Control of vibration in CLT floor structures using FEM sofware

HAMK
HÄMEEN AMMATTIKORKEAKOULU
HÄME UNIVERSITY OF APPLIED SCIENCES

Bachelor's thesis

Degree program in Construction Engineering

Autumn 2021

Alexey Klimentyev

HAMK
HÄMEEN AMMATTIKORKEAKOULU
HÄME UNIVERSITY OF APPLIED SCIENCES

Construction Engineering degree program Abstract
Author Alexey Klimentyev **Year** 2021
Subject Control of vibrations in CLT floor structures using FEM software
Supervisors Cristina Tirteu, Jaakko Yrjola, Juuso Salonen

The thesis aimed to check whether FEM is a good tool for checking the vibration performance of CLT slabs supported by steel-concrete composite beams DELTABEAM® by comparing the results from Autodesk Robot, hand calculations, and measurements taken on site. Another purpose of the thesis was to develop instructions for running a vibration analysis of CLT slabs in Autodesk Robot Structural Analysis Professional software.

Alongside FEM analysis, hand calculations were done according to SCI PUBLICATION P354 to review the accuracy of FEM results. To support the theoretical part, a concrete study case was taken – the floor structure of Hopealaakso kindergarten located in Helsinki, where the measurements have been taken, analyzed, and compared to come up with a conclusion.

The results of the study case proved that FEM is a reliable tool for checking the vibration performance of CLT floors supported by DELTABEAM®. The outcome of the thesis is a written instruction on how to utilize Autodesk Robot FEM program for carrying out the floor vibration analysis and it can be used by the employees of Peikko Finland Oy.

Keywords Composite beam, CLT slab, FEM, vibration analysis
Pages 47 pages and appendices 58 pages

Contents

1 Introduction 1

1.1 Background 1

1.2 Objectives 1

1.3 Scope and limitations 2

2 Study case - Hopealaakso kindergarten 2

2.1 Data about the structure 2

2.1.1 DELTABEAM® 3

2.1.2 CLT slabs 4

3 Robot Structural Analysis professional software 5

3.1 FEM 5

3.1.1 The Mesh 7

3.1.2 Basics of FEM calculation process 7

3.2 Modal analysis 8

4 Dimensioning of the floor structure in Autodesk Robot 9

4.1 Initial data 10

4.2 Nodes and their coordinates 11

4.3 Support conditions 13

4.4 Dimensioning of slabs and beams of the building 14

4.5 Design of steel-concrete composite beams profiles 16

4.6 Design of CLT-concrete composite slabs 20

4.7 Mesh generation 25

4.8 Analysis types 26

4.8.1 Modal analysis 27

4.8.2 Footfall analysis 28

4.9 Calculation and results of FEM model 29

4.9.1 Results from Modal analysis 30

4.9.2 Results from Footfall analysis 31

5 Measurements acquired from the site 34

5.1 Overview of the results 35

5.2 Graphical data from the accelerometers 36

6 Hand calculations by publication SCI P354 37
6.1 Accelerations and steady-state responses of the floor 37
6.2 Acceleration and transient responses of the floor 40
7 Overview of the results 42
7.1 Conclusions 42
7.2 Acceptability of the vibration 44
8 Summary 44
References 46

Annex

1 Introduction

A combination of CLT slabs with composite beam DELTABEAM® has been considered a reliable and functional solution for floor structures. CLT is a cross-laminated timber floor, and composite beam is a structural element which consists of two or more materials with different physical properties. This system is widely used for various buildings in Finland and around Europe. The design of the CLT floors requires the analysis of the floor behavior when it is subjected to human-induced vibration, which are vibrations caused by human footfall. That analysis can be done in FEM software Autodesk Robot.

1.1 Background

The research for the thesis is based on the theoretical knowledge of floor vibration according to the publication SCI P354. Hopealaakso kindergarten is taken as a study case for the thesis, where measurements have been taken. A model of that building is designed in Autodesk Robot to conduct the modal and footfall analysis. From these analyses, natural frequencies and response factors are acquired. The same analysis is done by hand according to SCI P354.

1.2 Objectives

The objective of the thesis is to check whether Autodesk Robot is a good tool for checking vibration performance of CLT slabs supported on composite beams DELTABEAM®, and make a guide for vibration analysis in FEM. That is done by conducting modal and footfall analysis of the study case building in the program and by hand calculations. Then, the values of natural frequencies, accelerations, and response factors retrieved from FEM, hand calculations, and measurement are compared to conclude whether Autodesk Robot is an acceptable tool for vibration check.

