{u}(t)\} + [K]\{u(t)\} = \{0\}$$

This system is composed of N differential equations, where N is defined as an integer number of DOF related to our structure. We may look for a solution in term of acceleration u, that satisfy the banal initial conditions at time t=0: u(0)=0 and $\dot{u}$(0)=0 .

Fundamental dynamics natural property, that does not depend on external conditions but merely on the mass m and the stiffness k is the <u>modal shapes</u> ϕ_n (known also as "eigenmodes" or "natural modes of vibration").

If free vibration is initialized by imposed displacement, the structure will deflect, and afterwards vibrate with some unique harmonic motion that maintains the initial deflected shape: corresponding to some Eigen mode ϕ_K with natural frequency f_k .

The previous system of equations can be solved with what called the "Eigenvalue Problem", whose solution brings the natural frequencies and modal shapes:

$$\{u(t)\} = q_n(t)\ \{\phi_n\}$$

Where the quantity that is time dependent is the harmonic oscillator:

$$\{q_n(t)\} = A_n cos\ \omega_n t + B_n sin \omega_n t$$

A_n and B_n are constants that depend on the initial conditions.

Combining the previous two equations lead to $u(t) = \phi_n(A_n cos\ \omega_n t + B_n sin\omega_n t)$. One possible solution for this equation is the banal solution, where no motion occur such that $\phi n = 0$. This solution may be neglect.

Other possibility is when the following condition is satisfied:

$$[K]\{\ \phi_n\} = \{\omega^2{}_n\}[M]\{\ \phi_n\} \quad \text{can be rewritten as} \quad ([K] - \{\omega^2{}_n\}[M])\{\ \phi_n\} = \{0\}$$

Where [K] and [M] are known properties of the structure, and the unknowns are $\omega^2{}_n$ and ϕ_n. The last equation has non banal solution while the determinate is expanded as so:

$$det([K] - \{\omega^2{}_n\}[M]) = 0$$

For N number of DOF we will obtain a polynomial of order N. The stiffness matrix [K] is symmetric and positively defined because we consider rigidly supported structures, that prevent rigid motion. The same applies to the mass matrix [M], where the DOF with null lumped masses are deleted in the static analysis. This symmetricity and positivity features brings us to N real and positive roots for $\omega^2{}_n$.

In conclusion, the solution leads to N natural circular frequencies ω_n (arranged from the smallest to the largest $\omega 1 < \omega 1 < \ldots <$ C) with corresponding N eigenvectors, that represent the shape of vibration corresponding to each natural frequency.

1.3.4. Normalization of modes (Chopra 2012)

The N eigenvectors previously discussed can be gathered tighter into square Modal matrix Φ :

$$[\Phi] = [\{\phi i\}, \ldots . \{\phi \mathrm{N}\}]$$

The spectral matrix Ω contains the all-natural frequencies:

$$[\Omega^2] = \begin{bmatrix} \omega^2 i & 0 & 0 \\ 0 & \ddots & 0 \\ 0 & 0 & \omega^2 n \end{bmatrix}$$

In order to move forward with the analysis, especially if we are interested in applying a superposition effect in a pre-uncoupled system, the normalization property of the modal shapes must be checked-the scaled effect should not disturb the forward computational approach. This statement can be described and verified mathematically by:

$$[\Phi]^{T}[M][\Phi] = [I]$$

$$[\Phi]^{T}[K][\Phi] = [\Omega^2]$$

Where [I] is an identity matrix.

1.3.5. Application of the theory of elasticity –Determination of the stiffness matrix (Chopra 2012)

As in this thesis we are dealing with a normal structure made by homogenous material that undergoes a relatively small deformation, we can consider a general approach for the linear systems. We can implement a unique calculation procedure, based on equilibrium method for structures with a small DOF.

"The force f_{si} at DOF I associated with displacement u_j , where j=1…N , is obtained by the superposition : $f_{si}=k_{i1}u_1 +k_{i2}u_2+....+k_{ij}u_j+....+k_{iN}u_N$ " (Chopra 2012).

f_{si} are static external force applying on the system .The mentioned relationship can be seen in a system representation as :

$$\{f_s\}=[K][u]$$

This formulation has an analogy to the Hooke Law (Hooke 1678). Each jt'h column of the stiffness matrix [K] can be determined easily by assuming unitary static forces applied on the system, such as $\{f_s\}=[1]$. We can make a substitution:

$$\{1\}=[K][u]$$

Or

$$[K] = [u]^{-1}$$

Practically, when unitary forces are applied on a normal structure, their relative displacements are equal to the inverse of the stiffness's relate to each node of DOF. At this point, we can proceed with an easy computation of the stiffness matrix.

2. Experimental Investigation

2.1. Scaled model

2.1.1. Static scheme, Choosing of DOF and axes

Common civil structures are called "Frame structures". These structures have the combination of beams, columns, and slabs that can resist lateral and gravity loads. Their main aim is to overcome the large bending moments developing due to the applied load. Because of our adherence to the theory of elasticity, elements that share the same displacement will absorb loads proportional to their stiffness. We choose a physical scaled 3 stories fix ending rigid frame as a reference for this kind of research.

The masses m1, m2 and m3 are the slabs, represented by hollow rectangular boxes made of aluminum. Due to their rigidity, they cannot deform in their plan. Moreover, they take most of the horizontal forces applied on the frame.

The columns are represented by thin plates made of steel. Their torsional stiffness is much higher comparing to the bending one, allowing deformation in the horizontal direction where the load is being applied horizontally.

The following is our arbitrary choice of Cartesian axes: Z is in the vertical direction and positive towards gravity, X is the principal horizontal direction of the motion and Y is the fixed inner direction that does not allow motion.

Smaller steel plates, rings and bolts are used to create a non-perfect rigid connection.

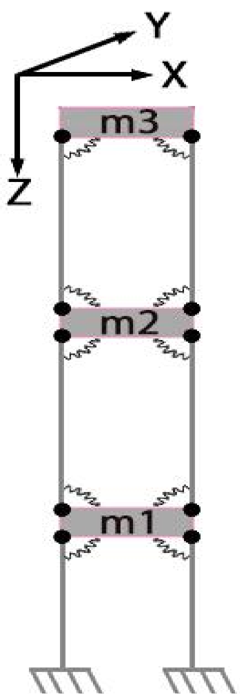


Figure 3-static scheme of scaled model in XZ Plane

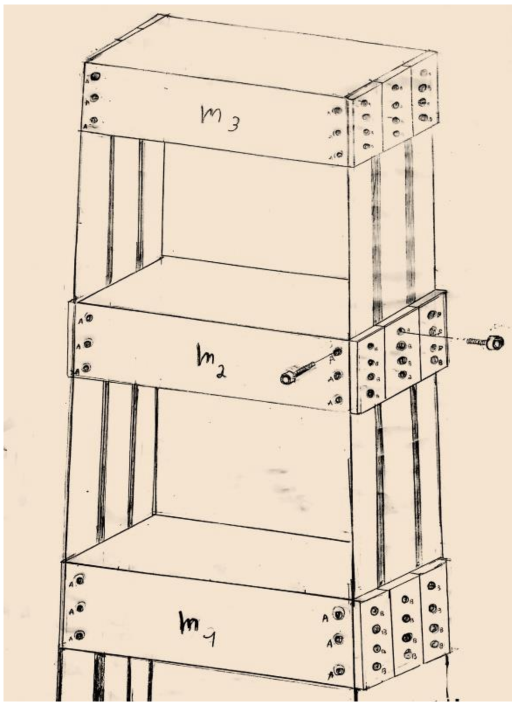


Figure 4- Hand sketch of the scaled model in [3D]

Since the configuration of this model does not permit motion along the Y direction, the problem can be simplified to into [2D] planer, where the motion is allowed in the horizontal X direction and in the vertical Z. Nevertheless, the 3 different independent masses require an MDOF consideration. Therefore, we can define the number of DOF as:

$$N = 3$$

The static scheme is represented in figure 3. Considering the bases, the structure is perfectly fixed to the ground in all directions. The connection between the boxes and the lateral plates was made by manual bolts and rings that were not welded. Therefore, we can consider each joint (except the bases joints) as a superposition of a hinge and rational springs. Each spring can assume a particular value of stiffness.

2.1.2. Materials

Chromium-nickel austenitic stainless spring steel X10CrNi18-8 (DEUTSCHE EDELSTAHLWERKE GMBH) :

The combination of high chromium and restricted nickel content produces a metastable austenitic structure. The chemical alloy composition is the following [%]: C (0.05 – 0.15), Cr (16.00 – 19.00), Ni (6.00 – 9.50), Mo max 0.80, N max. 0.11. This can be used as a spring steel to 300 °C.

Its density is equal to 7.9 [kg/m^3], Electrical resistivity at 20 °C is 0.73 [½ mm2/m] and Thermal conductivity at 20 °C is 15 [W/mK]. The Mechanical properties are the following: Yield strength is 400 [N/mm^2], Tensile strength is 710 [N/mm^2] with tensile elongation up to 40 [%].

Aluminum AlMgSi0, (Hans-Erich Gemmel & Co. GmbH)5

The material have the following alloy components[%] : Mg 0,5 , Si 0,5 , Fe 0,2 .

Its density is equal to 2.7 [kg/m^3], Thermal conductivity at 20 °C is 35 [m/ohmxmm^2]. The Mechanical properties are the following: Yield strength is 230 [N/mm^2], Tensile strength is [N/mm^2] 260 with tensile elongation up to 25 [%].

2.1.3. Mass matrix (Chopra 2012)

Before the model was assembled, each singular element was measured and weighted.

First of all, each element was measured with a digital ruler. The height, width and thickness were taken into account in order to calculate the volume. This kind of measurement is useful in order to build a future FEM Model using commercial software. Reference to the measurements is mentioned in appendix 1.

Afterwards, each single element was weighed by using a mass balance instrument. The results of this measurement are mentioned in appendix 2.

Our box elements are the parts that represent the floor slabs of a multi-story building. Because of their constraining effects, the mass matrix can be simplified. Each of the slabs may be considered as infinitely rigid in the XY PLANE, but flexible along the vertical Z direction. Practically, the XY DOF of all nodes in the floor level are a function of the [3D] DOF of the same nodes in the floor level, because of rigid correlation.

From this explanation, we can determine that the mass matrix is diagonal with:

$$m_{ij} = 0 \; i \neq j \;\; , m_{ij} = m_j \; or \; 0$$

"where mj is the lumped mass associated with the jth translational DOF, and mjj =0 for a rotational DOF " (Chopra 2012).

The total mass of the model equals to 5.774 [Kg], where mass m1 =1.946[Kg], mass m2= 1.926 [Kg] and mass m3 =1.608[Kg]. The computational of the mass matrix is as follows:

$$\text{M [Kg]} = \begin{bmatrix} m1 & 0 & 0 \\ 0 & m2 & 0 \\ 0 & 0 & m3 \end{bmatrix} = \begin{bmatrix} 1.946 & 0 & 0 \\ 0 & 1.926 & 0 \\ 0 & 0 & 1.608 \end{bmatrix}$$

2.1.4. Stiffness uncertainty

The connection between the boxes and the lateral plates are made by manual bolts. This kind of joints can be statically represented as a hinge with rational spring in the upper and lower part of each slab box. Each spring can assume a particular value of stiffness. In further steps we have to deal with this kind of uncertainty. The stiffness value of each node must be known in order to obtain a precise model. This kind of computation will be done through iteration between FEM SAP2000 software and numerical statistical MATLAB software.

2.2. Design and preparation

2.2.1. Measurement devices

PIEZOELECTRIC ACCELEROMETER (PCB Piezotronics) - This instrument is a sensor that generates an electrical output proportional to the applied acceleration. It is designed to measure the vibration and shock in a wide variety of applications with a wide frequency range. It combines a piezoelectric sensing element with a crystalline atomic structure which outputs an electrical charge when subjected to an excitation with quit null deflection. The output charge that occurs instantaneously, makes it ideal for dynamic measurements.

Figure 5 -Acceleometer TLD356A17 (PCB Piezotronics)

After the crystal is stressed, a lead wire transfers the generated electric charge to the electrical connector –kind of plug. The inner particular elements shear configuration made by a seismic mass and a sensing element. Both are mechanically preloaded under a center post that connects them through a thin ring.

The mounting surface has almost no influence on the frequency response due to the stiff characteristic of this sensor. Nevertheless, the mounting technique influences the achievable high frequency measurement range. We chose to use an adhesive mount technique with wax material. It is important to lubricate the mount surface and use a small portion of adhesive material.

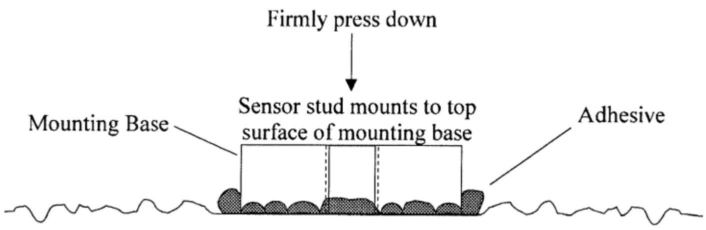


Figure 6-mounting by adhesive material (PCB Piezotronics)

The physical quantity measured and the electrical signal generated by the sensor should necessarily be linked to some degrees of accuracy. Therefore, a calibration procedure should be done. An automatic calibration analysis is provided by the sensor with the documentation of physical parameters, operational limits, electrical characteristics and environmental influences.

Figure 7-accleometer after installation

We use a 3AXIAL 14X20.3X14 [mm³] accelerometer, where the housing is made by an anodized aluminum, and the sensor by ceramic material. Considering its performance, the accelerometer has sensitivity of 51[mV/m/s²] with an error pf 10[%]. It is able to measure a frequency between 0.4-4000[Hz] with a resonant frequency of ≥14 [kHz].

Impact Hammer (PCB Piezotronics) – mechanical dynamical behavior can be determined by striking impulse over a test object with the force instrument hammer. The resultant overall motion derivate as a measuring acceleration by accelerometer can be combined together with the generated impulse. This mutual relationship between action and response will be considered.

The hammer consist of an integral quartz excitation force sensor mounted on the striking end of the hammer head. The impact blow is transferred by the sensor through an electrical signal for display and analysis. The sensor structure consists of rigid quartz crystal and a built in, microelectronic, unity gain amplifier. The cable is connected to the end of the handle for convenience, and in case of a missing hit event it avoids connector damage.

By eliminating hammer resonances in the frequency range of interest, we can ensure that the structural characteristics of the hammer do not affect computational measurement results. This results in more accurate and stable measurements. The hammer is an integral structure with its components connected through laser welding. This characteristic makes its operation reliable in diverse environments.

Figures 8,9-Impact hammer 086C03 and Tipping head (PCB Piezotronics)

The stiff tipping head, transfers the force impact to the sensor and protects the sensor from any damage. Different stiffness tips allow us to vary the pulse width and frequent content of the force. The remaining metallic mass allows the tuning by concentrating more energy at lower frequency .The tip type influences the hammer sensitivity, that in our case can vary from 2.185-2.314 [MV/N].The frequency range of the hammer can also vary based on the tip being used.

Signal Conditioner (HBM)- Is a measurement tool that is used to perform a process of data acquisition. Basically, it converts an electrical or mechanical input signal to an output one. The use of a signal conditioner is essential, because it helps provide precise measurements through accurate data-acquisition process.

The improvement or adopting of the signal is influenced by the Signal Conditioner functionality. Its main propose is to work as a signal convertor. It picks up a signal and converts it into a higher level of electrical signal. The signals generated may need to be converted to be usable for the instruments they are connected too.

In order to achieve a high level of accuracy, certain signal conditioners can a perform linearization process when the signals produced by a sensor do not have a straight-line relationship with the physical measurement. During the sensor calibration the linearization parameters are automatically evaluated.

Other secondary functions of the signal conditioner can be introduced. One of them is a signal amplification. This process requires increasing the signal to noise ratio for processing or digitization. Another functionality of the conditioner is filtering, where the signal frequency spectrum is filtered to only include the valid data and block any noise. The filters can be made from either passive and active components or digital algorithm.

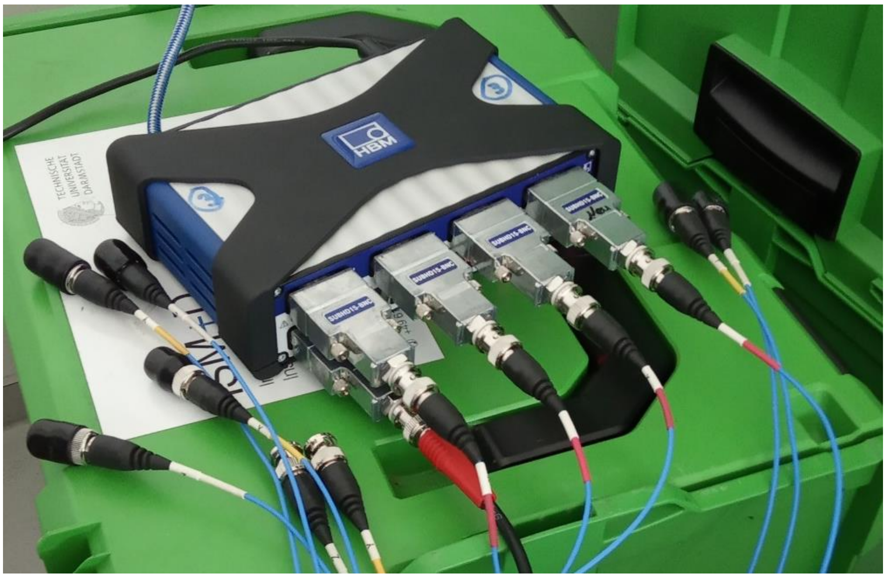


Figure 10-Signal conditioner plugged with Conductors (HBM; PCB Piezotronics)

Other instruments used in the experiment are : 4 cabels condcures –with the porpuse of connecting and transferring the electric signal from to accelereometers to the signal conditioner. A Personal Computer is used for the live data process and implementation .

2.2.2 Experiment arrangement

A special attention about the arrangement of the experiment (figure 11) was paid in order to prevent errors. It was important to document the configuration of each of the accelerometers sensors with one of the conditioner inputs: sensor SN205652 (m1) with input 1, sensor SN2056503 (m2) with input 2, sensor SN207583(m3) with input 3, SN206563 (m base) with input 4. Where the impact hammer was connected to input 5. The mass in the base is not considered in the system DOF, but has the purpose of being a reference for null ground acceleration. In order to prevent ambient influence, each cable was tightly glued to the model by a technical tape.

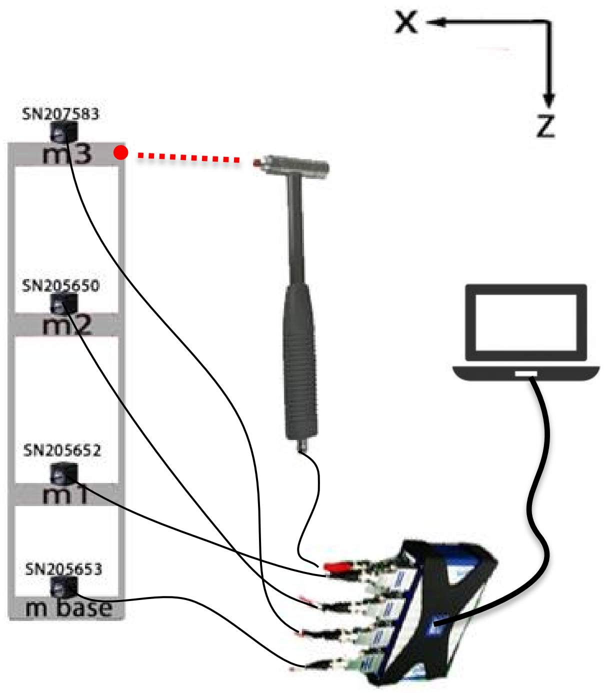


Figure 11-Arrengment of of the instruments

Each accelerometer was mounted with a small piece of wax in the exact center each floor's mass. Moreover, in order to correctly consider the motion along the horizontal X direction, each lateral X side of each sensor was placed with 90 degrees [°] compared to the X axes. The hitting point was in the exact center of the mass of the laterl side of m3(red point).

2.3. Performance

2.3.1. Methodology (PCB Piezotronics)

The principal aim of the experiment is to understand the real motion quantities of the scaled model. As we are dealing with N=3 DOF we are expecting to have 3 stable results in terms of natural cyclic frequencies f_n and modal shapes ϕ_n. These quantities will be used as a true reference for the future computational validation of the numerical models.

Modal Analysis involves fixing the accelerometers to The DOF of interest. By striking the scaled model in one point with the impact hammer, a motion is created. This kind of impulse forces [N] cause a disturbance to the model, and therefore it pulls it out of its static equilibrium position. At the points of interest we can measure an acceleration [m/s²], that its integration yields to velocity compliance, impedance and mobility. The hammer impulse consists of a nearly constant force over a broad frequent spectrum, and is therefore capable of exciting all resonances in that range. Naturally, the force impulse amplitude and frequency (wave shape) are function of the impact hammer's physical properties and the velocity at the impact time. The impact tip affect the hammer impulse frequency content and energy level that are both interrelated. Because we are dealing with a test light weight structure, a stiffer tip will be used for higher frequency results.

Before starting with the experiment, all instruments must be installed and operating specifically according to their manual instructions (references (PCB Piezotronics)). Ensuring a well electric contact is fundamental.

Testing and calibration may be performed by tailoring the frequency bandwidth of the forces. This can be done by applying some sample impacts and process the results. Considering the hammer frequency content, we can check if the force spectrum is sufficient to cover the structural resonances present in the acceleration spectrum. "often signal energy is sufficient to excite structural resonances at 20 [dB] below initial low frequency force level (PCB Piezotronics)". Regarding the accelerometers, an automatic calibration process is basically done under normal conditions.

The measurement experiment was executed 2 times for a total time of 15 [min], where the acceleration of each mass in [m/s²] and the impact of the hammer in [N] was being measured. The data acquisition interval was fixed for a value of 0.833[ms]. A special consideration about the high sensitivity of the accelerometers was taken into account. Soft impact hits were applied in order to not exceed the potential registration spectrum. Not any kind of motion in the surrounding area was allowed.

The Execution of the experiment is illustrated in figure 12.

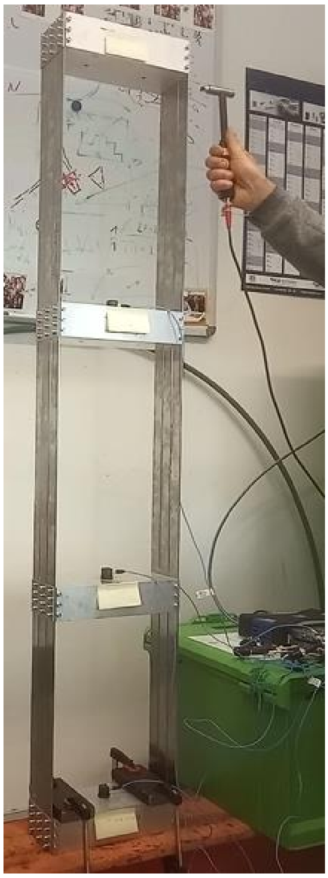


Figure 12-Execution of the experiment

2.3.2. Data collection (HBM)

For this kind of registration we used Catman AP (HBM) to have a significant representation of the signals. This kind of commercial software allows as to visualize the sensors acceleration [m/s²] versus the time [s], and the impact converted impulse [V] versus the time [s], in live. The experiment was monitored by pre-defined boundaries of spectrum frequency and noises that in case of exceedance alerted the system interface.