1.3 Scope and limitations

The thesis is limited to calculations of the study case with two beam lines supporting CLT slabs. CLT slabs are high-frequency floor structures with concrete topping. Beam supports are considered to be pinned.

2 Study case - Hopealaakso kindergarten

The main point of this thesis is to check how close are the results of vibration analysis acquired from FEM software to the actual situation on the site. Therefore, the FEM model of an already existing building has been created for the investigation to be maximally correct. In the model, only slab and beam structures were included, as the vibrational behavior of slabs is the point of interest. Hopealaakso kindergarten is a building located in Helsinki (Figure 1). Beams in that project were designed and manufactured by Peikko Finland Oy in 2020. Beams were designed so that they were able to withstand the permanent and variable loads and the deflection limit for vibration of slabs. They were used on the first floor of the building; the total amount is 11 beams.

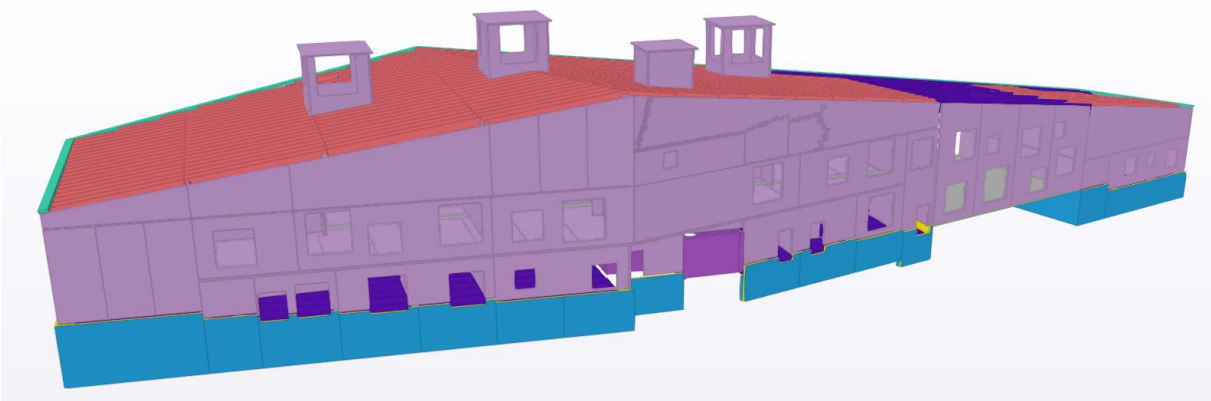

Figure 1. Model of the building made in Tekla

2.1 Data about the structure

The structure component that is of the main interest is a slim-floor composite system. That was the component where the vibration analysis was carried out. A composite system is a

structural element that consists of two or more parts with different materials and physical properties. From a structural point of view, these materials perform better together than separately. The crucial requirement to form a composite is a proper structural connection or bond between the elements. Then they will share the best of their mechanical properties and contribute to effective load distribution. One type of the composite was also used in Hopealaakso kindergarten – slim-floor composite beam DELTABEAM ® with CLT slabs. DELTABEAM® (Figure 2) is a superior composite beam enabling slim-floors for multi-story buildings of any type, whether low-rise or high-rise (Peikko Group, 2021). Due to the high load-bearing capacity, spans up to 16 meters can be achieved without installing additional supporting columns. Cross-laminated timber (CLT) is a planar slab product typically composed of an uneven number of lamination layers, which consist of several finger-jointed and glued panels (Piekko Group, 2020)

Figure 2. DELTABEAM supporting CLT slabs (Peikko Group, n.d.-a).

2.1.1 DELTABEAM®

DELTABEAM® is a slim-floor composite beam that consists of a trapezoid-shaped metal section and concrete which is filled inside the section. A steel beam is manufactured in the factory by welding metal plates together, and on-site, it is completely filled with concrete (Peikko Group, 2021). When the infill concrete has hardened and reached the required strength, both materials form a composite structure. DELTABEAM® can be used with all common types of floor. In Hopealaakso kindergarten the beam was supporting CLT slabs. There are two types of DELTABEAM®: D-type and DR-type. D-type (Figure 3(a) has ledges on both sides of the beam, and it can carry the floor structure on both sides. DR-type (Figure 3(b) has only one ledge and a web on the opposite side, so it can carry the floor structure only on one side.