Because more than 2 milions of registaration points were took into account for the modal analysis, this documantation will not be rapresented graphically in this thesis but as a external registration file. The following is an example :

Figure 13- Acceleration versus time

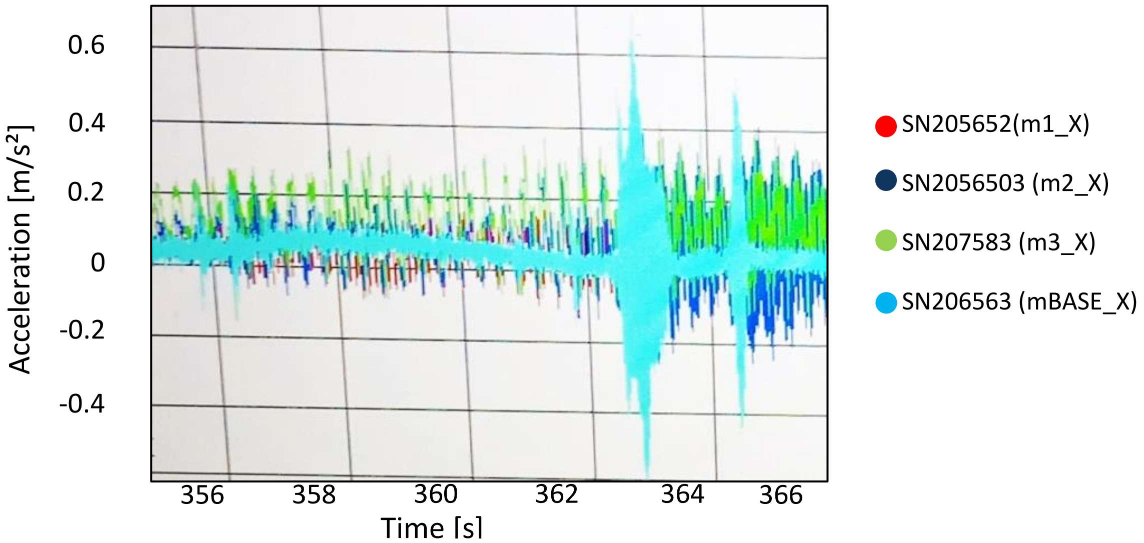
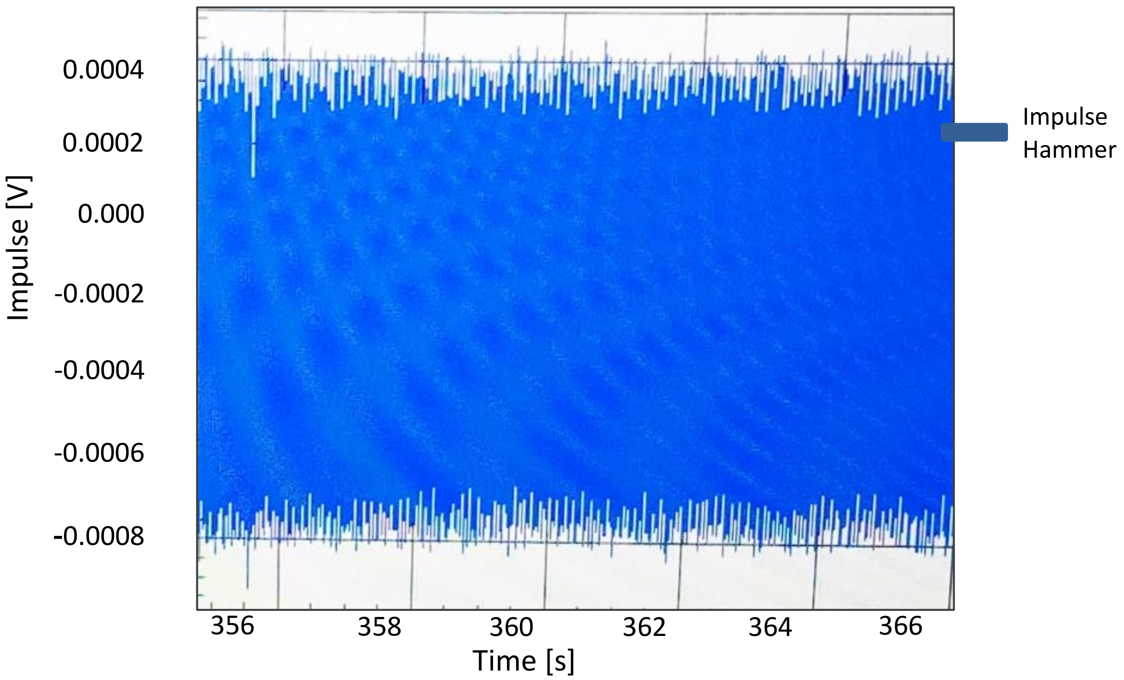


Figure 14- Impulse versus time



3. Evaluation of the Computational Data (REYNDERS, SCHEVENELS and DE ROECK; MACEC)

3.1. Modal analysis of the measurement data

3.1.1. Introduction of MACEC

After performing the experiment and the data collection process described in paragraph 2, we could have performed dynamic modal analysis using the numerical computation commercial software MATLAB. This software is an independent platform and calculates with matrices, which solves and visualizes mathematical problems. Because programming in MATLAB is time-consuming; we used a mini software–toolbox based on MATLAB.

MACEC is a modal analysis toolbox based on MATLAB, developed by the researches of the Civil Engineering Department at KU Leuven University. MACEC deals with every step of the modal analysis procedure, except for the data collection. After inserting raw measurement data as an input, MACEC can apply different functions for the visualization and processing of the data, the identification of system models and the determination and visualization of the structure's modal characteristics. Those are easy to handle by using a particular graphical user interface (GUI).

Our goal is to perform an operational modal analysis using the different methods that are available in MACEC.

3.1.2. Geometry definition

MACEC requires the definitions of a simplified geometry. As a first step, we constructed a grid of nodes that connects to the beams for visualization. Because we deal with acceleration in only one direction in a rigid structure, we can assume that the displacement and its derivatives at each point of the same floor will be equal. Therefore, our frame can be idealized into a spine (1 continuous vertical beam) structure with a concentrated mass that represents the slabs' masses.

We define ASCII file (grid.asc) that contains the definition of a measurement node. Each row has four columns, containing the node number and it's (X, Y, Z) coordinates, as represented in table 1

NODE NUMBER/COORDINATE	x	Y	Z
1	0	0	401
2	0	0	903
3	0	0	1404
4	0	0	0

Table 1 –definition of ASCII grid

After that the measurement grid has been defined, the only geometrical information that is missing is the links between the measurement points. These links are defining for the visualization purposes. A link of ASCII FILE (link.asc) of "beam" type will be used. Practically, it allowed us to connect each node on a line. Each row of the ASCII file contains two nodes that are connected by a beam (figure 15).

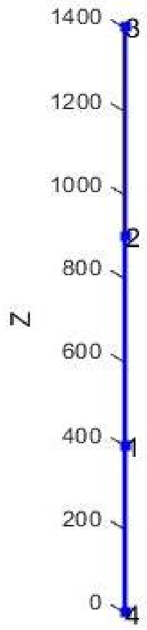


Figure 15-Idealized spine model (MACEC)

3.1.3. Processing the measured signals

As a following step of the analysis, we have to adopt the raw measurement data to the defined geometry. Our raw data text file generated by CATMAN AP may be adopted into a MATLAB DATA file (input accelerations. mat) by elementary steps, to use it in MACEC. This kind of file is basically a [1105700x5] double format matrix that contains the accretions measured for each defined sensor (the chronological order is the same as before).

The data consist of 1105700 time samples, obtained at a sampling frequency of 1200 Hz.

The matrix file must be converted to mcsignal (multi-channel signal) variables file by a simple command given in MACEC. This kind of variables contains all the information about the measured data in one single MATLAB variable. Among the rest mcsignal objects are : the number of samples measured "N", the sampling frequency "F" [Hz], the frequency resolution "df" [Hz], the total measurement duration "T"[s], the Period "dt" [s] and the proper quantity relate to each sensor "quantity"[-].

In order to complete the conversion into mcsignal, the sensors and the impact hammer must be defined. we were defined an ASCII file that contain string characters ,with information relate to 5 channels .each channel consists of 5 or more columns which contain the sensor type, the sensor number, the manufacturer type, the serial number and the sensitivity. We have defined 4 accelerometers and 1 force sensor .Because the measurement data have the physical meaning of accelerations, the measurement units, the sensitivity and the amplification fields are defining by their default values.

The ASCII converted file (input accelerations_conv.mat) is shown in figure 16, as seen in the MACEC software interface.

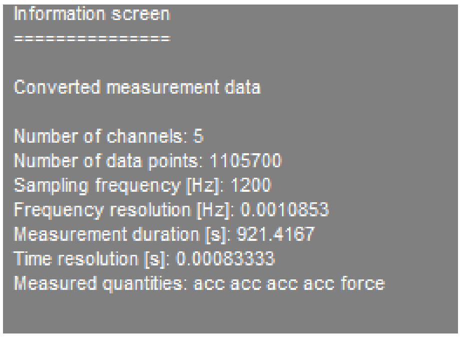


Figure 16-mcsiganl converted ASCII file in MACEC (MACEC)

The next step is the actual processing of the measurement signals. The simulated measurement data can be visualized and observed by the representation of the time history and the frequency connection of the different signals. In this step, I found it important to make an interpretation of the signals time history, as represented below:

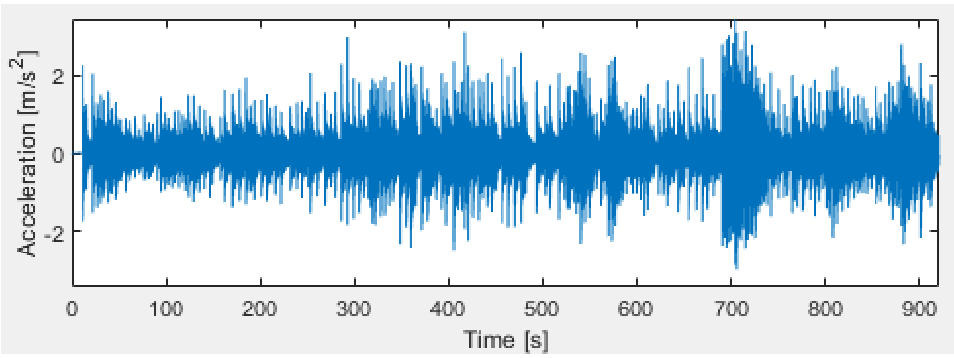


Figure 17-Time history of sensor 1(mass m1) (MACEC)

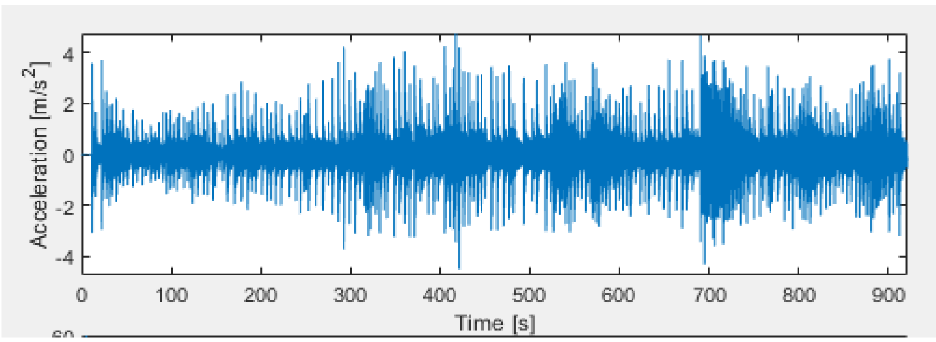


Figure 18-Time history of sensor 2(mass m2) (MACEC)

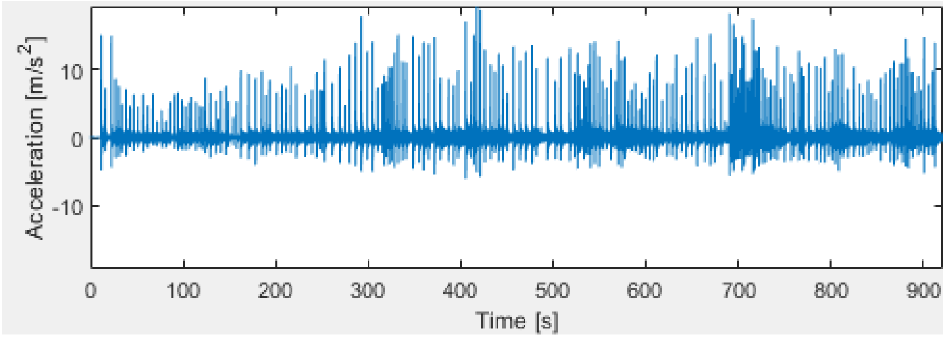


Figure 19-Time history of sensor 3(mass m3) (MACEC)

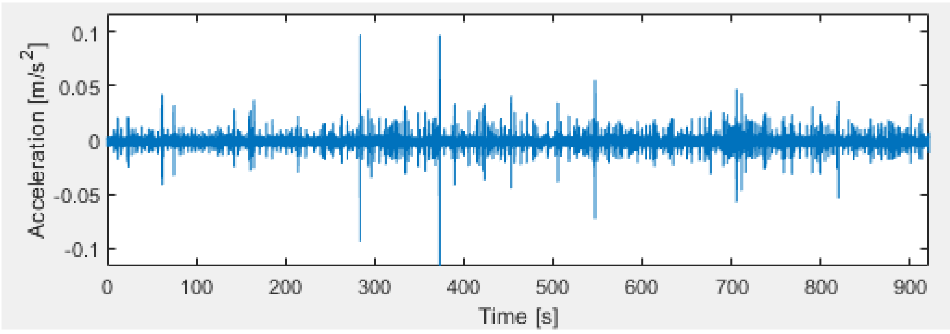


Figure 20-Time history of sensor 4(base reference mass) (MACEC)

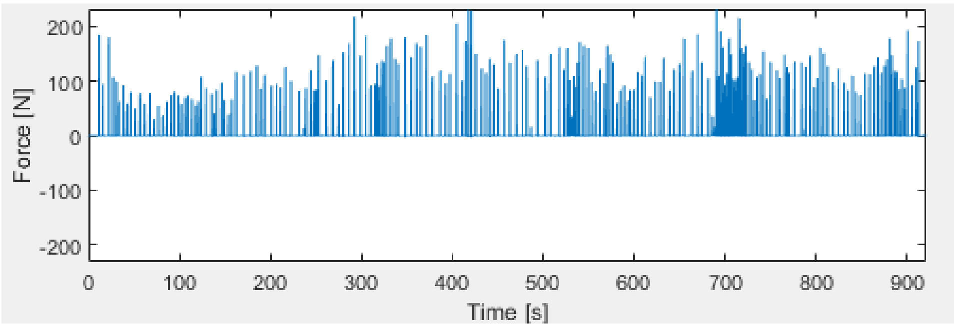


Figure 21 -Time history of sensor 5(Impact Hammer) (MACEC)

The time history graphs above represent physical dynamic behavior with logical evolution during all the 15 minutes of the experiment. According to the graphs regarding the masses sensors (1-4), each mass undergoes an acceleration [m/s²] with a distribution inside the ranges of [2,-2], [4,-4], [10,-5] and [0.05,-0/05]. As we expected, the upper mass m3 undergoes a higher static force comparing to the lower mass m1. Therefore m3 is subjected to a higher acceleration. The referring bottom mass as quit null acceleration and can be considered inside a range of tolerance. That brings us to verify that the boundaries' conditions are well defined, and there is a minor influence on the experiment by the surrounding environment. The positive Impact hammer time history, demonstrates a well behavior too.

The last step before the approximation of the system identification is the coupling between the measurement nodes defined in the grid file and the measurement channels. For each node, we have to adopt the correlated sensor and it's DOF.

Moreover, the measurement direction must be defined by the azimuth and elevation angles.

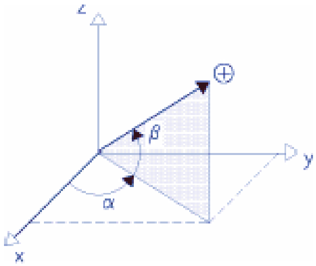


Figure 22- Definition of the direction angels (REYNDERS, SCHEVENELS and DE ROECK)

This procedure done in the ASCII file (input_acceleration_proc.mat) as the following:

Channel	Node	DOF	Azimuth (°)	Elevation (°)
1	1	X	0	0
2	2	X	0	0
3	3	X	0	0
4	4	X	0	0
5	3	X	0	0

Table 2- Coupling between the measurement nodes and channels (MACEC)

3.2. System Identification

3.2.1. Stochastic subspace identification (SSI) method (PEETERS, B. AND ROECK, G. D.)

When performing vibration a test, the use of artificial external force (shakers, drop weights) is not always convenient because of the high cost and other practical reasons. On the contrary, the ambient excitation is freely available (traffic, trains) but it brings out other challenges. The ambient input is difficult to approximate, and the system identification algorithms have to deal with output-only measurements. In other words, we know the dynamic effect on the structure but its trigger remains mathematically unknown.

This method solves the civil dynamics problems, where developed data driven from stochastic subspace algorithms avoids the computation output covariance. These algorithms are based on the approximation of the row space of the future outputs into the row space of the past outputs. One disadvantage of this procedure is that not all degrees of freedom can be measured together, but that they are divided into several set-ups with overlapping reference sensors. These reference sensors are the principal key for the approximation of the global modal shapes.

We can consider the differential equation of motion with n_2 masses. As referred in paragraph 1.3.2

$$M\{\ddot{U}(t)\} + C\{\dot{U}(t)\} + K\{U(t)\} = F(t) = B2\{u(t)\}$$

Where M,C,K$\in \mathbb{R}^{n2\times n2}$ are the mass, damping and stiffness matrices , F(t) $\in \mathbb{R}^{n_2\times 1}$ is the excitation force, and U(t) $\in \mathbb{R}^{n_2\times 1}$ is the displacment vector at continous time t. F(t) can be factorized into a matrix $B_2 \in \mathbb{R}^{n_2\times m}$ that describes the inputs in space and a vector u(t) $\in \mathbb{R}^{m\times 1}$ describes the m inputs in time.

The above equation cannot precisely describe the behavior of dynamic structure in case of system identification, because its continuous properties do not fit well into a point sampling measurements. Moreover, not all DOF can be measured and some noise term needs to be taken into account as well. A more suitable formula may be used:

$$\dot{x}(t) = A_C\{x(t)\} + B_c u(t)$$

Where

$$x(t) = \begin{pmatrix} U(t) \\ \dot{U}(t) \end{pmatrix}, \quad A_C = \begin{pmatrix} 0 & I_{n2} \\ -M^{-1}K & -M^{-1}C \end{pmatrix}, \quad B_C = \begin{pmatrix} 0 \\ -M^{-1}B_2 \end{pmatrix}$$

$A_C \in \mathbb{R}^{n\times n}$ is the matrix state ($n=2n_2$), $B_C \in \mathbb{R}^{n\times M}$ is the input matrix and x(t) $\in \mathbb{R}^{n\times 1}$ is the state vector. If not all DOF is considered and that the measurment is done with "l" singal sensors , the problem can be written into :

$$y(t) = Ca\{\ddot{U}(t)\} + Cv\{\dot{U}(t)\} + Cd\{U(t)\}$$

Where y(t) $\in \mathbb{R}^{1\times 1}$ are the outputs, and Cd,Cv,Ca $\in \mathbb{R}^{1\times n_2}$ are the outputs matrices for displacment, velocity and acceleration as defined :

$$C=[Cd\text{-}CaM^{-1}K \;\; Cv\text{-}CaM^{-1}C_2] \text{ and } D=CaM^{-1}B_2$$

The equaition of motion can be transmitted into :

$$y(t) = Cx(t) + Du(t)$$

Where $C \in \mathbb{R}^{1\times n}$ is the output matrix and $D \in \mathbb{R}^{1\times m}$ is the direct transmission matrix. This equation is still continuous time deterministic. Continuous means that each expression can be evaluated at any desire time t, and deterministic mean that the input-output quantities u(t) and y(t) can be measured exactly without any noise (error).

Once again, this assumption must be modified in order to describe the behavior precisely, because the sample time and noise will influence the analysis. Our measurements were evaluated at some discrete time instants k Δt where k$\in \mathbb{N}$.

$$X_{k+1} = AX_k + Bu_k$$
$$y_k = CX_k + Du_k$$

Where Xk=x(k $\triangle$ t) is the discrete time vector, A=$e^{(Ac\Delta t)}$ is the discrete matrix and B=[A-1] Ac^{-1}Bc is the discrete input matrix. In this way the discrete time interval is taken into account. In order to consider the noise (the stochastic component), we assume that Ac is not invertible. For this reason, B must be modified into a different term.

The discrete-time combined deterministic-stochastic state-space model is obtained:

$$Xk+1 \ = AXk \ + \ Buk + wk$$
$$yk = CXk \ + \ Duk + vk$$

Where wk ∈ $\mathbb{R}^{\mathrm{n}\times 1}$is represents the noise due to the model inaccuracy, and vk ∈ $\mathbb{R}^{1\times 1}$ refers to the measurement noise due to sensor inaccuracy. Both noises are not physical quantities and therefore cannot be measured, But we can assume that they are white (with equal intensity at different frequencies) and with null meaning E=0, with E as the mean operator. If the input contains some dominant frequency components in addition to white, the white noise assumption is violated. These kind of frequency components cannot be separated from the Eigen frequencies *fn* of the system and they will appear as poles of the discrete matrix *A.*

We would like to obtain better stochastic state-space. We can define new properties for our model:

$$\Sigma = A\Sigma A^T + Q, \mathrm{Ao} = C\Sigma C^T + R \ \ , \mathrm{G} = A\Sigma C^T + S$$

Therefore , we can determine that "the output covariance's can be considered as impulse responses of the deterministic linear time invariant system A,G,C,Ao (PEETERS, B. AND ROECK, G. D.)", in the following equation:

$$\mathrm{Ai} = CA^{i-1}G$$

A Typical problem for structures is that not all outputs can be measured at the same time, but that they are classified into several set-ups with overlapping sensors. Some reference sensors can be placed at optimal locations on the structure, where we can completely measure the dynamic responses in terms of modal shapes.

Assuming that the "l" sensors of the outputs are arranged to have the" *r"* references first

$$yk = \begin{pmatrix} yk^{ref} \\ yk^{\sim ref} \end{pmatrix} \quad \text{where} \quad yk^{ref} = Lyk\,, \ \ \mathrm{L=[l,0]}$$

yk^{ref} ∈ $\mathbb{R}^{\mathrm{r}\times 1}$ are the reference outputs and $yk^{\sim ref}$ ∈ $\mathbb{R}^{(\mathrm{l-r})\times 1}$ are the remaining. L∈ $\mathbb{R}^{\mathrm{r}\times 1}$ is the selection matrix that clasifies the references.

Our model can be seen as :

$$\mathrm{Ai}^{ref} = CA^{i-1}\mathrm{G}^{ref} \ \text{, with } \mathrm{Ai}^{ref} = AiL^T \ \text{ and } \ \mathrm{G}^{ref} = GL^T$$

After the distinction between "normal" and "referne" outputs measurments were defined, the whole data should be organized Inside one unique operator - the Handel Matrix. The Handel Matrix is a "2i" block rows and" j" columns. The first "i "blocks have "r" rows, the last "l" have "l" rows. This matrix is an important tool in the SSI method, because it can split into a past reference and a future part.

The Hankel matrix is the following:

$$H = \frac{1}{\sqrt{\mathrm{J}}} = \left(\begin{array}{cccc} yo^{ref} & y1^{ref} & \dots & yj-1^{ref} \\ y1^{ref} & y2^{ref} & \dots & yj^{ref} \\ \dots & \dots & \dots & \dots \\ yi-1^{ref} & yi^{ref} & \dots & yi+j-2^{ref} \\ \hline yi & yi+1 & \dots & yi+j-1 \\ yi+1 & yi+2 & \dots & yi+j \\ \dots & \dots & \dots & \dots \\ y2i+1 & y2i & \dots & y2i+j-2 \end{array} \right) = \begin{pmatrix} Yp^{ref} \\ Yf \end{pmatrix} \begin{array}{c} \updownarrow ri \\ \updownarrow li \end{array} \frac{\text{"}past\text{"}}{\text{"}future\text{"}} \in \mathbb{R}^{(\mathrm{l+r})\mathrm{i} \times \mathrm{j}}$$

Where p and f refer to the past and the future. The matrixes Yf and Yp^{ref} are defined by splitting H into two parts of i block rows.

3.2.2. Operation of SSI method in MACEC

We will use the data-driven stochastic subspace identification (SSI-data) method for the identification of our system.

As this method is based on the approximation of the inputs by knowing the outputs, the first step is to tell our system where each sensor is related to. We define our input as channel 5 of the impact hammer, and as outputs the channels 1-4 of the accelerometers.

The second step is the construction of the Hankel matrix. The half number of rows "i" or the minimal value for i in the SSI-data algorithm, is an important information for the system estimation. Higher "i" is, probably for more accurate estimation of our system. MACEC calculates this number automatically by considering a higher number then twice the DOF-the system expected order.

$$expected\ system\ order = \frac{sampeling\ frequency}{2 \times fo} = 30 \gg 60$$

With f0=20[Hz], the assembly of Henkel matrix and the QR Factorization can be completed.

The third step is to estimate the real system order by looking at the singular values graph calculated from the last step. Knowing that the system order is equal to number of non-zero singular values.

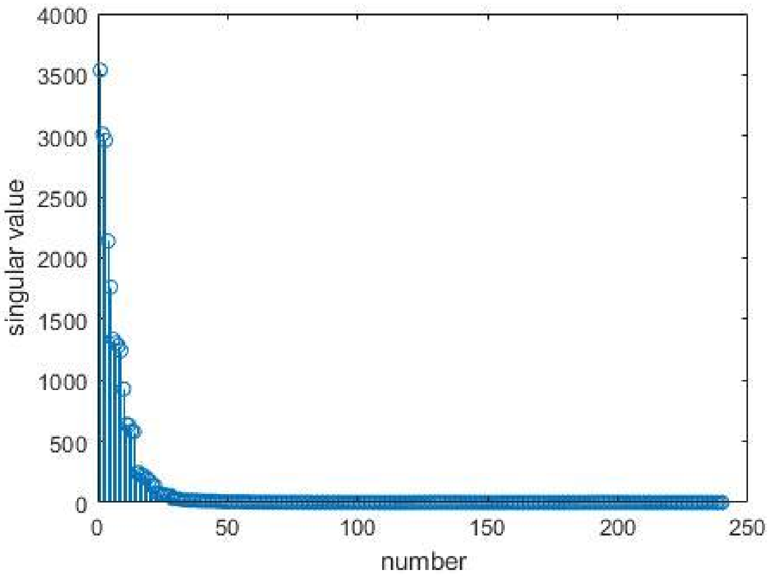


Figure 22- singular values in decreasing order of magnitude (MACEC)

As can be seen in figure 22, the first 20 singular values are much bigger than the other ones, which indicates that a model order of 20 can well describe the dynamic behavior of our model. The other singular values are not exactly zero, but it can be seen that choosing a model order higher than 20 scarcely influences the dynamics of the identified system. The calculation of the system matrices can be completely done by inserting 2:2:20 as system orders.

The system identification was completed, and a MATLAB file (identified_system.mat) that contains the dynamic characteristics was created: vector of the Eigen frequencies, vector of the model orders, vector of the damping ratios, a matrix of the mode shapes, vector of the transfer norms, vector of the mean phases, vector of the modal phase collinearities and vector of the mean phase deviations.

3.3. Results

3.3.1. Modal analysis of the identified system models

After the identification of the system, MACEC brings us the option of automatic calculation of the modal parameters and a creation of a stabilization diagram. This diagram represents frequency [Hz] versus the model order [-] the all possibilities of the system modes, and basically among all system modes we have to choose the real physical one that are more stable. This can be done by the assistance of the PSD function that are calculation returns as the stabilization diagram with a continuous function that indicates the stable modes (figure 23).

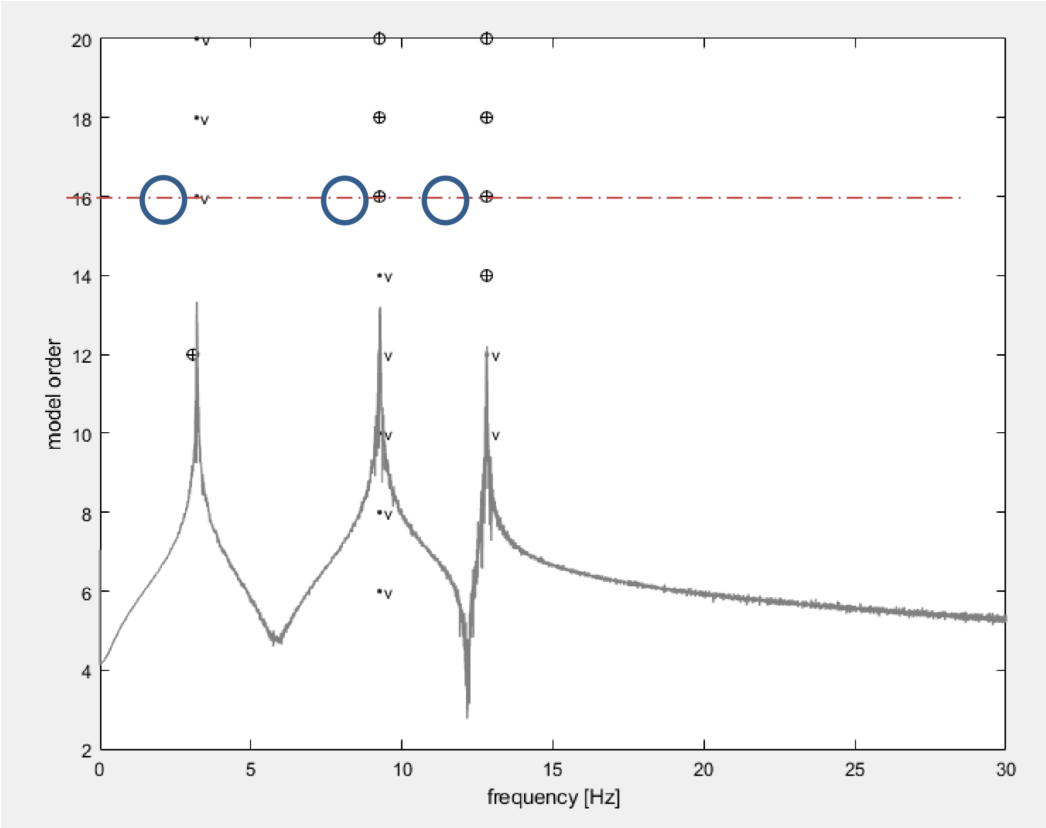


Figure 23- stabilozation diagram with PSD function

The complete stabilization representation IS a 3D space with an additional axes of the modes intensity and with a bandwidth frequency between 0-600 [Hz]. Because we are interested in the most relevant modes, we represent the bandwidth between 0-20 [HZ] in 2D diagram.

The peaks values of the PSD function indicate us the concentration of the real modes we are looking for. As seen in figure 16, 3 peaks are represented. Then it is enough to choose a value of model order that tends to coincide with the PSD function. We will choose the order 16 as a reference (red dashed line).Therefore, 3 point were choose as the real modes for our model (blue circles). For verification, MACEC is able to represent them in the complex space as a function of the 4 outputs sensors inserted (this interpretation is out of this work aims).

The operational modal analysis results can be represented graphically by the plotting of the 3 modal shapes in the real XZ plane.

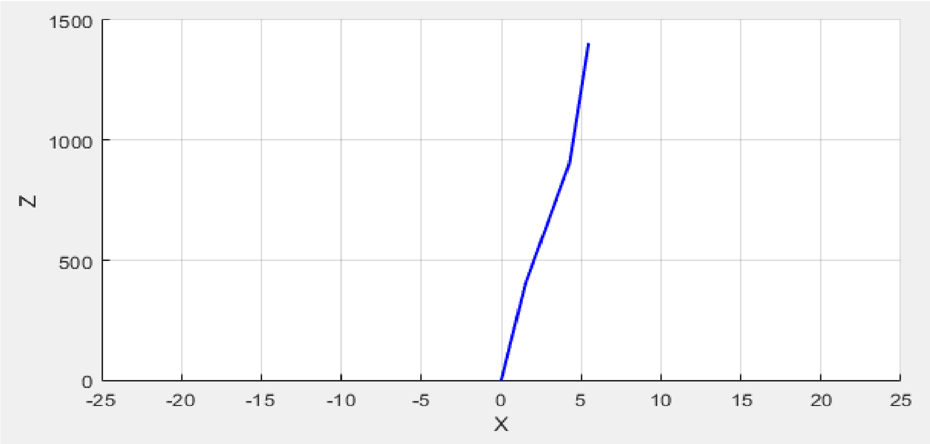


Figure 24- mode 1 with Eigen frequency 3.2 [Hz] (MACEC)

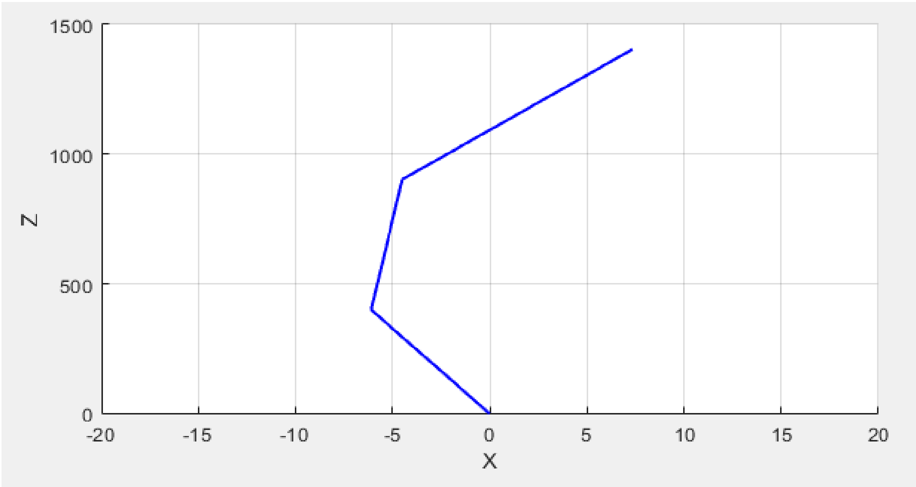


Figure 25- mode 2 with Eigen frequency 9.26 [Hz]

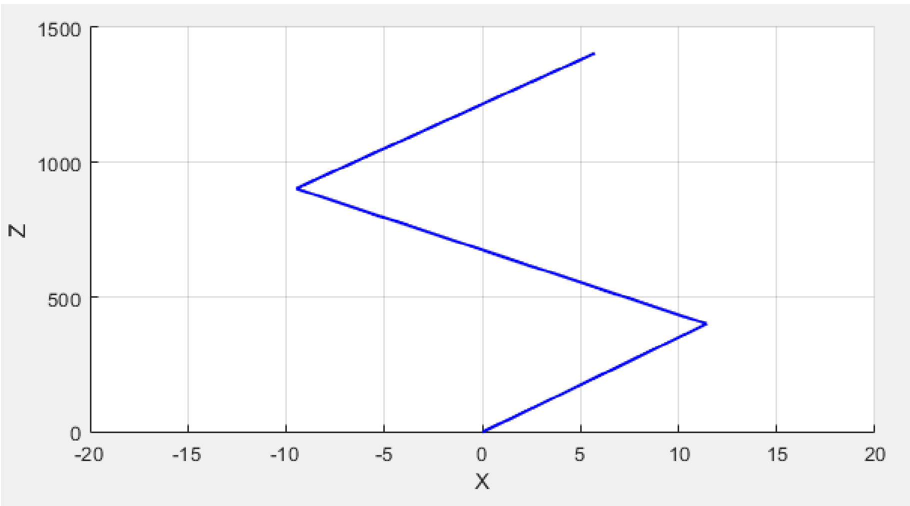


Figure 26- mode 3 with Eigen frequency 12.81 [Hz]

The results of the identified modal parameters are shown numerically in tables 3 and 4 :

	Mode 1	**Mode 2**	**Mode 3**
Eigen frequencies [Hz]	3.20491	9.25938	12.8051
Damping ratios [%]	0.17913	0.15259	0.1597
Modal phase collinearities [-]	0.99994	1	0.99996
Mean phases [°]	-8.9794	4.5141	-2.5569
Mean Phase Deviations [°]	0.24021	0.052185	0.19747

Table 3-numerical results of the modal analysis (MACEC)

	Mode Shapes		
	Mode 1	**Mode 2**	**Mode 3**
Third Floor	0.20714-0.03217i	-0.16793-0.013305i	-0.088723+0.0044806i
Second Floor	0.16112-0.026321i	0.10282+0.0082728i	0.14714-0.0069119i
First Floor	0.057014-0.0086096i	0.13981+0.010867i	-0.17728+0.007374i
Base Floor	6.1349e-05-5.1521e-06i	8.7663e-05+4.0608e-06	-0.00013568+3.113e-05i

Table 4-numerical results of the modal analysis (MACEC)

3.3.2. Interpretation and conclusion

The mentioned results can be considered as the real ones, and well representing the dynamic behavior of our scaled structure. 3 real stable modal shapes were obtained that fit the pre-assumption for the 3 DOF systems.

The application of the SSI Method guaranteed us the consideration of the noise measurement. In the analysis procedure, the white noise was not violated because of its consideration as a part of the natural Eigen frequencies. Therefore, the real stochastic influence was taken into account in our calculations.

The stability of the system was well considered. That was done by the selection of the modes that were demonstrated to be the closest to the physical state, among of the all the other possibilities. That was done with the indication of the PSD function.

Further validation step can be done with the observation of the mean phase angle results. When real modes are expected, the value of this angle is null. All 3 modes have MPA angle less than 1[°], very close to the expected values. For this reason we can be sure about the approximation of the 3 modes.

4. Numerical Model as a Schematic Representation

4.1. FEM model (Computers and Structures Inc.)

4.1.1. SAP2000 and Finite Element Method

SAP2000 is a FEM commercial software designed for civil engineering, produced by Computers and Structures Inc. of Walnut Creek, California. It has very versatile features and the modeling, analysis and verification phases are integrated in a single graphic object-oriented environment. It is able to perform linear analyzes in all the various modes: static, modal dynamics with response spectrum and time history.

Finite Element Method (FEM) is a numerical method, based on stress analysis in a continuous mechanical model that consists of solving stress-strain problems inside the domain of the problem. During the solution, we can use analytical methods and produce close form solutions. The region of the global problem is divided into a number of discrete elements, what allows us to provide a physical approximation of the displacement and stress. The main problem is that we cannot compute every physical property in any point, therefore we define some nodes of interaction for the elements, where the equations may be solved there in terms of displacement. After the displacement in the nodes was calculated, we could derive the strain and stress as second quantities. Boundary conditions must be applied in order to solve the global equations that represent the approximate solution.

4.1.2. Model design

The measurements about the mass and volume, the information about the structural static scheme and the materials properties, together brings us enough knowledge for the design of the scaled model in SAP2000. The procedure was divide into elementary phases, until the model was completed. The unit of length was defined as millimeters [mm] and the unit of force in newton [N]. The axes convention was chosen as: Z(3) axes positive in the vertical direction towards the sky (opposite from the gravity), X(1) is the principal horizontal direction of motion and Y(2) is the inner direction that is fix and does not permit motion .

We started with the choosing of the template type. As we are dealing with a 3D frame, the most convenient template to use is a grid type. The grid can be defined by 3 stories and their height [mm], width of x and y direction. The grid consists of basically of 2 elementary objects: joints and frames, that can be implement mechanically afterwards (for example as a beams, shells or planes).

After giving the definition of a grid we need to define our materials: spring steel and aluminum (paragraph 2). The mechanical properties that were considered are young modulus E, poison coefficient V and density.

The first structural element that was designed is the "lateral plate "made by spring steels. These elements are beams/frame type. They represent the flexible later beams that permit the motion of the structure.

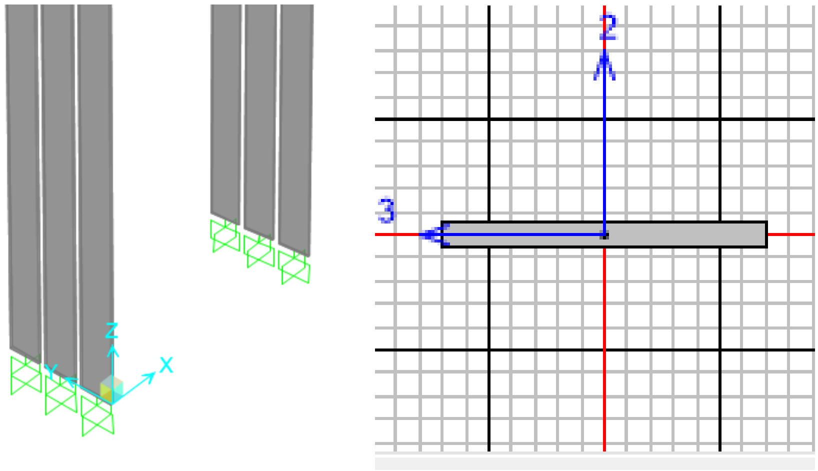


Figure 27-Lateral plates elements in 3D (left) and cross section(right) (Computers and Structures Inc.)

The second element is the aluminum "box" and represents the rigid floors of the model (figure 28). In order to obtain a structural behavior as close as possible to the reality, this element was defined not as a beam with tube cross section, but as a close surface composed by 6 thick shells. In this way the box will not deform when excitation applied along X, but will translate as a rigid body.

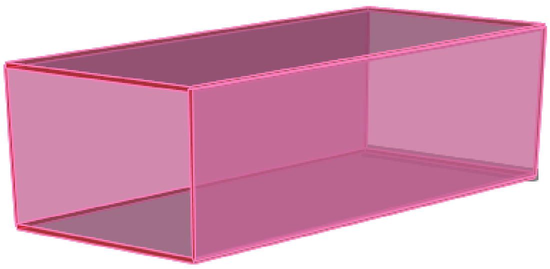
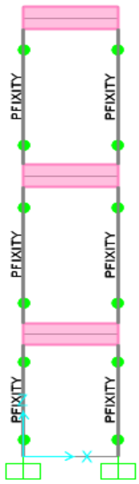


Figure 28-Box element compused by 6 thick shells (Computers and Structures Inc.)

Additional fictitious masses without structural importance were added to the floors in order to precisely describe the weight of the model. The 6 bottom plates beam were fixed to the ground in all directions (figure 22), they are the only restrained joints in our model.

Consider the static scheme represent in figure 29. In order to approximate the real connection between the boxes and the lateral plates, partially fixed (PFIXITY) hinges were assigned to the lateral beams (figure 29). These kinds of hinges are equivalent to the theoretical rotational springs and also have the same dimension of stiffness. The hinges are fully released when considering the axial load along X, shear along Y and Z, bending moment around Z and torsional moment around X. They partially restrict the bending moment around Y with a value of $1{*}e^{18}\ [\frac{N*mm}{rad}]$.



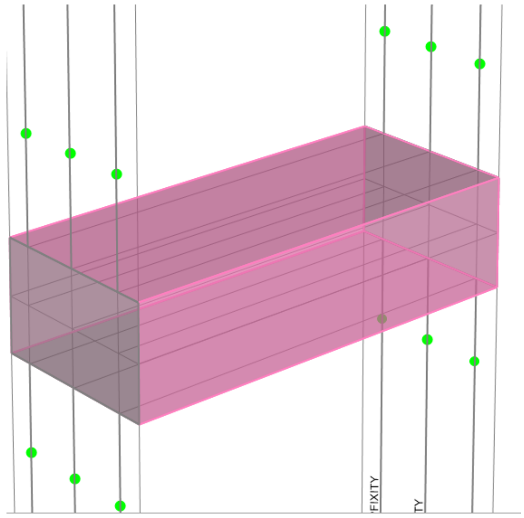


Figure 29-Static scheme in XZ plane (left) and Partial Fixity hinges (right) (Computers and Structures Inc.)

After the assignment of the hinges the model is done. We define SAP2000 using a "plane frame XZ analysis" (for faster computattion) with available DOF considering the translation on X (UX), translation on Z(UZ) and rotation on Y(RY). Complete rapresentation of the model shows in appendix 3 .

4.1.3. Static Analysis

In the dynamic analysis we consider the mass properties of the structure as well. Therefore, a dead weight verification should be done. We will perform static analysis in order to check that the mass matrix of our numerical model is coincide with the real one. The mass matrix computation that was written in paragraph 2.1.3 demonstrates that the total mass of the structure is 5.773[Kg].

As a structural output of the static analysis results, the total reactions on the bases can be shown. The structural purpose of the bases is to transfer the loads to the foundations; therefore the total weight of the structure must coincide with the total vertical reaction (FZ) on the bases.

$$GlobalFZ\ Normal = 56.62[N]\ \ , divid\ by\ g = 9.8[m/s^2]\ ,56.62/9.8 = \ 5.77[kg]$$

The mentioned computation demonstrates that the mass properties of our numerical model fits to the realty, so we can proceed with the dynamic analysis.

4.1.4. Dynamic analysis

After the verification of the mass properties, we are ready to evaluate the dynamic properties. SAP2000 provide "Eigenvector" specific load case, in which the analysis determines the un damped free-vibration mode shapes and frequencies of the system. The modal analysis is considered as linear, and the modal load case may be based on the stiffness of the complete unstressed structure. Moreover, the mass used for the modal analysis is changing automatically while multiple mass sources have been defined. This kind of linear sources can have the same origin as the nonlinear ones (used for the calculation of the stiffness).

For an optimal process, we have to define the maximum and minium number of modes to be found. SAP200 will not exceed this kind of specification. This number includes any static correction mode requesed. The software may calculte less modes than the requested in case of fewer available DOF masses. The defintion of DOF masses is specific just the ones that undergo translation or rotational moment of inertia, and may be assigned directly to the joints or other connected elements. Among all modes,only the stable physical ones wil be rapresented in the interface.

The restriction of a particular frequency range has significant importance. Some modes with zero frequency can be represented in case of unloading of an unstable structure. This kind of error may correspond to a rigid- body motion of an inadequately supported structure, or because of some unstable mechanism that appear during the analysis. This error can be reduced by a specification of the frequency range.

SAP2000 using an accelerated sub -space iteration algorithm (Wilson and Tetsuji 1983), for the computation of the eigenvalue- eigenvectors pairs. During the solution phase, the software input the Eigen values after each iteration .The convergence eigenvalues removed from the sub space and new approximate vectors are introduced. The relative convergence specification was specified for a value of: $tol = 10^{-9}$. Baisically, the iteration for a particular mode will proceed until the relative change in the eigenvalue between the sucessive iterations is less than 2 times tol.

After we understood how SAP2000 executes the modal analysis, the results will be shown graphically and numerically.

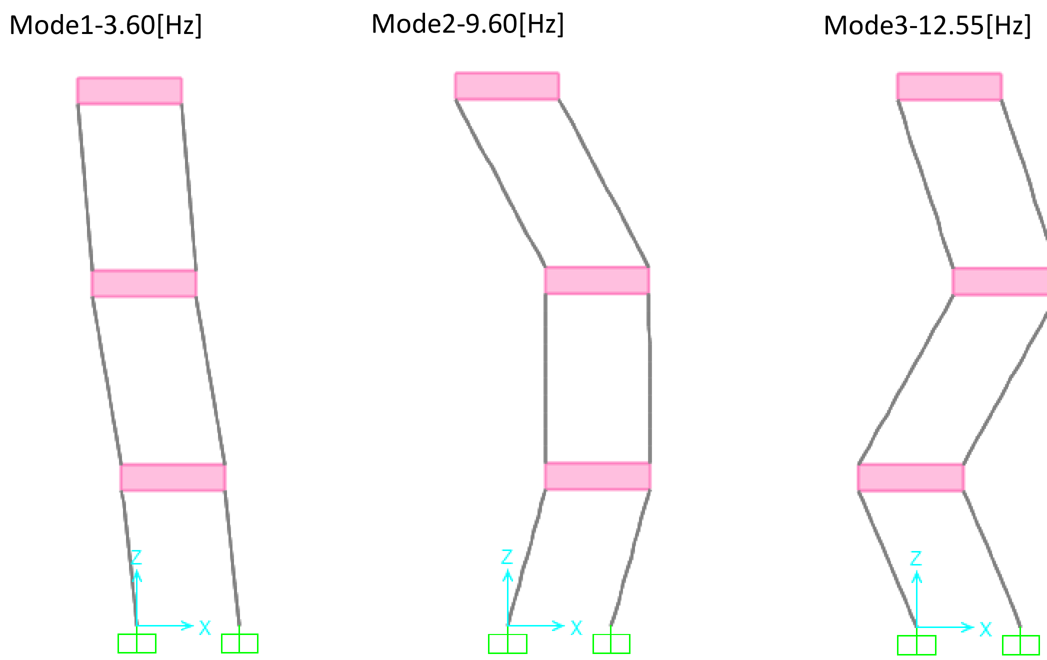


Figure 30-Modal shapes of SAP2000 MODEL (Computers and Structures Inc.)

We can cosider them as deformed shapes. It can be deduced by observation that the floors masses do not deform as the later beams, but are translated as a rigid body. This behaviour is realistic and rapresents the real motion we expected.

Regarding the modal shapes, all of them rapresent a well dynamic behaviour. The first mode has an error of 9.7[%] comparing to the real result obtained in the experiment. Morover,the motion polarity (+/-) is the same as in the experiment. For the second and the third modes the errors are 3.5[%] and 2[%], and they both have the same polarity motion comparing to the experiment modes.

The results are shown numerically in table 5:

	Mode Shapes		
	Mode 1	**Mode 2**	**Mode 3**
Third Floor	-19.2	-16.99	-6.12
Second Floor	-14.54	12.2	11.8
First Floor	-4.899	12.29	-18.73
Base Floor	0	0	0

Table 5 (Computers and Structures Inc.)

4.2. Matlab model (MathWorks)

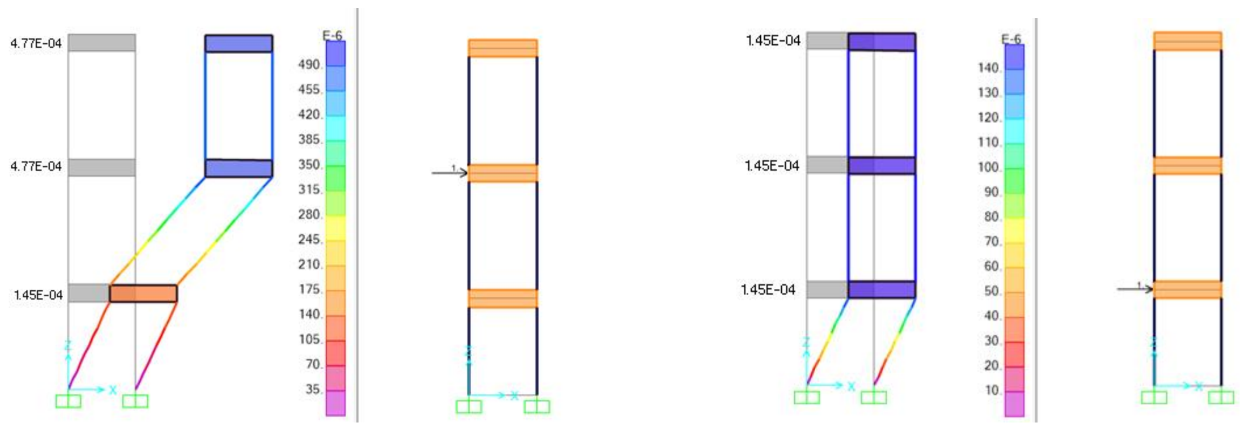
4.2.1. Determination of displacement and stiffness matrix

After the FEM was completed, we are able to use the application to the theory of elasticity represented in paragraph 1.3.5.

Practically, when a unitary forces is applied on a normal structure, their relative displacements are equal to the inverse of the stiffness's relate to each node of DOF. An easy computation of the stiffness matrix [K] can be proceeded where the displacement matrix is known [U].

$$[K] = [U]^{-1}$$

In SAP2000 we applied a unitary concentrated force in the X direction with magnitude of 1 [N]. The application was proceeded considering the center of mass of each story.



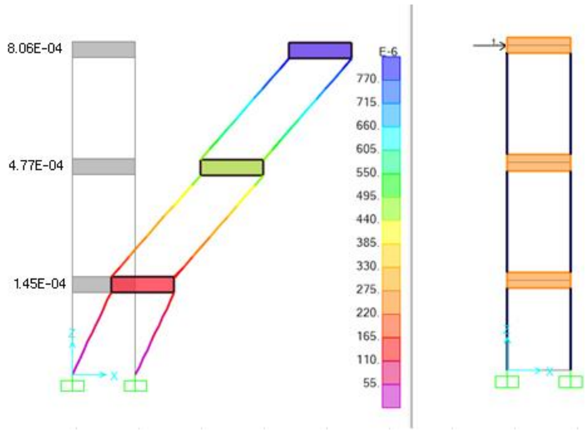


Figure 31-Application of unitary force on each story and the relative displacements (Computers and Structures Inc.)

The structure undergoes external excitation and responds with deformation in order to reach equilibrium. The relative displacement of each story was measured and arranged in the displacement matrix [U], as the following:

$$U[\mathrm{m}] = \begin{bmatrix} 0.0001448 & 0.0001449 & 0.0001451 \\ 0.0001449 & 0.0004771 & 0.0004774 \\ 0.0001451 & 00.0004774 & 0.0008057 \end{bmatrix}$$

[U] is symmetric and square, therefore can be invertible. the computation of the stiffness matrix [K] is direct, and programed in MATLAB by a simple command: `K=inv(U)`

$$K[m^{-1}] = \begin{bmatrix} 9921.38 & -3009.920 & -3.291 \\ -3009.92 & 6061.75 & -3049.70 \\ -3.29 & 6061.75 & 3048.78 \end{bmatrix}$$

4.2.2. Programing in MATLAB

Another numerical model was designed in MATLAB commercial software. This model was made in order to decrease the level of uncertainty by holding 2 numerical models that describe the same physical structure. Moreover, the programing in MATLAB require us to understand each step in the modal analysis processing, unlike the FEM designing. Therefore, in case of error we can point out the precise problem in the model program.

We would like to preform model analysis in Matlab. First part is related to the definitions of the mass matrix [M] and the stiffness matrix [K], both repeat the same representation as described in last paragraphs. The second part is the use of command "eig":

```
[F,E]=eig[K,M]
```

"`[F,E] = eig(K,M)` returns diagonal matrix E of generalized eigenvalues and full matrix F whose columns are the corresponding right eigenvectors (MathWorks)."

After we understood how MATLAB execute the modal analysis, the results will be shown graphically (by means of the modal shape plotting with the correspondence natural circular frequency [Hz]) and numerically by the plotting of F and E.

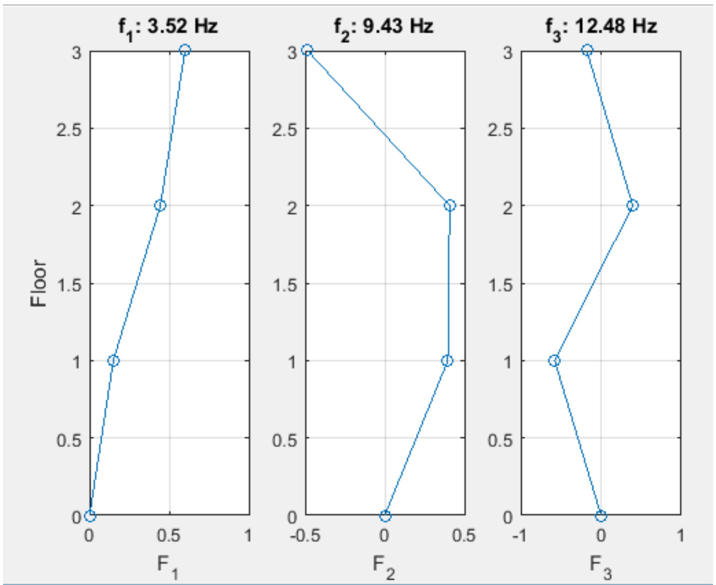


Figure 32-Plotting of the modal shapes (MathWorks)

$$F = \begin{bmatrix} 0.149 & 0.396 & -0.578 \\ 0.445 & 0.407 & 0.394 \\ 0.599 & -0.484 & -0.176 \end{bmatrix} \quad E = \begin{bmatrix} 489.6 & 0 & 0 \\ 0 & 3510.1 & 0 \\ 0 & 0 & 6151.3 \end{bmatrix}$$

F is the modal shapes matrix, where E is the natural circular frequencies squared matrix.

An interpretation of the modal properties can be done by considering the modal normalization on paragraph 1.3.4. If we are interested in applying a superposition effect in a pre-uncoupled system, the normalization property of the modal shapes must be checked:

$$[\mathrm{F}]^{\mathrm{T}}[\mathrm{M}][\,\mathrm{F}] = [\mathrm{U}c]$$

$$[\mathrm{F}]^{\mathrm{T}}[\mathrm{K}][\,\mathrm{F}] = [\mathrm{F}c]$$

$$Fc = \begin{bmatrix} 489.6 & 4.2e^{-13} & 4.6e^{-13} \\ 4.5e^{-13} & 3510.1 & -9.6e^{-13} \\ 0 & -6.8e^{-13} & 6151.3 \end{bmatrix} \quad Uc = \begin{bmatrix} 1 & 5.6e^{-17} & 1.1e^{-16} \\ 5.6e^{-17} & 1 & 8.3e^{-17} \\ 1.1e^{-16} & 8.3e^{-17} & 1 \end{bmatrix}$$

In fact the matrix Uc is representing an identity symmetric matrix with unitary value along the diagonal and quit null values outside the diagonal. The matrix Fc has squared natural frequencies terms along her diagonal and quit null values outside the diagonal. Both matrixes satisfy the requirement of normalization; therefore the modal matrix F is valid for further dynamic analysis and the natural frequencies obtained can be consider as stable .

4.3. Comparison between the physical model and the numerical models results

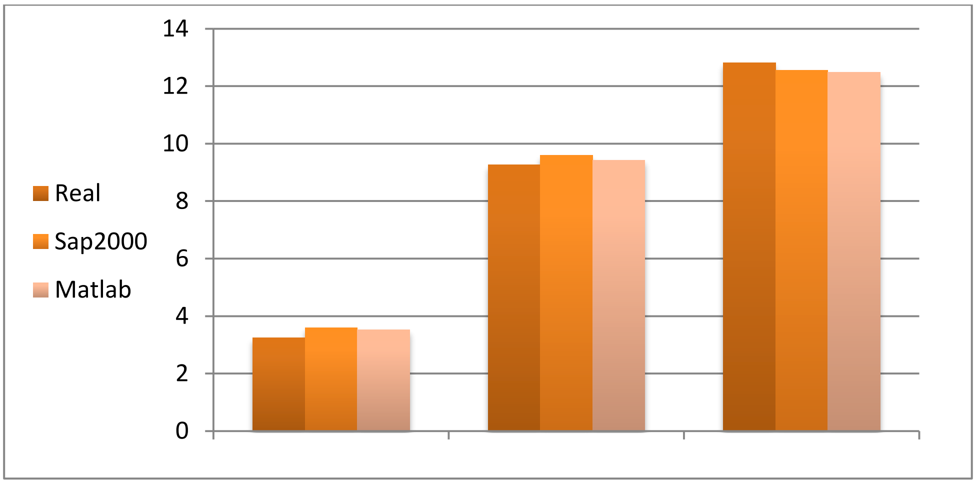
In order to understand if the numerical models are able to describe the dynamic behavior of our structure, we have to make some objective comparison between the evaluated modal characteristics - in terms of natural frequencies, motion polarity and modal shapes. The first one can reflect relevance if the error between the physical and the numerical models is less than 10 [%], while the second characteristic must be exactly the same for both 3 models. The modes shape may represent the same behavior as well.

On the one hand, the experiment results performed by the stochastic subspace identification in Macec (MACEC) are approximation of the structural characteristics, on the other hand, these results are obtained by sophisticated techniques and therefore can be considered as the real physical results.

The interpretation of the natural frequencies is shown in table 6 and graph 1:

	Experiment	Sap2000		Matlab	
mode	frequency[Hz]	frequency[Hz]	error [%]	frequency[Hz]	error [%]
1	3.25	3.6	9.7	3.52	7.7
2	9.26	9.6	3.5	9.43	1.8
3	12.81	12.55	2.0	12.48	2.6

Table 6



Graph 1 –Natural frequency [Hz] in vertical axis with respect the mode number [-] horizontal axes

As shown numerically in graphic table 6 and graph 1, both SAP2000 and Matlab models' natural frequencies have lower than 10[%] error rate.

The polarity motion is a vector property with a significant importance. It represents the direction of the modal shapes. Considering the X horizontal direction of motion, in any modal shape each floor undergoes an independent displacement that can be positive or negative. We would like to understand if the relative motion between the 3 floors for each mode is the same for all models.

The motion polarity attached in table 7:

Floor	Experimental			Sap2000			Matlab		
	mode 1	mode 2	mode 3	mode 1	mode 2	mode 3	mode 1	mode 2	mode 3
Third	+	+	+	-	-	-	+	-	-
Second	+	-	-	-	+	+	+	+	+
First	+	-	+	-	+	-	+	+	-

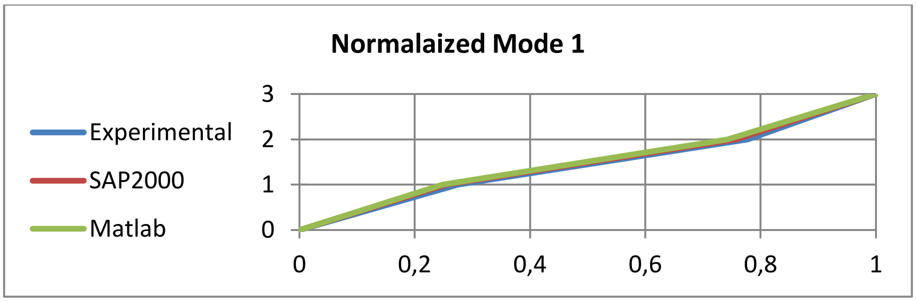
Table 7

After we considered the direction of the modal shapes, we would like to compare also between their magnitudes. In order to execute this kind of comparison, the modes magnitude must be scaling in the same range of orders. Therefore, a Normalization procedure should be done -in each mode shaped we dived each floor value with respect the floor undergo the highest displacement.

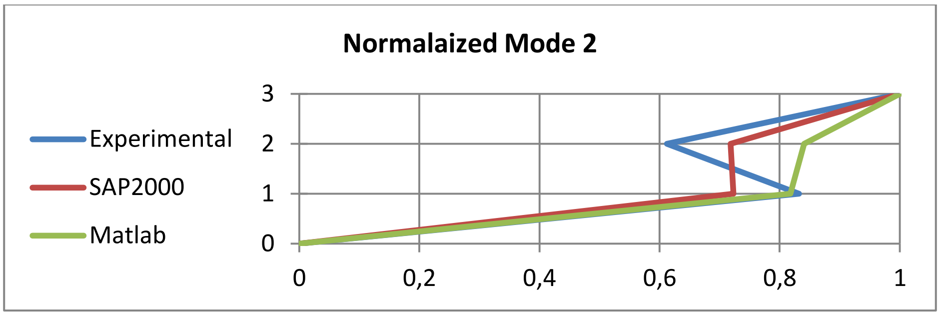
The Comparison between the magnitude of the normalized modal shapes shown in table 8 numerically and plotting in Graphs 2-4:

	Experimental					
	mode1		mode2		mode3	
Third Floor	1.000		1.000		0.500	
Second Floor	0.778		0.612		0.830	
First Floor	0.275		0.833		1.000	
	SAP 2000					
	mode1	error[%]	mode2	error[%]	mode3	error[%]
Third Floor	1.000	0	1.000	0.0	0.327	34.7
Second Floor	0.757	2.6	0.718	17.3	0.630	24.1
First Floor	0.255	7.3	0.723	13.1	1.000	0.0
	Matlab					
	mode1	error[%]	mode2	error[%]	mode3	error[%]
Third Floor	1.000	0	1.000	0.0	0.304	39.2
Second Floor	0.743	4.5	0.841	37.3	0.682	17.9
First Floor	0.249	9.6	0.818	1.7	1.000	0.0

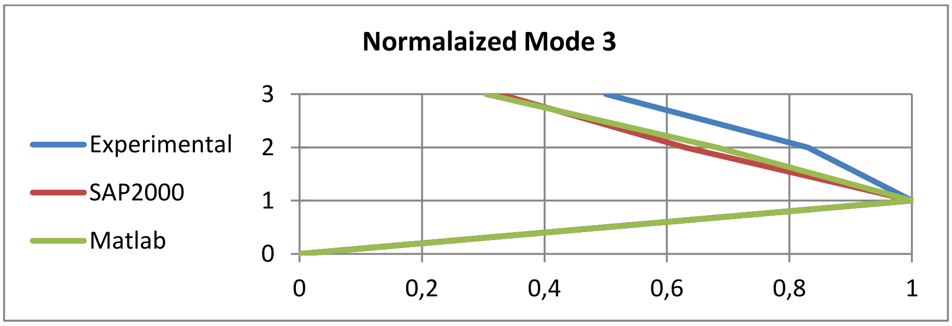
Table 8



Graph 2



Graph 3



Graph 4

The comparison between the modal shapes magnitude demonstrate the correlation in the first mode. In the second mode on the second floor the error represented by SAP2000 and Matlab is 17.3 [%] and 37.3 [%] respectively, much higher than requested. The upper floors of the third mode also shows a high value of error.

In conclusion, both SAP2000 and Matlab numerical models are representing an adequate approximation for the real dynamic behavior. The natural frequencies are quit the same with less than 10 % error, meaning that the models will respond in the same way in case of dynamic excitation. Moreover, the tendency of motion of each floor is exactly the same for both models as the real one, their rigid resistance on the horizontal direction has the same polarity.

Due to relatively high values of error shown in the second and the third modal shape, we chose to improve our numerical models by adjusting their structural properties. An optimization procedure should be done in order to fix the exact values of the hinges stiffness in the numerical models.

5. Optimization Iteration Process

5.1. Stiffness manipulation:

The particular connection be between the boxes and the lateral plates described in paragraph 1, can be represented by a hinge with upper and lower rotational springs. The implementation of this connection in SAP2000 was defining as a hinge with infinite value of stiffness -partial fixity. This assumption was done to define the numerical model. But after comparing the numerical results with those obtained by the experiment in the last paragraph, we understood that to have a more precise numerical model, the stiffness must be manipulate- The correct values of each spring must be defined.

The first step we have to follow In order to optimize the stiffness values, is to run SAP2000 from the background in a batch mode. This will allow us to interact with the general model without physically opening it.

The second step is to implement in MATLAB iteration code. The code will bring us the possibility to modifying the stiffness values in the input files and importing them into the SAP2000 model automatically. Afterwards, Running SAP2000 in batch mode will result output files that may compare with the experimental results that in case of inequality will modify the stiffness and start the loop again.

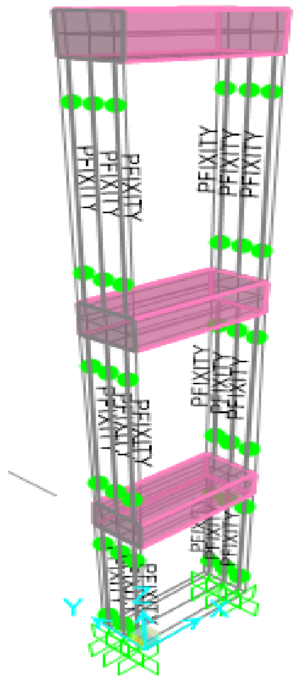


Figure 33-Partial Fixity hinges (Computers and Structures Inc.)

5.2. Batch mode

5.2.1. Introduction

Batch Processing is a non-interactive execution working method in computing systems, where offline predetermined computation is done, without direct interactive work. This kind of process is resource-intensive and takes a long time to run. Its work pattern is designed to maximize the computational output.

The whole process is operating by using of batch files that include commands in specific script languages. In Particular, Batch files are consisting of series of commands that can be executed using the command line interpreter and can be stored in a low consuming file data as a text files. They may contain any command that the interpreter accepts interactively and use constructs that enable conditional branching and iteration within the batch file.

5.2.2. Batch files in SAP2000 (Computers and Structures Inc.)

Before starting the iteration procedure, a specific batch model should be defined based on the primitive model (mymodel.sdb). When a SAP2000 batch file is ran, SAP will open the listed model files in succession, run their analyses, and manage the analysis files without requiring any action by the user.

The definition of the batch file in SAP2000 requires an access to the “batch file control” command. We specified the new batch file (batchfile.sbf), with a path that coincide with the path of the primitive model. Next step is to add the batch file to the control list so that SAP2000 will recognize its location and its path. In the “load cases to run “, we have to specify the load case of interest, that in our case is the self-weight or all cases. Further step is the specification of the correlated SAP2000 model, that the batch file is based on. Before saving the batch file we should define specific “option for analysis files” among the following:

- **Save All**. SAP2000 will save all of the analysis files generated after the analysis are ran.
- **Save Recovery Only**. SAP2000 will save the minimum number of files needed to generate results. Use this option when file storage space is limited.
- **Delete All**. SAP2000 will delete all of the analysis files generated when the analyses are ran.

We will choose the first option, and proceed by saving the batch file as a text file.

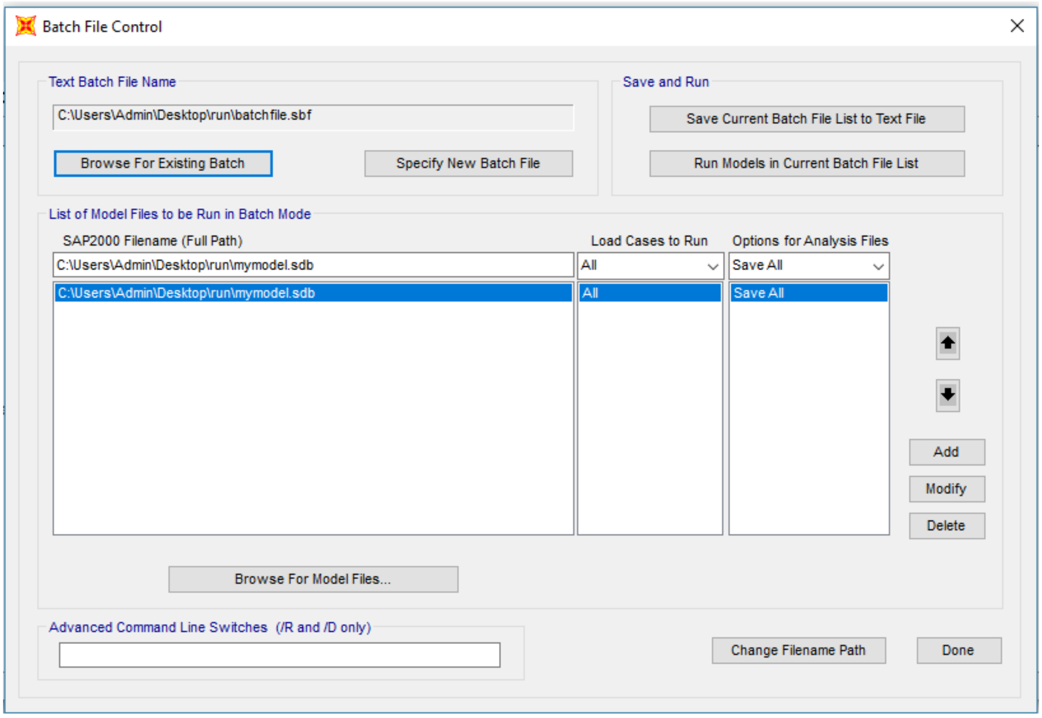
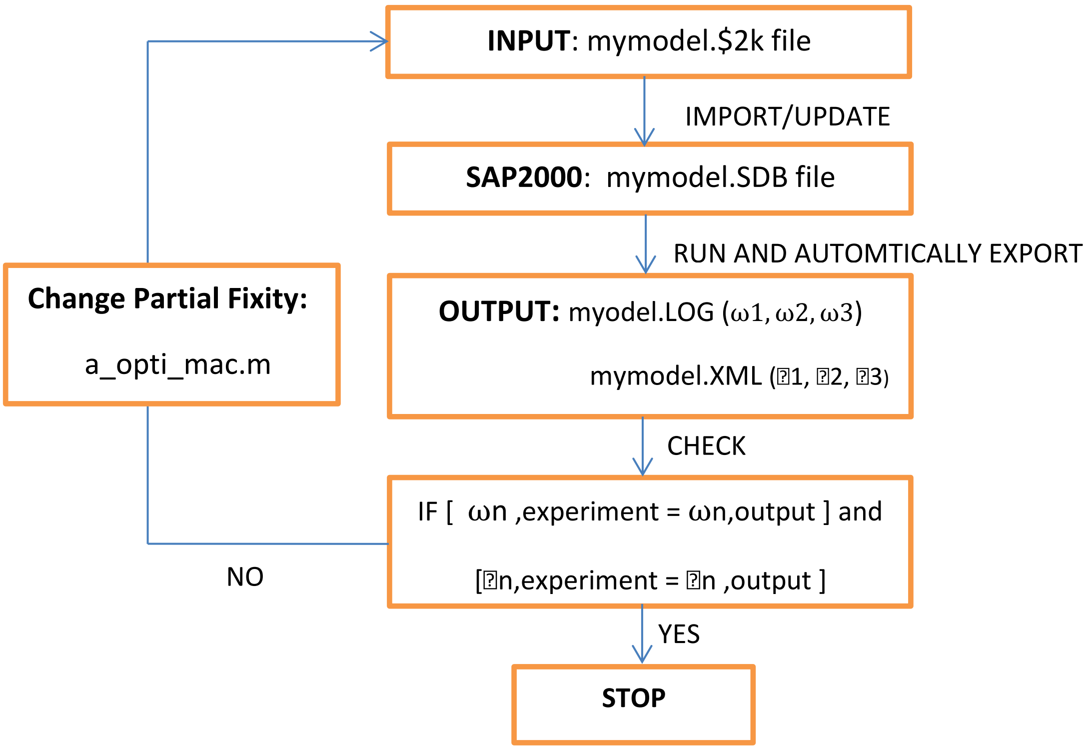


Figure 34 –Batch file control in SAP2000 (Computers and Structures Inc.)

5.3. Implementation

5.3.1. Workflow

Before implementing the iteration process in MATLAB, a flow chart that contain the main steps was made. This diagrammatic representation of the algorithm is a step by step approach that helps us simplify the task, and find a solution for the stiffness uncertainty problem.



In order to execute this kind of iteration procedure, MATLAB has to call SAP2000 automatically. We should use the command "dos" and implement the following expression from the editor:

```
dos('"C:\Program Files\Computers and Structures\SAP2000 20\SAP2000.EXE"
"C:\Users\Admin\Desktop\mymodel.$2k"  /R /C');
```

Using the path to SAP2000.exe, the path to a SAP2000 data file, and two optional switches which control software operation after the data file opens (Computers and Structures Inc.):

- C:\SAP2000\SAP2000.EXE -This is the path to SAP2000.exe
- C:\DATA\MYMODEL.$2k-this is the path to our model, where the .$2k is an extension to the .sdb and it is a type of text file which the software will attempt to import.

- /R - automatically runs analysis after the data file has been successfully opened or imported.

- /D - automatically performs all possible design after the data file has been opened or imported and analysis results are available.

5.3.2. Input files into a_ansys.m:

The input file mymodel.$2k contain the overall geometrical and structural properties of the SAP2000 model. As we are interested in the values of hinges stiffness, we should program our algorithm to use this information.

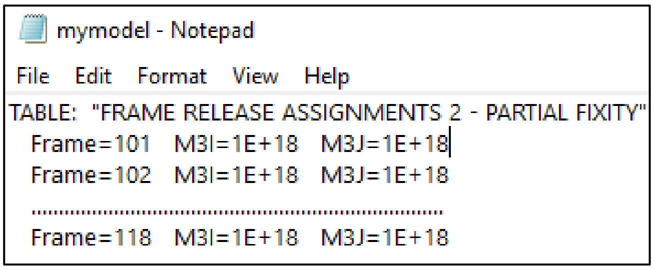

mymodel - Notepad
File Edit Format View Help
TABLE: "FRAME RELEASE ASSIGNMENTS 2 - PARTIAL FIXITY"
Frame=101 M3I=1E+18 M3J=1E+18
Frame=102 M3I=1E+18 M3J=1E+18
....
Frame=118 M3I=1E+18 M3J=1E+18

Figure 35 – mymodel.$2k partial fixity values (Computers and Structures Inc.),Microsoft Notepad

Consider 18 different beam frames, counted from 101 to 118 in the txt file and from n1 to n2 in the matlab code, with hinges in the starting (upper part) and ending(lower part) points. In all of them the same initial conditions for the stifness were imposed $= 1 * 10^8 \left[\frac{N*mm}{rad}\right]$.

Before the implementation of the import data in the file, The Parameters for the steepest descent method were defined comparing to the hinges. The 36 different hinges can be simpified into 12 paramters, because of the fact that along the Y axes, hinges in the same height and side will demonstrate the same behaviour. Morover, we may change the data type from string to number.

```
n1= ['Frame=101   M3I=' num2str(Parameterset(1,1,1)) 'M3J=' num2str(Parameterset(1,1,2))];
n2= ['Frame=102   M3I=' num2str(Parameterset(1,1,1)) 'M3J=' num2str(Parameterset(1,1,2))];
n3= ['Frame=103   M3I=' num2str(Parameterset(1,1,1)) 'M3J=' num2str(Parameterset(1,1,2))];
…….

n16= ['Frame=116   M3I=' num2str(Parameterset(1,1,11)) 'M3J=' num2str(Parameterset(1,1,12))];
n17= ['Frame=117   M3I=' num2str(Parameterset(1,1,11)) 'M3J=' num2str(Parameterset(1,1,12))];
n18= ['Frame=118   M3I=' num2str(Parameterset(1,1,11 ))'M3J=' num2str(Parameterset(1,1,12))];
```

We should define a transition array that connect the parameters with the imported data:

```
ue_ges=cell(1,18);
```

At this moment we are able to import the input file mymodel.$2k to the matlab code, taking the number of lines into account:

```
file_in='mymodel.$2k';
num_lines=683;
fileID=fopen(file_in,'r');
change1=cell(1,num_lines);
for i=1:1:num_lines
```

```
    change1{i}=fgetl(fileID);
end
fclose(fileID);
for k=1:1:18
    change1{250+k}=ue_ges{k};
end
```

5.3.3. Output files into a_ansys.m

As the input file and parameters were defined, the modal analysis of the SAP2000 model executed in batch mode automatically. The termination of the analysis lead to an outcome in terms of modal shapes and natural frequencies. SAP 2000 generate us 2 different output files:

- mymodel.LOG – text file that contain the complete dynamic results –where we are interested in the natural frequencies $\omega 1, \omega 2$ and $\omega 3$. This file is a function of the iteration number, meaning that it changes by time. Each file shows the current results and the previous ones. Therefore, the position of the data of interest is changed by each iteration.
- mymodel.XML - extensible markup language file. This kind of output file is not generated automatically every time the analysis is complete, but must be defined priory by a user. As we are interested in the modal shapes, we have to command SAP2000 to plot the joints displacements (Analysis options). The data shown in the file is always given by the actual iteration. Therefore, the position of □1, □2 and □3 maintain the same.

Because we are dealing with a rigid motion, joints that belong to the same floor will undergo the same displacement. For that reason, we choose 3 reference joints, that all are part of the same vertical beam: joint 1 for the upper floor, joint 9 for the middle floor and joint 29 to the lower one.

Firstly we would like to deal with the modal shapes detailed in the xml file. This file contains a mix of numbers and letters –string data. The first step is to read each row of the file and insert it into a vector [1x2402] where 2402 is the total number of rows. Each row will be inserted into a cell element.

```
%read xml
file_xml='mymodel.xml';
num_lines_xml=2402;
fileID_xml=fopen(file_xml,'r');
change2=cell(1,num_lines_xml);
for i=1:1:num_lines_xml
    change2{i}=fgetl(fileID_xml);
end
```

As mentioned above, the position of the joints displacement will remain constant during the evolution of the iteration procedure. Thus, we can physically point out the position of each joint displacement comparing to the 3 modes:

```
joint1mode1=change2{68};
joint1mode2=change2{81};
...
joint29mode3=change2{991};
```

Each parameter jointNmodeN still contains letters and number. We will isolate the numbers only by using the command regexp. Then each expression will be divided into cells that contain the whole numerical expressions. Following that, the expression in each cell may be transferred to numbers. Simple algebraic steps can help obtain each mode shape. For example for the first joint with the first mode:

```
%JOINT 1
A11 = regexp(joint1mode1,'\d*','Match');
B11 = cellfun(@str2num,A11);
M11=B11(2)+(B11(3)*10^(-6));
```

The mode shapes can be defined and the matrix F [3X3], after being normalized compared to the absolute value of the biggest value of each mode:

```
mode1=[M11;M91;M291];
mode2=[M12;-M92;-M292];
mode3=[M13;-M93;M293];
F=[mode1/max(abs(mode1)),mode2/max(abs(mode2)),mode3/max(abs(mode3))];
fclose(fileID_xml);
```

Secondly we can treat the natural frequencies in the LOG file. As the XML, also this file contains a mix of numbers and letters. But, Because of the fact that the position of the variables of interest is changing with each iteration, we have to assume different technique in order to be able to select the correct frequencies among the all data.

In order to import the file, we used the command textread with some specifications (MathWorks): %s – give a cell array of character vectors as an output , delimiter - act as delimiters between elements.

```
fileID_txt=textread('mymodel.LOG','%s','delimiter');
```

For each natural frequency we will follow the same technique in order to obtain the one relevant to the actual iteration step. For example for the first natural frequency: The command find and strfind were used in order to obtain the all possibilities of rows that contain the first natural frequency. All possibilities were arranged in c array with size [m,n]. Knowing that the actual data will be represent in the last cell, we can call c in his maximum position m.

```
c1=find(~cellfun(@isempty,strfind(fileID_txt,'Found mode      1')));
[m1,n1]=size(c1);
omega1=c1(m1);
```

The information inside each cell should be manipulated. Once again, regexp divides the expression into different cells of string data. This cells were transformed into array and the string data type into double. By understanding the size of each transformed cell, we can understand if the natural frequency is smaller or bigger than 0 and manipulate it accordingly:

```
find1=fileID_txt(omega1);
b1=regexp(find1,'\d+(\.)?(\d+)?','match');
out1=str2double([b1{:}]);
```

```
a1=size(out1);
if a1==[1,7]
    MW1=out1(5)*10^(-out1(6));
else
    MW1=out1(5)  ;
end
```

The 3 natural frequencies were arranged on the diagonal of [3X3] matrix W.

```
W=[MW1,0,0;0,MW2,0;0,0,MW3];
```

5.3.4. Minimization function F

The file a_func.m is an algorithm that approximates F the minimization function. This function will be used in the further steps for the steepest descent method. Using 2 parameters as an input:

- Parameterset- is the variable defined in the a_ansys.m algorithem that contain the 12 parameters of interest .
- wicht_fakt_in -is the weight of each mode shape, represents its influence on the results.

```
function [ funk_out ] = a_funk( Parameterset ,wicht_fakt_in)
```

Every time an iteration procedure is executed, a_optic_mac.m file calls a_func.m in order to calculate its minimization threshold function. This will be done by linear regression analysis technique based least squares method. In the space of $\mathbb{R}^{12x12}$ parameters, we can define a complex surface that its distance from the data points is the least square sensitivities.

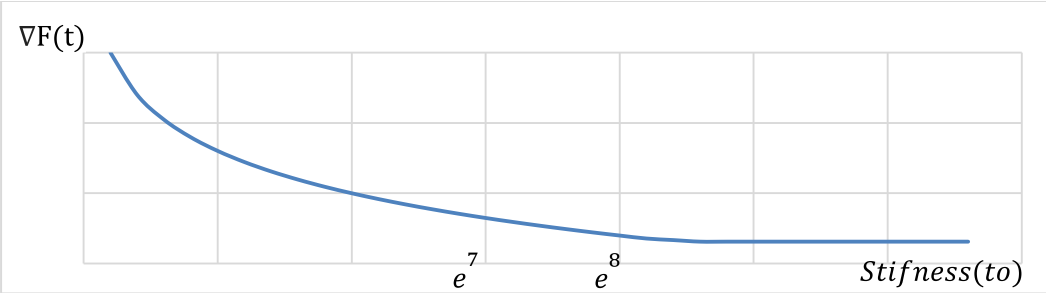
Due to the complexity of the space orientation in a [12D] space, the visual perception of the shape of F is impossible. Therefore; we would like the represent F in [1D], considering a single parameter. Because of the fact that this function has an important influence on the optimization process, some particular properties where obtained by observation and may be explained.

<u>Observations:</u>

The first observation is related to the dependency of the gradient of F on the initial conditions of the global stiffness matrix K:

$$\frac{\mathrm{dF}}{\mathrm{dx}}\mathrm{dx} = \nabla \mathrm{F} = \mathrm{f}(\mathrm{K}(\mathrm{t}=0)) \text{ Where } x \in \mathbb{R}^{1x12}$$

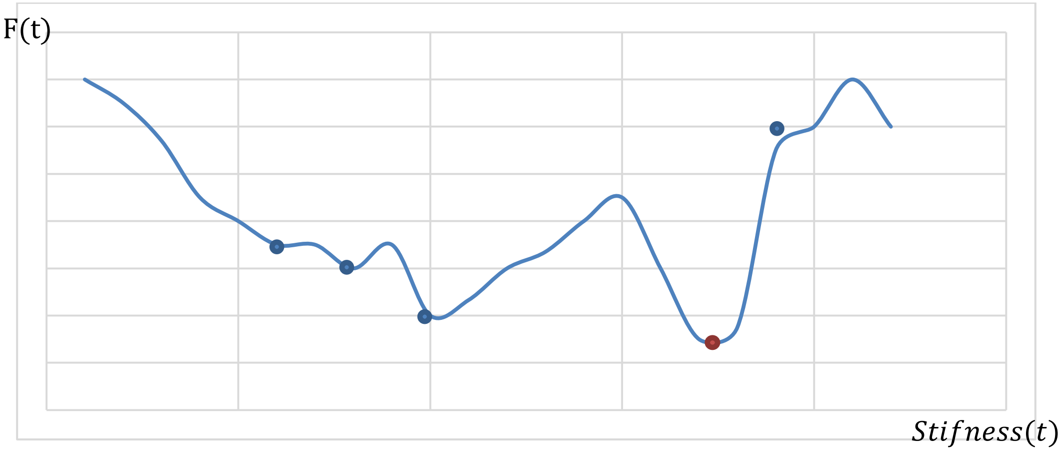
The variation of the gradient of the function by the time depends on the pre-defined values of the partial fixity in time t_0=0[sec]. In particular for higher values, the function does not change from iteration to iteration, while for moderate values the variation is noticeable.



Graph 2-Changing of the gradient of F with the stiffness initial conditions

By observing our model case, we deduce that for a value of stiffness higher than e^8, there is no significant changing in the value of the minimization function.

The second observation relates to the function extreme points. When the algorithm is running, it acts in order to find the minimum point of the minimization function, by increasing or decreasing the parameters (stiffness). As the shape of the function is unknown, it is possible that the algorithm will find local minimums that are not the absolute one. An arbitrary example shown in graph 3:



Graph 3-Local (blue) and Global (red) minimum points of minimization arbitrary function F

Therefore, the output of this function is a number that represents the first minimum point that the algorithm encounters. This number is kind of threshold value, in which the optimization algorithm is tending. The smaller this number, the closer is the model to a more desirable result. We are expecting that from iteration to iteration this number will decrease until optimal value reaches close to null.

5.3.5. Method of steepest descent (Weber e Arfken 2004)

The optimization algorithm file a_optic_mac.m is based on the method of steepest descent. This kind of method tries to approximate the behavior of a function for large values of a variable or some parameters. We can consider our parameter as 's', that represent the asymptotic behavior of the function herself.

The analytic function I(s) can be defined as the definite integral of F:

$$I(s) = \int_C F(z, s)dz$$

F is the analytic function in z that depends on the parameter s. The path C is considered as the real axes of the function that in case of complex plane can be deformed from C to C´. The absolute value |F| is the one governing the behavior of the integral mentioned before. For this reason, the contours of the absolute value |F| at constant intervals |F| can be assumed more closely spaced as s becomes larger.

Two mathematical principals should be explained before going on:

- Saddle point- can be defined as a critical point. A point on the surface of such a function where its derivatives (slopes) in orthogonal direction are all null. The point particularity is not being a local or global extreme of the function –not a peak.
- Singularity-a general un defined point of such a function.

Jensen's theorem explains why only saddle points and singularities are so important to the function being integrated. Jensen's took a complex function F, with a real part F=U and imaginary part F=V such as:

$$|F(x+iy)^2| = U^2(x,y) + V^2(x,y)$$

The plot of the absolute value squared of F over the complex plane is called the analytic landscape of Jensen.

Jensen demonstrates that – "The analytic landscape has only saddle points and troughs, but no peaks. Moreover, the troughs reach down all the way to the complex plane, that is, go to |F| = 0. In the absence of singularities, saddle points are next in line to dominate the integral (Weber e Arfken 2004)". We will use the saddle point method of Jensen, in order to understand the asymptotic behavior of:

$$I(s) \; for \; s \longrightarrow \infty$$

The real part U of F has a local maximum, what means that:

$\frac{\mathrm{dU}}{\mathrm{dx}} = \frac{\mathrm{dU}}{\mathrm{dY}} = 0$, by the use of the Cauchy–Riemann also $\frac{\mathrm{dV}}{\mathrm{dx}} = \frac{\mathrm{dV}}{\mathrm{dY}} = 0$

So also the imaginary part V of F has a minimum. The theorem of Jensen prevents the possibility of both U and V having a maximum or minimum as represented in figure 36:

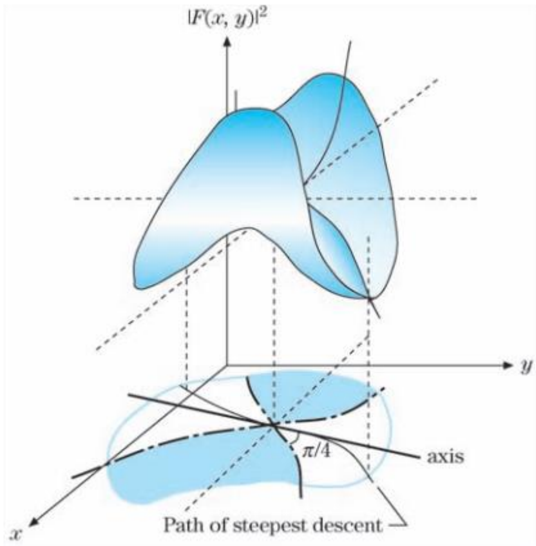


Figure 36-Saddle point of a complex function (Weber e Arfken 2004)

From now on we will choose the path C along the valley of the saddle points, so basically the saddle point dominates the value of I(s).

We would like to prove that there are no peaks along this path. To do so, we can assume one peak at point Z_0, such as:

$$|F(Z_0)^2| > |F(Z)^2|$$

For all Z in neighborhood $|Z - Z_0| < r.if$:

$$F(Z) = \sum_{n=0}^{\infty} an\,(Z - Z_0)^2$$

Is the Talyor expansion at Z_0, meaning value m(F) on the circle $Z = Z_0 + re^{i\varphi}$ can be seen as :

$$m(F) = \sum_{n=0}^{\infty} |F(Z_0)|^2$$

m(F) is the averge value of $|F|^2$, laying on a radius r of a circle. Therfore there must be a point Z_1 on the circumference such as $|F(Z_1)|^2 \geq m(F) \geq |F(Z_0)|^2$,that infact contract out assumtption – that lead in to the conclusion that there is no peak point.

We can assume that there is no minimum at Z_0,such as:

$$0 < |F(Z_0)|^2 < |F(Z)|^2 \text{ for all Z in neighborhood } Z_0.$$

Baisically the tendency of the dip in the valley is never going down to the complex plane. Becuase of that : $|F(\mathrm{Z})|^2 > 0$ and since $\frac{1}{F(\mathrm{Z})}$ is analytic in this neigborhood -that mean that it has a taylor expansion where the point Z_0 will be the extremum of $\frac{1}{|F(\mathrm{Z})|^2}$ which can not happen .

The Jensen therorem was demonstrated and we can use the saddle point method for our case of study. To do so , we would like to transform F to exponential form $F(Z) = e^{f(z,s)}$. This kind of transformation will bring us two properties :The exponential function has no zero in the complex plane, and any saddle point with $F(\mathrm{Z}) = 0$ becomes a trough of $|F(\mathrm{Z})|^2$ because of$|F(\mathrm{Z})|^2 > 0$.

Because $F(\mathrm{Z})$ is a continouis periodic function, we can define its phase at $F(\mathrm{Z}_0)$ such as $\Im f(\mathrm{Z}_0)$. At Z_0 the conditions that locates the saddle point can be defined –where the tangential plane is horizontal :

$$\frac{df}{dz} = 0 \quad or \quad \frac{dF}{dz} = 0 \quad , \; where\, \mathrm{Z} = \mathrm{Z}_0$$

The main goal of the saddle point method is to <u>define the direction of the steepest desent</u>. We can use the power expansion of f at Z_0 :

$$f(z) = f(\mathrm{Z}_0) + \frac{1}{2} f''(\mathrm{Z}_0)(Z - \mathrm{Z}_0)^2 + \cdots,$$

Using elementary mathematical steps we can demonstarte that F has costante phase along the saddle point axes :

$$arg(Z - \mathrm{Z}_0) = \frac{\pi}{2} - \frac{1}{2} arg f''(\mathrm{Z}_0) = costant$$

The line through Z_0 defined by $f''(\mathrm{Z}_0)(Z - \mathrm{Z}_0)^2 = t^2$, with t real number.This line is orthogonal to the axes themselves (dashed line in figure 28) .If we consider the curves $\Re f(Z) = \Re f(Z_0)$,we can express their angles as :

$$arg(Z - \mathrm{Z}_0) = \begin{cases} \frac{\pi}{4} - \frac{1}{2} \arg(f''(\mathrm{Z}_0) + \mathcal{E}), & t > 0 \\ -\frac{\pi}{4} - \frac{1}{2} \arg(f''(\mathrm{Z}_0) + \mathcal{E}), & t < 0 \end{cases}$$

Where ε is term that collect orders of magnitude with less influence. This curves (dot dashed in figure 28) divide the zone of the saddle point into 4 principal sections: two with $\Re f(Z) > \Re f(Z0)$ (shaded part in figure 28) and other two with $\Re f(Z) < \Re f(Z0)$. This curves district with $\pm\frac{\pi}{4}$ angels from the axes. The integration path tries to slip away from the shaded areas where the Absolut value of F rises. We may consider the fact that rapid oscillations of the function in exponential form and cancelling contributions to the integral, can occur if we choose a path that run the slopes above the saddle point.

Next step is to specialize the integrand F further in order to bring up the specific path with asymptotic behavior as $s \to \infty$. The parameter s is considered as linearly variable in the exponential. The zone far away from the saddle point is not influenced directly by the parameter s, but may be considered by the function $g(z)$. The integral in new form can be represented as:

$$I(s) = \int_C F(z,s)dz = \int_C g(z)e^{sf(z)}dz$$

We would like to estimat the integral in the neigbrhood of the saddle point –this will give us the path of the steepest descent, where ε is to be neglected. We define:

$$Z = \mathrm{Z}_0 + Xe^{i\alpha} \text{ ,where } \alpha = \frac{\pi}{2} - \frac{1}{2}\arg f''(\mathrm{Z}_0) \text{ , } a \le x \le b$$

We define our intergal in the path from a $\to b$:

$$I(s) \approx e^{i\alpha}\int_a^b g\left(\mathrm{Z}_0 + Xe^{i\alpha}\right)e^{\left[sf\left(\mathrm{Z}_0+Xe^{i\alpha}\right)\right]}dx$$

We can develop the f and g functions using taylor expansion around the point $Z = \mathrm{Z}_0 + Xe^{i\alpha}$ .By using elementary mathematical steps we can find the new leading term of the integral :

$$I(s) = g(\mathrm{Z}_0)e^{sf(Zo)ti\alpha}\int_a^b e^{-\frac{1}{2}s|f''(Zo)|x^2}\,dx$$

We can let $b \to \infty$ and $\to -\infty$, since the last integral is null when x departs appreciably from the origin. We can consider the remaining integral as Gauss error integral:

$$\int_{-\infty}^{\infty} e^{-\frac{1}{2}c^2x^2}\,dx = \frac{1}{c}\int_{-\infty}^{\infty} e^{-\frac{1}{2}x^2}\,dx = \frac{\sqrt{2\pi}}{c}$$

We obatin the <u>governic asymphtotic formula for the steepest descent method</u> :

$$I(s) = \frac{\sqrt{2\pi}g(\mathrm{Z}_0)e^{sf(Zo)}e^{i\alpha}}{|sf''(\mathrm{Z}_0)|^{1/2}}$$

We can conclude by saying that this integral is calculating in the contour of a complex plane path, passing through the neighborhood of a saddle point. It brings us the <u>possibility to understand in which direction we have to go from our parameters state in the [12D] space to the minimization function.</u>

5.3.6. The Optimization algorithm into a_optic_mac.m

So far in this paragraph, we understood 3 fundamental components for the optimization of our stiffness parameters:

1. Their Actual state in the [12D] space-determined by the FEM analysis.
2. Minimization function F –is the optimal state we want to bring our parameter through every iteration.
3. Asymptotic integral $I(s)$ –tells us in which direction in the space we should go in order to reach F.

By integrate these ingredients with the experimental results, we have the all information necessary in order to implement the last piece for the optimization process.

This kind of implementation requires high programming capabilities with sufficient time. Therefore, a readymade MATLAB algorithm (a_optic_mac.m) was adopted for our case of study, thanks to the work of Hagen Berthold(Institute of Structural Mechanics and Design, TU Darmstadt).

Based on the method of the steepest descent, the algorithm contemporary call the a_ansys.m (running SAP2000 automatically and preform analysis) and a_func.m files, in order to have information about the parameters in exam and the minimization function. By combining these data, it approximates the asymptotic integral for each iteration or the direction in the space each parameter should follow in order to arrive to the minimization function. Following this path and by knowing the desired results, the algorithm calculates how far we are from the optimal results. In case they are not being reached, it increase or decrease (depending on the prediction of the saddle point of the minimization function) the parameters and prepares for another loop.

As already discussed, the stiffness initial conditions have a great influence on the optimization process. Because the shape of the minimization function is an unknown for us, we had to try different values of stiffness in order to understand with which of them the algorithm could process better. For each iteration we checked if the difference between the actual and the previous values of the minimization function is reasonable. After some trails we reached the conclusion that it is better to process the algorithm by assigning stiffness values in order of magnitude of10^7. The second property that influences the procedure is the weight of the natural frequencies and the modal shapes.

The total number of iterations was 160 and it lasted 18.5 hours. It was divided into 2 different cycles.

In the first cycle 100 iterations were performed, for initial conditions of stiffness with order of magnitude of10^7. The weight factors chosen for modes 1-3 respectively in this cycle were:

- Eigen frequencies : 1 , 2, 4.
- Modal shapes: 1, 2, 4.

In the second cycle, another 60 iterations were performed. We assigned as initial conditions of stiffness –the values we obtained after the first 100 iterations (mymodel.$2k file). The weight factors chosen for modes 1-3 respectively in this cycle were:

- Eigen frequencies : 10 , 1, 1.
- Modal shapes: 100, 1, 1.

5.4. Results

The main goal of the optimization process was to find the exact value of stiffness for each of the 36 hinges necessary to achieve the modal quantities we obtained in the experiment. These values are represented in SAP2000 the bending moment around the 3rd axes we should apply in order to assign some partial fixity, as shown in table number 9:

Frame	M3I $[\frac{N*mm}{rad}]$	M3J$[\frac{N*mm}{rad}]$
101	196209.64	372274.05
102	196209.64	372274.05
103	196209.64	372274.05
104	114779.84	309877.04
105	114779.84	309877.04
106	114779.84	309877.04
107	176259.87	177619.88
108	176259.87	177619.88
109	176259.87	177619.88
110	147848.28	119132.3
111	147848.28	119132.3
112	147848.28	119132.3
113	149833.56	122083.7
114	149833.56	122083.7
115	149833.56	122083.7
116	130143.29	137094.94
117	130143.29	137094.94
118	130143.29	137094.94

Table 9: hinges partial fixity values after optimization (Computers and Structures Inc.)

The first modal quantity we would like to represent is the natural circular frequencies. Comparing between the experimental results with those obtained before and after the optimization on SAP2000 are showing on table 10:

	Experiment	Before Optimization		After Optimization	
mode	frequency[Hz]	frequency[Hz]	error[%]	frequency[Hz]	error [%]
1	3.25	3.6	9.7	3.2202	0.9
2	9.26	9.6	3.5	8.5216	8.7
3	12.81	12.55	2.0	11.2127	12.5

Table 10: Natural frequencies result after optimization (Computers and Structures Inc.)

The interpretation of this kind of results is ambiguous. If we consider the first mode, we obtained results well with an error of less than 1 %. The first mode is the most relevant one, it defines how the structure will interact with the rest of the system around it when vibrating, and hence we can be satisfied for the improvement of the 1st natural frequency. On the other hand, the other 2 frequencies are quit far from the experimental results. We can explain it by considering the sensitives of our scaled model, that are far smaller with comparing to that of a real civil structure (Bridge an example). The optimization algorithm required much more information and time in order to find the exact parameters, also after 160 iterations.

The second modal quantity we would like to represent is the modal shapes. Comparing between the experimental results with those obtained before and after the optimization on SAP2000 are showing on table 11:

Experimental Results						
	mode1		mode2		mode3	
Third Floor	0.771		0.695		0.359	
Second Floor	0.600		-0.426		-0.596	
First Floor	0.212		-0.579		0.718	
SAP 2000 After optimization						
	mode1	error[%]	mode2	error[%]	mode3	error[%]
Third Floor	0.783	1.5	0.701	0.9	0.223	37.9
Second Floor	0.589	1.8	-0.51	20.3	-0.511	14.2
First Floor	0.198	6.5	-0.49	14.4	0.830	15.6

Table 11: Modal shapes result after optimization (Computers and Structures Inc.)

The same assumption can be done for the modes shapes. Considering the 1st mode shape, it well represents the correlation with the experimental result with a negligible error of less than 10 [%]. The other 2 modes are bit far from the requested results because of the sensitivity issue. But as the 1st mode is the most relevant one; we arrived into conclusion that these results are acceptable.

6. Validation

6.1. Time history analysis for a given signal

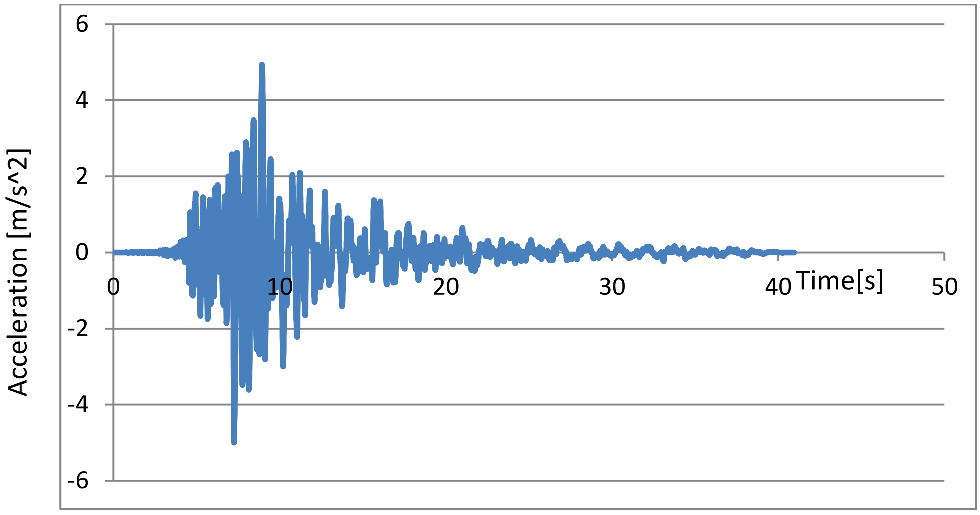
From a structural point of view, after we optimized the stiffness parameters of our numerical model as mentioned in the last paragraph, we can deduce a statically real behavior. Nevertheless, the dynamic properties at issue (natural frequencies and modal shapes), do not imply directly that our model fits the reality from a dynamic point of view. Because, In addition to the stiffness and the mass, we have to consider the damping for the global motion problem as well.

In this paragraph, we would like to control the behavior under a real base excitation application. First, we will check the response of the scaled Model, by simulating ground motion using a shaking table. Then we will apply a time history analysis, both in the SAP2000 and MATLAB models, in order to check their structural validation to an earthquake.

For the time history, we may define a given ground motion $\ddot{u}g(t)$, knowing that the deformation response $u(t)$ of an SDF system is exclusively a function of only the natural vibration period of the system and its damping ratio (Chopra 2012). We will base both the analysis in SAP2000 and MATLAB on this assumption, but preformed in different techniques.

As reference ground motion acceleration signal, we chose the Great Hanshin earthquake –of the city of Kobe, Japan on 17 of January 1995. It lasted around 40 seconds and registered as a magnitude 7.3 on the Richter scale. The quake's focus was about 16 km under the ground surface. Its estimated death toll of 6,400 made it the worst earthquake to hit Japan since the 1923 Tokyo earthquake, which had killed more than 140,000 (©2019 Encyclopædia Britannica). A record file (kobe.txt) of acceleration [$\frac{m}{s^2}$] versus time[s] will be use with a 0.01 [s] time interval.

Figure 37-Failure of building foundation by Kobe earthquake 1995 (©2019 Encyclopædia Britannica)



Graph 4- Kobe Japan time history of displacement versus time (Excel)

6.2. Optimized stiffness matrix

As we know the exact values of the stiffness of the hinges, we can once again apply the theory of elasticity in SAP 2000 - in order to evaluate the displacement of each story. Knowing that when unitary forces are applied to normal structure, their relative displacements are equal to the inverse of the stiffness's relate to each node of DOF. Therefore, the optimized stiffness matrix $[\mathrm{K}]\mathrm{optimized}$ can be proceeding where the displacement matrix $[\mathrm{U}]\mathrm{optimized}$ is known.

$$[K]optimized = [U]optimized^{-1}$$

The optimized displacement matrix was obtained because of an application of 1[N] concentrated horizontal force, on the center of mass of each floor.

$$[U]optimized\ [\mathrm{m}] = \begin{bmatrix} 0.0001806 & 0.0001807 & 0.0001809 \\ 0.0001807 & 0.0005943 & 0.0005948 \\ 0.0001809 & 0.0005948 & 0.001015 \end{bmatrix}$$

The optimized stiffness matrix derive by the mentioned matrix:

$$[K]optimized\ [m^{-1}] = \begin{bmatrix} 7958.15 & -2418.80 & -0.909631 \\ -2418.80 & 4804.49 & -2384.38 \\ -0.909631 & -2384.38 & 2382.65 \end{bmatrix}$$

6.3. Damping matrix (Chopra 2012)

Based on the equation of motion, In order to understand the structure response (in terms of u and its derivatives) to a base excitation, we have to evaluate the damping matrix [C] of our structure. The global equation will assume the following form:

$$[\mathrm{M}]\{\ddot{u}\ (\mathrm{t})\} + [\mathrm{C}]\dot{u}(\mathrm{t})\} + [\mathrm{K}]\{\mathrm{u}(\mathrm{t})\} = \{\ddot{u}g(t)\}$$

The damping ratio ξ is a dimensionless property of the system that determines the amount of overshoot and the rate at which the oscillations decay. It is a function of the mass and stiffness. From the fact that the introduction of damping makes the response much less sensitive to the Period, the choosing of ξ has high significance.

The direct estimation of the damping ratios was done in the modal analysis by MACEC (paragraph 3.3.1). We would like to consider the following damping ratios with comparing to each mode:

	Mode 1	**Mode 2**	**Mode 3**
Damping ratios [%]	0.17913	0.15259	0.1597

Table 12 -Damping ratios for each mode

Particularly, it is not possible to determine the damping matrix [C] of our model from the damping properties of individual structural elements, just as we have done before with the mass and stiffness matrixes. It is impractical to determine the damping matrix in such a way, from the fact that unlike the young modulus, which enters into the computation of stiffness, the damping properties of construction materials are much more complex to analyze. The energy dissipation due to the friction between the connections of the elements is one example. [C] should be determined from the damping ratios which account for all energy dissipating mechanisms(Chopra 2012).

In order to evaluate the matrix related to our structure, we can use an analytical relationship between the damping ratio and the natural circular frequencies in case of viscous damping in free vibration motion (Chopra 2012):

$$\xi = \frac{c}{2m\omega k}[\%]$$

The mass term can be neglected for identity; therefore, the damping matrix [C] will assume the following form:

$$[C] = \begin{bmatrix} 2\xi 1\omega 1 & 0 & 0 \\ 0 & 2\xi 2\omega 2 & 0 \\ 0 & 0 & 2\xi 3\omega 3 \end{bmatrix} = \begin{bmatrix} 7.08 & 0 & 0 \\ 0 & 16.04 & 0 \\ 0 & 0 & 22.39 \end{bmatrix}$$

6.4. Application of a base excitation

6.4.1. Laboratory experiment (Wölfel)

The main part of the validation process is based on the performance of an earthquake application in environmental conditions. We would like to evaluate the response of the scaled model to the Kobe base excitation signal. Besides the table shaker system, most of the other measurement devices and the scaled model are the same as described in paragraph 2. Moreover, the design and the arrangement of this experiment have an analogy to the previous one.

The table shaker system chosen for our investigation is Wöfel BD.5 (Wölfel). Usually, shaker systems are used for the analysis of the dynamic properties of large mechanical structures, with the possibility to allow a precise and controlled vibration excitation. The shaker systems BD.5 (figure 38) ideally suited for a wide range of applications, but in our study case, we will use its functionality of subjecting our test object to define vibration loads.

Figure 38- BD.5 shaker in horizontal operation with floor brackets (Wölfel)

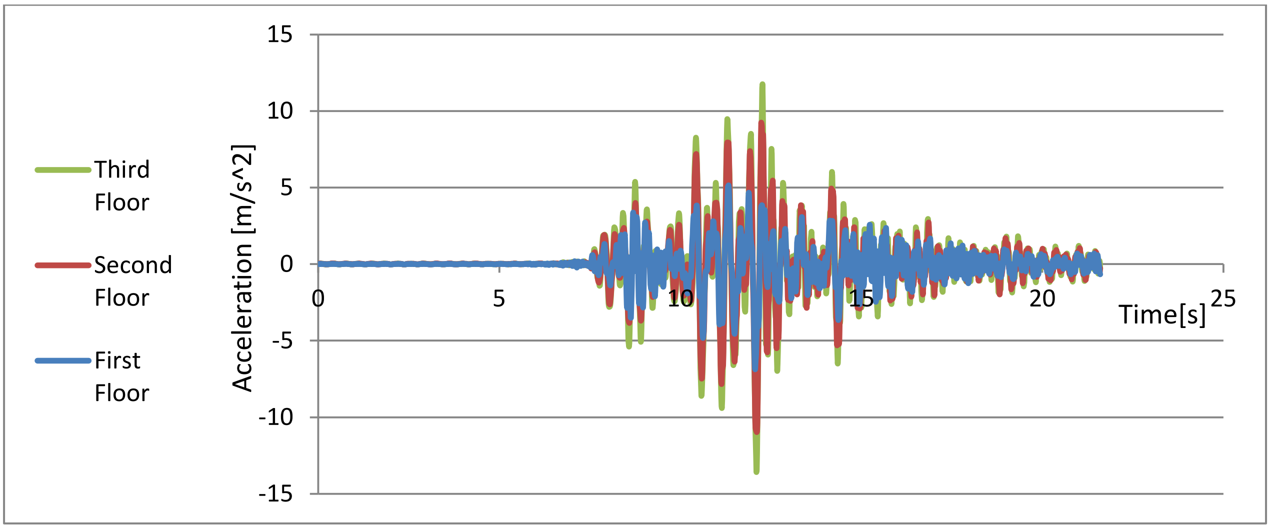
Operating by an electronic control unit, The BD.5 linear motor moves some reaction mass in a horizontal or vertical direction, that leads to a creation of inertia forces. Those transferred through the frame to our scaled model in exam. High power (maximum of 500[N]) can be produced in a range of frequencies between 0.5 - 200 [Hz], where the use of large strokes allows us the examination at low frequencies.

Considering its mechanical properties (Wölfel), BD.5 main element weight 35 [Kg], each one of the 4 slabs weight 4 [Kg] and the 2 angles for horizontal operation weight each one 4.5 [Kg]. The dimensions of the base unit are unit 185 x 160 x 840 [mm], where the dimensions of the electronic unit are 450 x 335 x 175 [mm].

Figure 39- Scaled model connected to the BD.5 shaker(Wölfel)

The txt signal file must be transformed into ASCII type, that allows the CATMAN software to use it correctly. After this procedure, the signal was applied by the shaker to the base of the 3 floors scaled model. During the experiment physically noticed peak and modest vibration intervals, caused by a translation movement of the floors along the X direction. The experiment was repeated 3 times for stable results, taking the most severe time interval of the first 21[s] into account.

The structural response was expressed in terms of the acceleration of the floors. The measurement was done by the accelerometers that were mounted in the center of each mass. The data that was manipulated by the convertor was afterwards collected by the CATMAN and interpreted by EXCEL. For each floor, we are able to plot the acceleration response spectrum, where we are interested in the maximum value of them, for the future comparison to the numerical models.



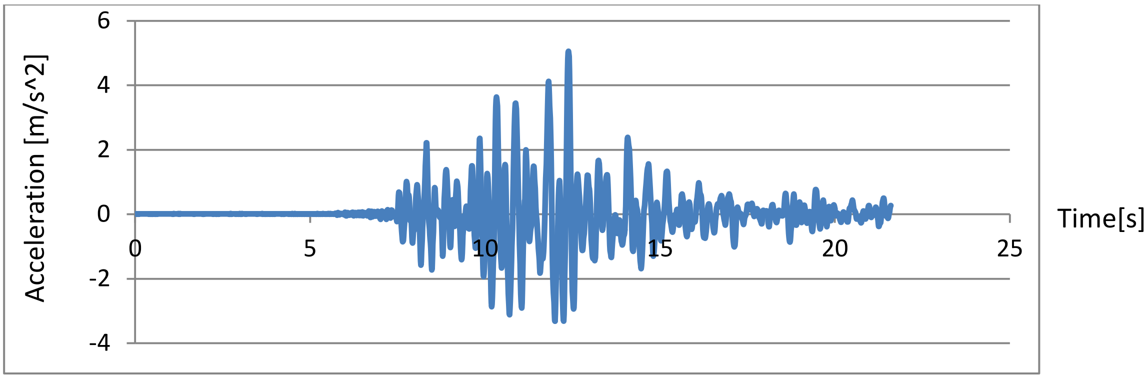
Graph 5-Time history of the floors, acceleration $[\frac{m}{s^2}]$versus time [s] EXCEL)

Floor	Max Acceleration $[\frac{m}{s^2}]$
1	5.15
2	9.24
3	11.75

Table 13-Maximum acceleration of the floors

As reliable instruments carried out in this experiment, the results are considered as those that approximate the real behavior of our model. Therefore, we would like to verify that the other numerical models may have identical dynamic behavior.

The same conditions must be assigned to the SAP2000 and Matlab numerical models. As the original signal of “Kobe” was transformed to ASCII and then was influenced by the laboratory ambient conditions, we cannot consider it. However, because the modified signal must coincide with the acceleration measured in the base floor, we will use that record as a reference for the numerical models.



Graph 6-Time history of the base floor, acceleration $[\frac{m}{s^2}]$versus time [s] (EXCEL)

6.4.2 Modal time history analysis in SAP2000 (Computers and Structures Inc.)

Validation of the FEM model should be done using Time history linear analysis. According to some specification time function, the structural response will be evaluated by reaching dynamical equilibrium. Initial conditions may be set by recalling the structural state from the end of the previous analysis.

The first passage is to define a time history function. Based on record file of channel 4 (the base floor) of acceleration $[\frac{m}{s^2}]$ versus time[s], we will define “kobe function” in SAP2000.

The second step is to add a new load case to our model. We will define “Base Excitation” as new load case of time history type and we will choose modal solution type, based on Linear analysis. The initial conditions imposed from time zero as an unstressed state, but from a dynamical point of view, it will be correlated to the previous modal load case (“Modal”). The Modal damping will be defined as a constant value of 0.015[-], with override factors that influence the other modes as following: 0.1791, 0.1526 and 0.1597. The time step data was defined with 4000[-] number of output time steps, and with a size of 0.01[-].

The last configuration is the applied load, Imposed as the “kobe” function applied as an acceleration load type In the U1 direction. For comfort, we will impose SAP2000 working with [m], such that we can choose the scale factor as 1.

We will run contemporary “Modal” and “Base Excitation” load cases In order to check the structure response. Firstly we would check that the analysis has proceeded exclusively along 1 DOF and along the X direction of motion. That could be done by checking the Modal load participation ratios, as represented in table 13.

Output Case	Item	Dynamic[%]	Static[%]
MODAL	UX	100	100
MODAL	UY	0	0
MODAL	UZ	4.72E-13	4.77E-10

Table 14-Load participation ratio of Modal analysis (Computers and Structures Inc.)

The Analysis was proceeded in 100[%] along the X direction as we expected, with negligible influence along the Z direction and null along the Y direction. Therefore the motion quantities for each floor can be calculated in terms of maximum displacement[m], velocity $[\frac{m}{s}]$ and acceleration $[\frac{m}{s^2}]$.

Floor	Displacement[m]	Velocity$[\frac{m}{s}]$	Acceleration $[\frac{m}{s^2}]$	Type	Output Case
1	0.006117	5.1596	5.26036	Max	Base Excitation
2	0.017871	5.1596	8.83665	Max	Base Excitation
3	0.024004	5.1596	11.43017	Max	Base Excitation

Table 15- Maximum relative displacement ,velocity and acceleration due to Earthquake application (Computers and Structures Inc.)

The acceleration function traces spectrum is showing graphically in figure 40, for each of the three floors.

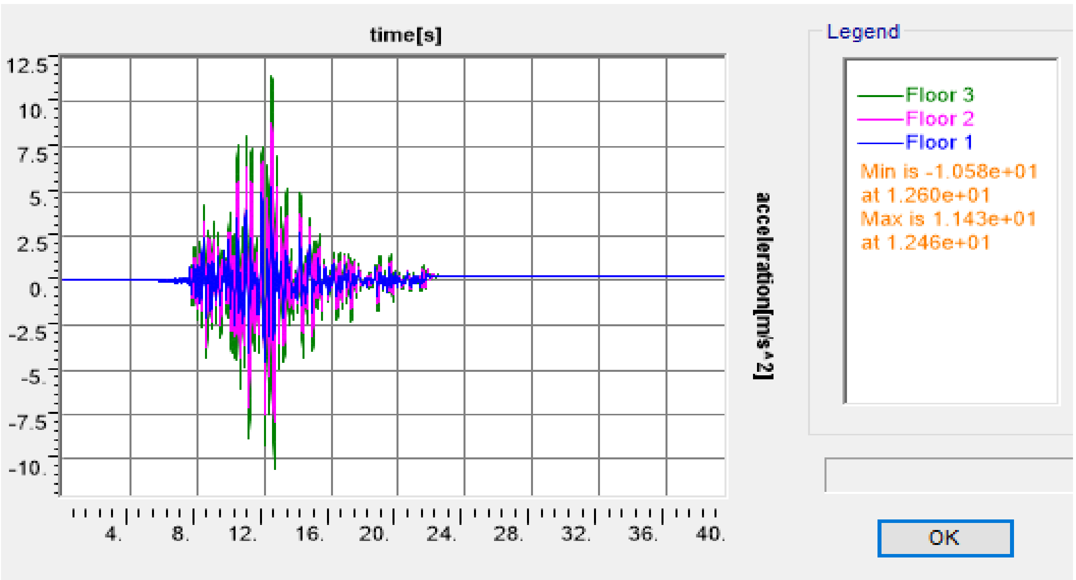


Figure 40-Accelertaion[m] versus time [s] response specturm (Computers and Structures Inc.)

6.4.3. Time history analysis in MATLAB using Convolution Method (Chopra 2012)

The dynamic response of a structural system subjected to an earthquake excitation may be validated in the MATLAB model. In order to implement a well time history code in MATLAB, We have to consider our earthquake as a non-periodic function. Therefore, the response in terms of displacement and acceleration can be developed by using the convolution integral method –considering small impulse intervals between the:

- IRF (Impulse Function Response) -$h(t)$
- The earthquake excitation “Kobe” function/signal - $\ddot{u}g(t\}$

Before starting with the procedure, we have to clarify the decoupling aspect. Considering the stiffness matrix; because non-diagonal entries are different from zero, the matrix is not diagonal. Therefore, there is a mutual influence between the floors' stiffness. In order to use the superposition effect we have to decouple our MDF Equation into a system of three SDOF equations. Basically, we transform our system from geometrical coordinates $p(t)$ into modal/fictitious coordinates $q(t)$. The modal motion equation can be seen as the following (Chopra 2012) :

$$\ddot{\mathrm{q}} + 2\zeta\omega\dot{\mathrm{q}} + \mathrm{w}^2\mathrm{q} = \Gamma\mathrm{p(t)}$$

$$\Gamma = [\mathrm{F}]^{\mathrm{T}}[\mathrm{M}]\mathrm{d}$$

Γ Is the modal participation factor that represents the importance of specific floor to a specific modal shape. Where d is the carrying over unit vector (3,1) that help as transform from scalar to vector properties.

For classic damping models, A complex-conjugate pair of eigenvalues are obtained by the problem of a Free vibration MDF system. The understating of these Eigenvalues is important in order to use the convolution integral method correctly. Practically, one of their components is the damping circular frequency d, that is used to expand our signal from finite to infinite:

$$\omega\mathrm{d} = \omega\mathrm{n}\sqrt{1-\zeta^2}$$

Where $\omega\mathrm{n}$ is the natural circular frequency in $[\frac{rad}{s}]$ and ζ is the mean value of the 3 damping ratios. The implementation in MATLAB is the following:

```
sW=sqrt(E);%natural frequency[rad/second]
sW=diag(sW);
Wd = sW*((1-Z^2)^0.5);
```

The main component of the convolution method is the IRF for the deformation of the n-th mode SDF system. It can be represented by $h(t)$, a vector of unit impulse response functions that practically describe in which way the system damping will decay by time. Considering an SDF system with vibration properties –natural frequency ωn and damping ratio ζn of the n-th mode of the MDF system (Chopra 2012). The IRF can be seen analytically:

$$h(t) = -\frac{1}{\omega d} e^{-\zeta n \omega n t} \sin \omega d t$$

We define Z as the vector that contains the 3 damping ratios:

```
Z1=0.17913; %damping ratio 1
Z2=0.15259; %damping ratio 2
Z3=0.1597; %damping ratio 3
Z=[Z1,Z2,Z3];
```

The Convolution integral for each SDF is the response of the whole MDF system to arbitrary ground acceleration. It can be expressed as :

$$u(t) = \int_0^t \ddot{u}g(\tau)h(t-\tau)d\tau$$

The implementation in MATLAB of this procedure can be done by the function “conv”. Afterwards, a transforming from modal to geometrical coordinates was done.

```
for i=1:3
    ht(i,:)=(1/Wd(i))*(exp(-Z(i)*sW(i)*time)).*sin(Wd(i)*time);
     R(i,:)=-GAMMA(i)*acceleration;
    Displacement_modal(i,:)=conv(ht(i,:),R(i,:))*ts;
end
Displacement_geometrical=F*Displacement_modal;
```

The geometrical displacement response spectrum of each floor can be seen graphically in figures 41-43:

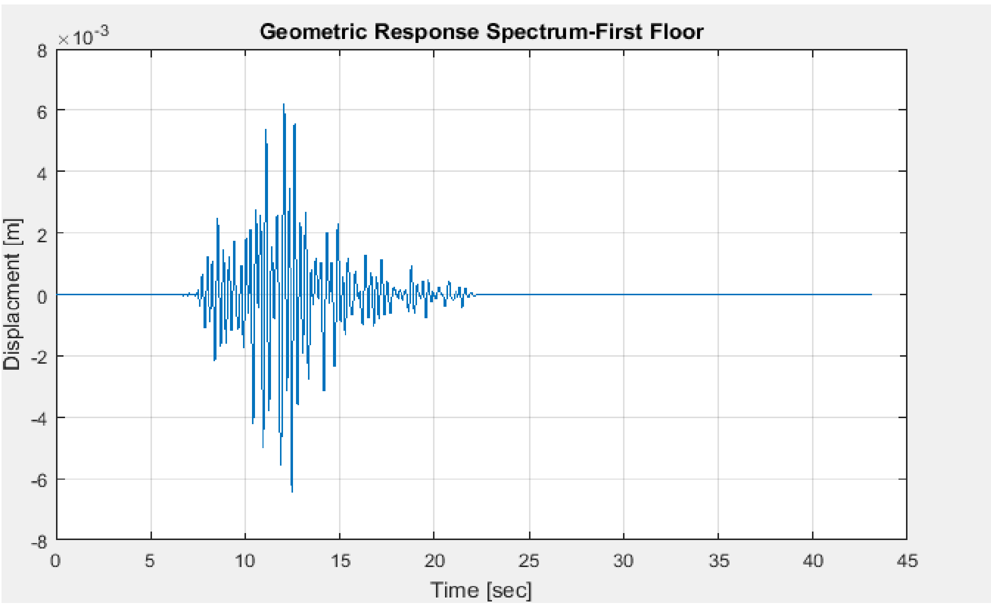


Figure 41-Displacment time history of the 1st floor (MathWorks)

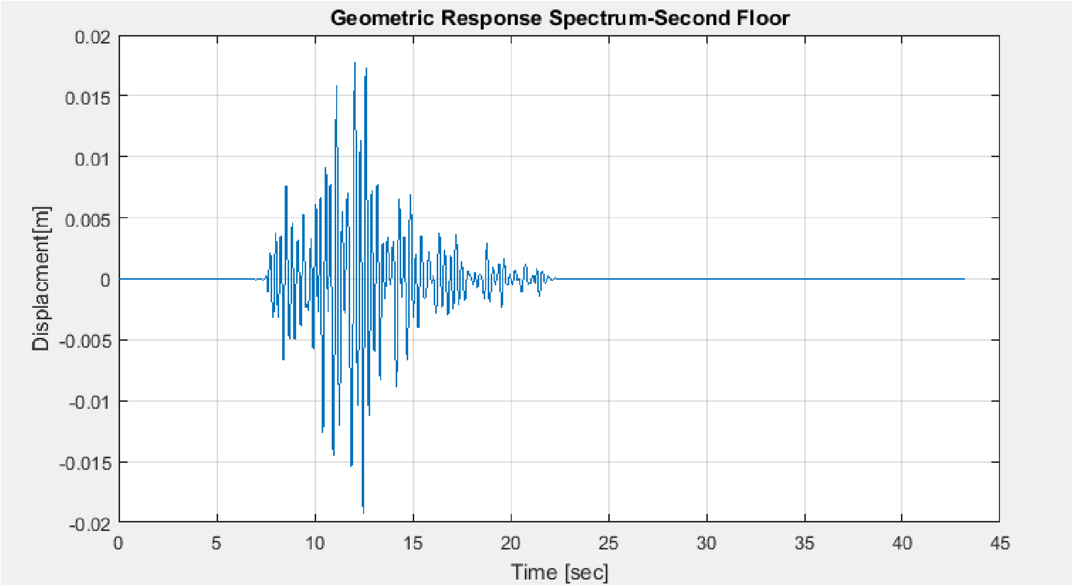


Figure 42-Displacement time history of the 2nd floor (MathWorks)

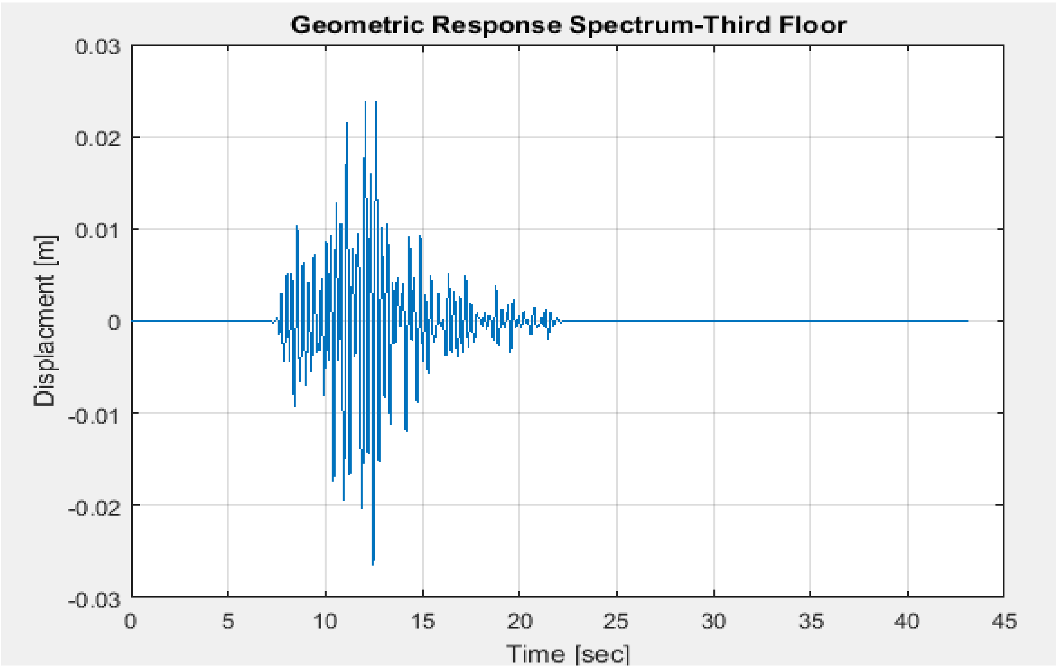


Figure 43-Displacement time history of the 3rd floor (MathWorks)

To adequately arrange the comparison between the numerical models, the response in terms of maximum displacement during the whole application time is represented graphically and numerically:

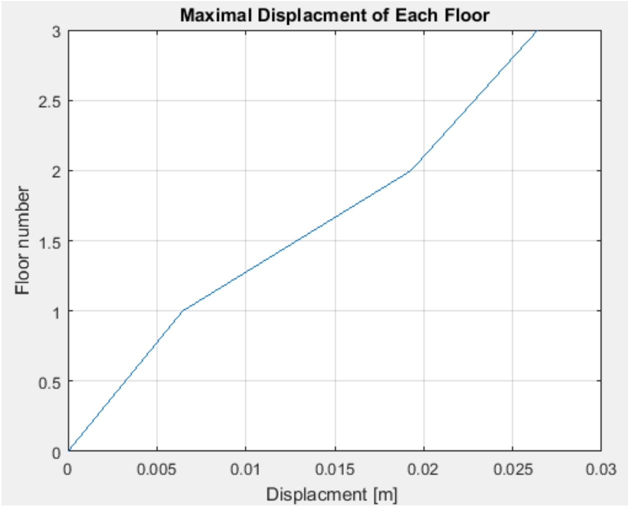


Figure 44-Maximum displacement of the floors undergo “Kobe” earthquake (MathWorks)

floor	ground	1st	2nd	3rd
Displacement[m]	0	0.0064	0.0192	0.0264

Table 16-Maximum displacement of the floors undergo “Kobe” earthquake (MathWorks)

6.5 Comparison between the models

The last step in the validation procedure is to check if the numerical models and the experimental one respond in the same way to dynamic excitation. The relative error between the motion quantities of the different models was defined as the parameter of control. We consider the experimental results as those that approximate the real dynamic behavior.

Firstly, we checked the response in terms of maximum acceleration of the experimental and SAP2000 models. The results represent a pleasing correlation between the models; the acceleration of each floor is quite the same for both models and with relatively less than 5[%] error rate. We can conclude that SAP2000 demonstrate exact dynamic behavior as our real model.

Max Acceleration[m/s^2]			
	1st Floor	2nd Floor	3rd Floor
Experimental	5.1500	9.2400	11.7500
SAP2000	5.2604	8.8367	11.4302
Error[%]	2.1	4.4	2.7

Table 17-Maximum Acceleration of the floors, SAP2000 and Experimental models. (Computers and Structures Inc.)

Secondly, we compared the response in terms of maximum displacement of the MATLAB and SAP2000 models. Considering the biggest model displacements of the 3rdfloor, the difference between them is about 2[mm] and therefore negligible. Generally, the displacement of each floor is quit the same for both models and with less than 10[%] relative error. Observing these results, the MATLAB demonstrates the exact dynamic behavior as the SAP200 model.

Max Displacement [m]			
	1st Floor	2nd Floor	3rd Floor
SAP2000	0.0061	0.0179	0.0240
Matlab	0.0064	0.0192	0.0264
Error[%]	4.4	6.9	9.1

Table 18-Maximum Displacement of the floors, SAP2000 and Matlab models. (Computers and Structures Inc.; MathWorks)

By making an analytical analogy between the acceleration as the second derivative of the displacement, we can say that as the SAP2000 and MATLAB models are correlated, therefore so are the MATLAB and the experimental models. Where these models will undergo identical base excitation, they will respond in the same way.

7. Summary

7.1. Conclusions

The work done in this master thesis required application of technical and analytical skills, together with deep knowledge in the field of structural dynamics. The demonstration was compromised numerical investigations and validation, based on live measurements of a scaled civil structure (experimental set up). Despite the fact that the investigation of inverse reconstruction of the force-time histories were not included in this work, schematic coherent steps were explained in order to allows it in the future .We found that before treating the inverse problem, other high importance principles must be clarify.

Designing of the dynamic model was a complex task that required sufficient time and dedication. Basic on the equation of motion, we were started from the evaluation of the mass and stiffness matrix. The unclear behavior of the second one required us to use different numerical methods and manipulation techniques in order to minimize the partial fixity uncertainty. Afterwards, the first experiment was performed in order to determine the modal shapes and natural frequencies of the structure. Elaboration of the measurement data was done in order to identify the real system parameters, based on the stochastic subspace identification method.

We figured out that the optimization of the stiffness parameters is a demanding assignment, that required creativity thinking side by side with programming capabilities .We had to tailor an algorithm that perfectly describe the workflow of our case of study .Based on the method of steepest descent ,we approximated an integral for each iteration .This integral was calculating in the contour of a complex plane path, passing through the neighborhood of a saddle point, and brought us the direction that the parameter have to follow in the 12D space in order to reach the minimization function. Because of the fact that the sensitives of our scaled model are far smaller with comparing those of a real civil structure, the optimization algorithm had to run much more time as we expected, in order to evaluate the exact parameters.

By observation, we gave a priority to the stability of the first mode quantities with respect to the second and the third. Considering the natural frequencies; before the optimization process, the error between the numerical models and the real one was about 7-10[%] for the first mode and 2-3[%] for the other modes. While after the optimization process, we obtained an error of less than 1 [%] for the first mode, where for the other modes the error was increasing. Because of the first mode defines how the structure will interact with the rest of the system around it, by taking most of the applying load , we can be satisfied for the improvement of the first natural frequency. The same analogy was demonstrated also for the modal shapes of the structure.

The selection of the FEM software SAP2000 influenced on the design of the numerical model. The application allowed us having spatial perception, with precise 3D geometrical representation of the scaled model . The possibility of being able to modifying the hinges partial fixity, that theoretically are rotational springs , was essential .The software comfort interface was allowed us to preform static and modal analysis in a clear way. For the modal one, specific load case was assign in order to determine the un damped free vibration mode shapes and frequencies of the system. The animated plotting of the results was also necessary to verify if our model behave as we expected. SAP2000 is well design to operate in a batch mode. We were running the software from the background, updating and importing the input files, in order to satisfy the optimization workflow. The software provides a modern time history analysis as well.

The programing in MATLAB requires us to understand in details each single step in the modal analysis processing, unlike the FEM designing. Therefore, in case of error we could point out the precise problem in the model program. The core operation of the iteration process was based on the use of MATLAB ,where sophisticated algorithm were programed for the approximation of the minimization function and the asymptotic integral .The last application of the software was the implementation of the convolution method ,in order to check the response of the model to a base excitation .

The work done in this thesis was reflected my knowledge, along with new engineering methods and principles that I was learning during the work . From the realization of system identification in Macec ,passing through the using of the steepest descent method ,and finally the time history analysis in 3 different ways. I was improving my theoretical knowledge side by side with programing skills. The practical performances of data measurements, together with laboratory experiments, were undoubtedly important for my professional experience as young engineer.

7.2 Foresee

Reading this thesis, one can understand that the work was finished but not completed. Initially, we defined the computational of an inverse excitation as the main purpose of this thesis .Even though that this argument did not take relevant part of the work, does not imply of shortcoming. In fact, we understood that in order to deal with the inverse problem, a complex model must be well constructed in different stages, and with schematic coherent steps.

Eventually, we decided to add another step for our case of study, by controlling the behavior under a real base excitation application. As mentioned in paragraph 6, this step was important for the validation of the numerical models.

Willingly, one can develop and apply substance techniques for the computational of reverse exaction, starting from the ending point of my work. A recommended literatures are the “identification of external structural loads from measured harmonic responses (S.E.S. Karlsson)” , and “review of the indirect calculation of excitation forces from measured structural response data (Dobson Rider)”.

Theoretical recommended numerical techniques are the frequency domain methods (FDM) and time domain methods (TDM). Both permit the calculation of excitation forces, starting from known response data and excitation boundary conditions. The idea behind them seems to be simply, just by reversing the ordinary process of the evaluation of the structural response .However, “whilst a defined forcing history will produce a unique set of responses the inverse is not true unless the location and the form of the forcing are known in advance” (Dobson Rider).

As a vision for the future ,a powerful tool can be integrating the structural dynamic into the world of building information modeling (BIM).For instance ,better interoperability between Matlab and FEM software’s will save time and improve results accuracy .

Bibliografia

©2019 Encyclopædia Britannica , www.britannica.com

Alonso, Marcelo; Finn, Edward J. (1992): Physics. Wokingham England, Reading Mass.: Addison-Wesley Pub. Co.

Chopra, Anil K. (2012): Dynamics of structures. Theory and applications to earthquake engineering. 4th ed. Upper Saddle River N.J.: Prentice Hall.

Clough, R. W.; Penzien, Joseph (2003): Dynamics of structures. 2nd ed.

Computers and Structures Inc.: SAP2000: CSI.

DEUTSCHE EDELSTAHLWERKE GMBH: Technical Bulletin.

Dobson Rider: A review of the indirect calculation of excitation. Dobson Rider.

Hans-Erich Gemmel & Co. GmbH: technical bulletin, https://www.gemmel-metalle.de/werkstoffe/legierungsbeschreibung-almgsi0-5f22.html.

HBM: HBM.

MACEC: MACEC: KU Leuven.

Massimo Coradi: A Short Account of the History of Structural Dynamics between the Nineteenth and Twentieth Centuries.

MathWorks: MATLAB®: MathWorks.

PCB Piezotronics: Installation and Operating Manual. ICP® Impact Hammer.

PCB Piezotronics: Model TLD356A17. Installation and Operating Manual.

PEETERS, B. AND ROECK, G. D.: REFERENCE-BASED STOCHASTIC SUBSPACE IDENTIFICATION FOR OUTPUT-ONLY MODAL ANALYSIS.

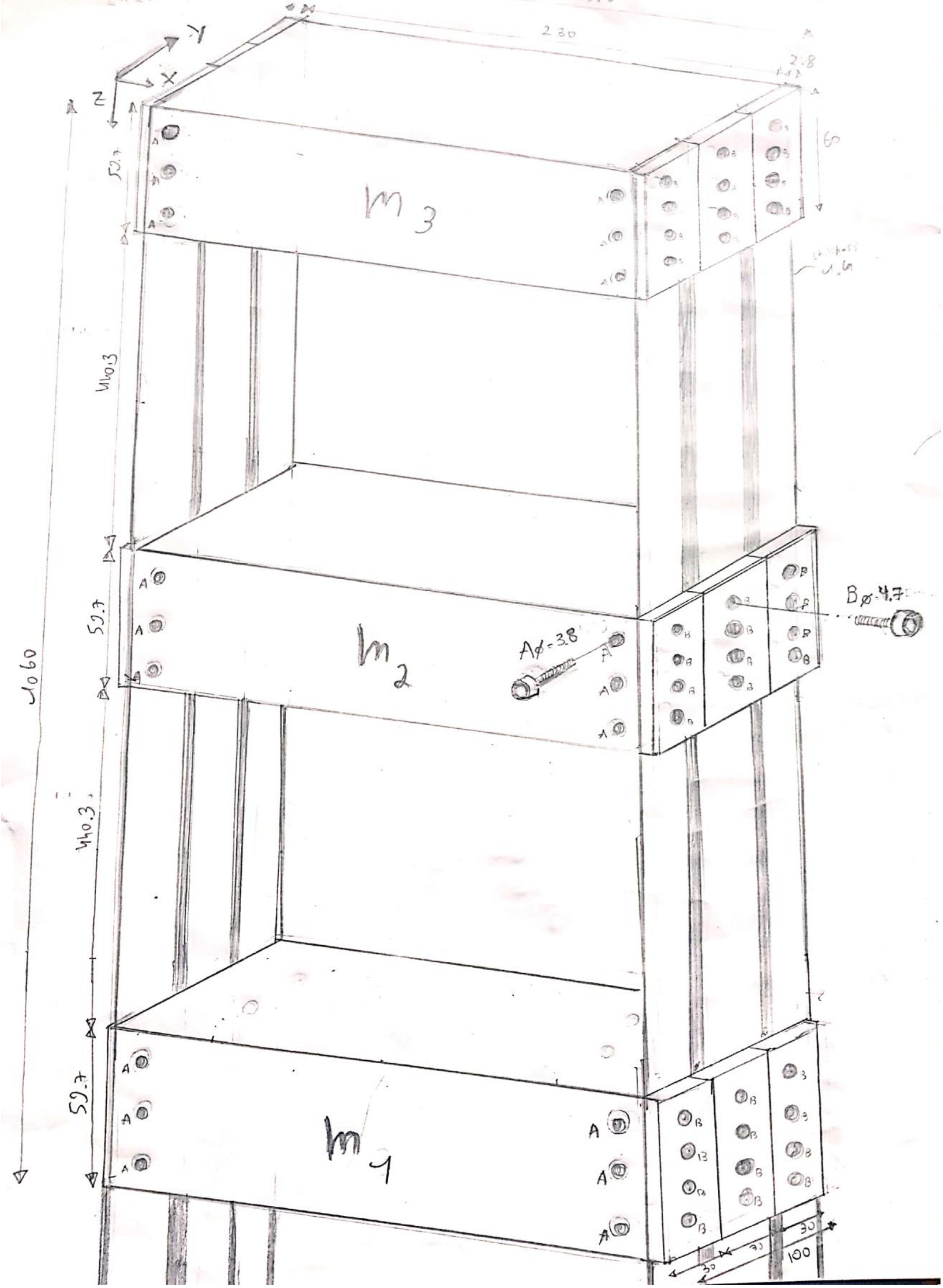
REYNDERS, SCHEVENELS and DE ROECK: Macec USER'S MANUAL JULY 2014 REPORT BWM-2014-06. MACEC 3.3 A MATLAB TOOLBOX FOR EXPERIMENTAL AND OPERATIONAL MODAL ANALYSIS USER'. KU LEUVEN.

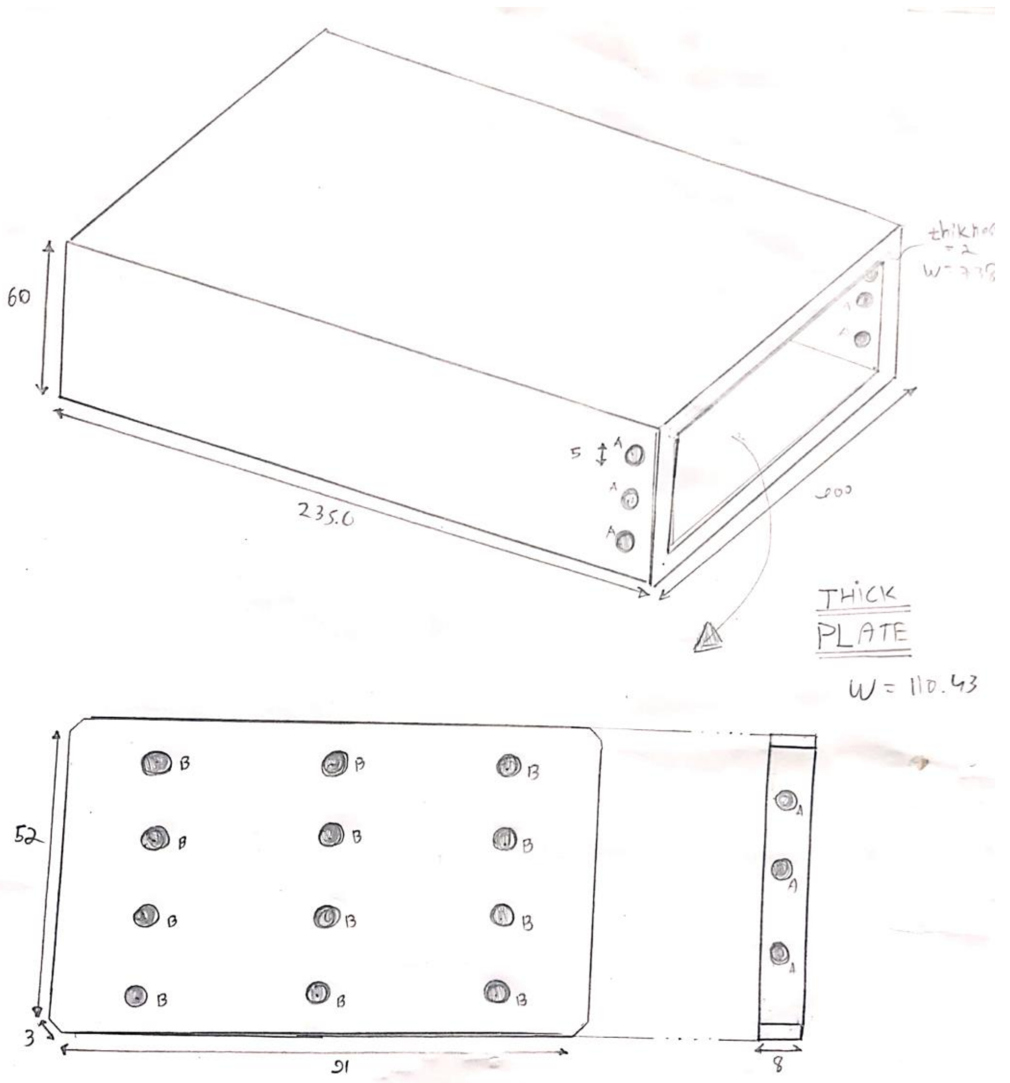
S.E.S. Karlsson: IDENTIFICATION OF EXTERNAL STRUCTURAL LOADS FROM MEASURED HARMONIC RESPONSES.

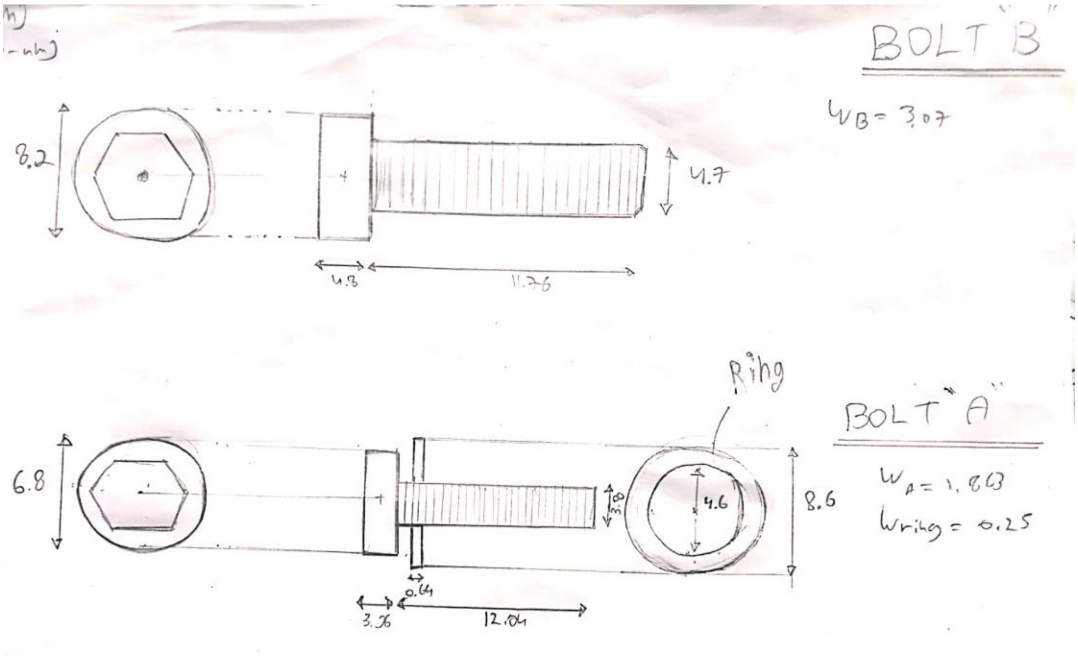
Weber, Hans-Jurgen; Arfken, George B. (2004): Essential mathematical methods for physicists. San Diego, CA: Academic Press.

Wölfel: BD.5 and BD.10 – Electrodynamic shaker systems for universal vibration excitation. Wölfel. Disponibile online in www.woelfel.de.

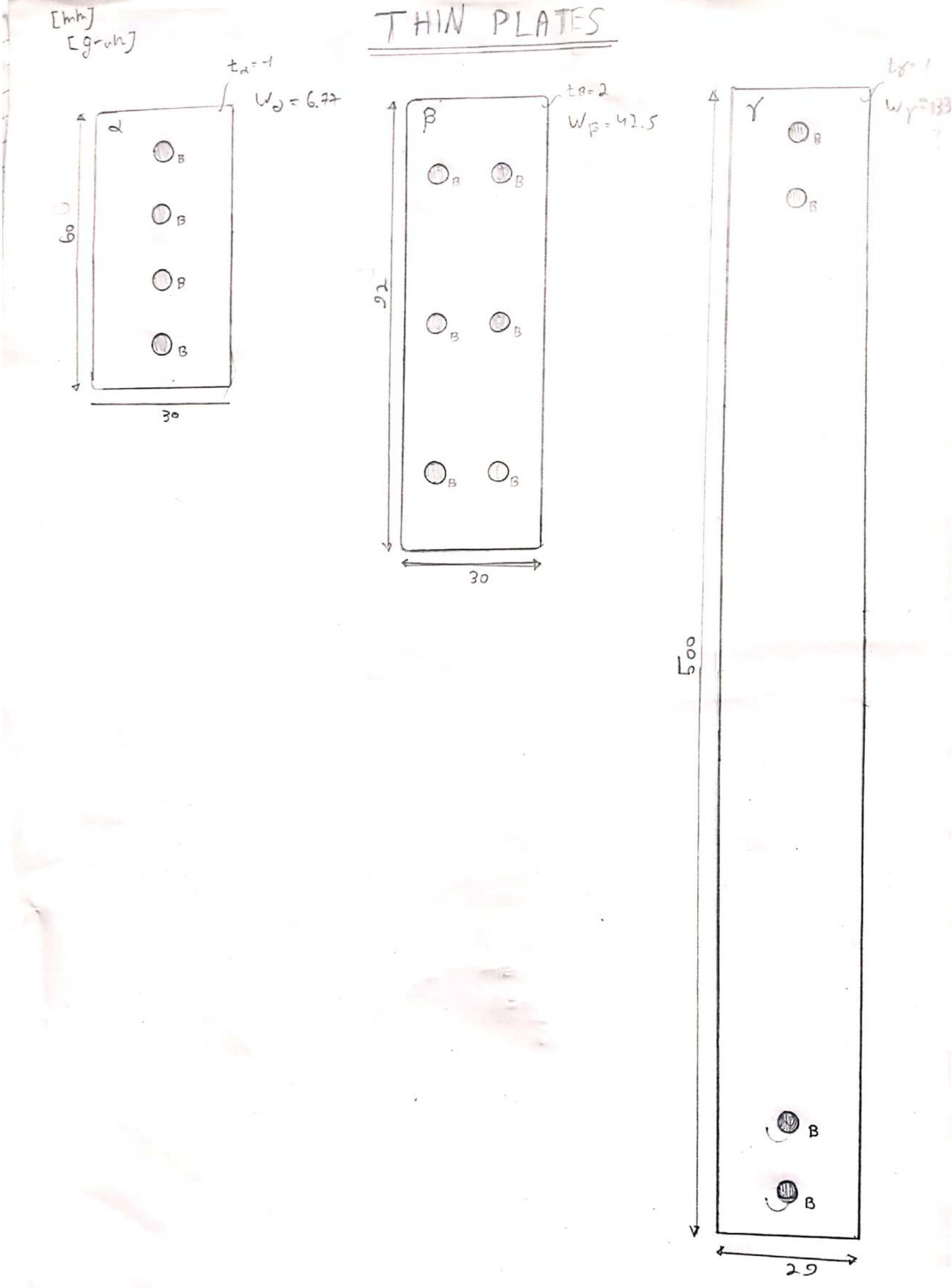
Appendix 1 –Hand sketches of the model elements




60
235.0
5
A
THICK
PLATE
W = 110.43
52
B
3
91
8


BOLT "B"
$W_B = 3.07$
8.2
4.7
4.8
11.76
Ring
BOLT "A"
6.8
3.8
4.6
8.6
$W_A = 1.863$
$W_{ring} = 0.25$
0.64
3.36
12.04

THIN PLATES



[mm]
[gm]
$t_\alpha = 1$
$W_\alpha = 6.77$
α
60
30
$t_\beta = 2$
$W_\beta = 42.5$
β
92
30
γ
500
29
B

Appendix 2- Weighting of Elements

Table 17 –calculation of the total mass of the model

Element	Quantities	Weight[g]	Tot For Element[g]	Standard deviation [g]
box	3	738	2214	
thick plate	6	110.43	662.58	
thin plate α	18	6.77	121.86	
thin plate β	2	42.5	85	
thin plate γ	18	133	2394	
bolt A	36	1.863	67.068	
bolt B	72	3.07	221.04	
ring	36	0.25	9	
total	191		5774.548	30.2332356
			weight total [kg]	5.774 ±0.03

Table 18 –calculation of mass m1

m1	weight[g]	quantity	tot[g]	Standard deviation [g]
box	738	1	738	
thick plate	110.43	2	220.86	
thin plate α	6.77	6	40.62	
thin plate β	42.5	0	0	
thin plate γ	133	6	798	
bolt A	1.863	12	22.356	
bolt B	3.07	24	73.68	
ring	0.25	12	3	
sum		63	1916.516	30.10342857
			1.946619429	

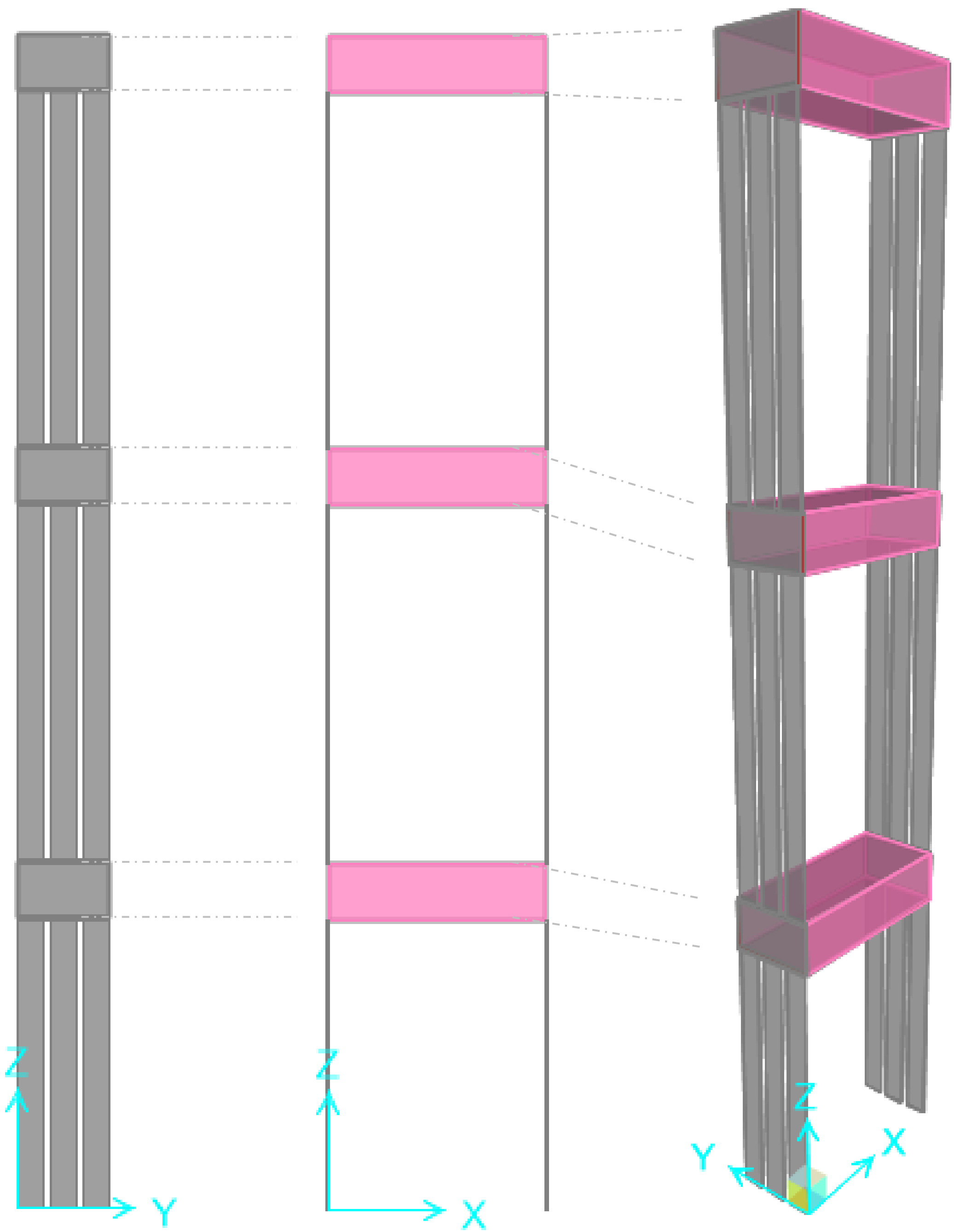
Table 19 –calculation of mass m2

m2	weight[g]	quantity	tot[g]	Standard deviation [g]
box	738	1	738	
thick plate	110.43	2	220.86	
thin plate α	6.77	6	40.62	
thin plate β	42.5	0	0	
thin plate γ	133	6	798	
bolt A	1.863	12	22.356	
bolt B	3.07	24	73.68	
ring	0.25	12	3	
sum		63	1896.516	30.10342857
tot[kg]			1.926619429	

Table 20 –calculation of mass m3

m3	weight[g]	quantity	tot[g]	Standard deviation [g]
box	738	1	738	
thick plate	110.43	2	220.86	
thin plate α	6.77	6	40.62	
thin plate β	42.5	2	85	
thin plate γ	133	3	399	
bolt A	1.863	12	22.356	
bolt B	3.07	24	73.68	
ring	0.25	12	3	
		62	1582.516	25.52445161
			1.582516	0.025524452
tot[kg]			1.608040452	

Appendix 3- FEM model in SAP2000



Statutory Declaration

I herewith formally declare that I, Paz Levi, have written the submitted thesis independently pursuant to § 22 paragraph 7 of APB TU Darmstadt. I did not use any outside support except for the quoted literature and other sources mentioned in the paper. I clearly marked and separately listed all of the literature and all of the other sources which I employed while producing this academic work, either literally or in content. This thesis has not been handed in or published before in the same or similar form.

I am aware, that in case of an attempt at deception based on plagiarism (§38 Abs. 2 APB), the thesis would be graded with 5,0 and counted as one failed examination attempt. The thesis may only be repeated once.

In the submitted thesis the written copies and the electronic version for archiving are pursuant to § 23 paragraph 7 of APB identical in content.

Darmstadt, Germany

19/11/2019 Paz Levi ______________________