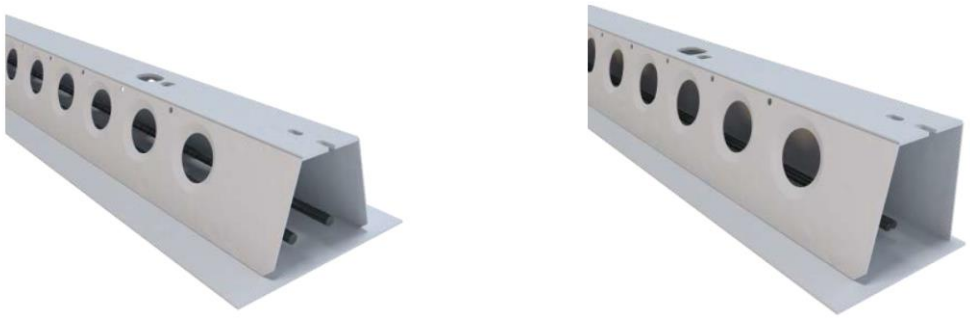

Figure 3. D-type DELTABEAM® (a), DR-type DELTABEAM® (b).

There are two process stages of the beam in which the structural behavior of the beam is different: erection and final stage.

- In the erection stage, the beam and the slabs have been installed, though the concrete hasn't reached its designed strength. The beam still acts as a steel beam, while the load of the floor and construction load are transferred to the DELTABEAM® through ledges. The effect of torsion should be taken into account in this stage, and propping can be used to prevent the beam from rotating at the supports.

- In the final stage, the infill concrete has gained its designed strength, and the beam behaves as a composite member. The loads are transferred from the slabs to DELTABEAM® through the inclined web. Transverse reinforcement, which goes through the beam's web holes, secures the transfer of loads, while the ledges don't have any load coming through.

2.1.2 CLT slabs

CLT slabs are solid timber panels that consist of at least three timber layers glued to each other. Every second layer is installed in the transverse direction to ensure better bonding. The dimensions for the CLT slabs vary up to 3,5 meters in width and 16 meters in length (Figure 4). The wood species used to produce the slabs is spruce with the strength class C24 (Stora Enso, n.d.-b). All layers are glued together with formaldehyde-free PUR adhesive. The size of a board that forms the layer is usually 40 mm thick and 300 mm wide.

The strength and stiffness of the CLT depend on the direction in which the load is applied. Timber has the highest strength in the direction of fibers, so slabs should be designed in a way that the load is transferred in the load-bearing direction. CLT slabs can also form a composite if the concrete layer is cast on top. The concrete topping is nailed or screwed to the timber slab forming a shear connection. In that way, the composite slab has higher bending stiffness which allows utilizing longer slab spans.

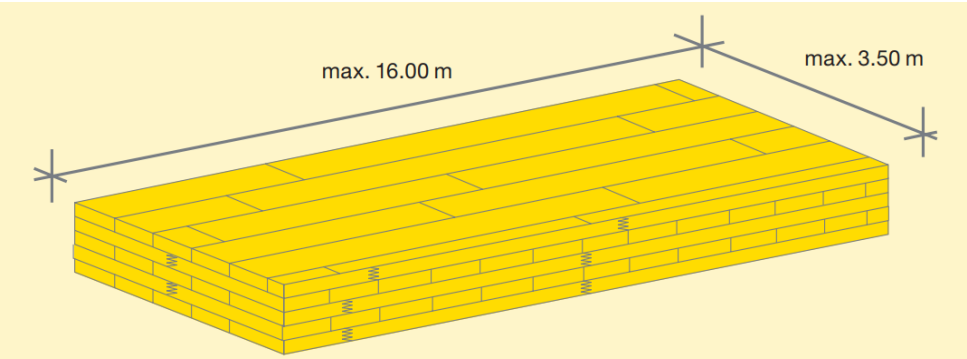

Figure 4. CLT slab.

3 Robot Structural Analysis professional software

Robot Structural Analysis Professional software is a finite element analysis program used to design and analyze structures made from different materials and exposed to any environmental conditions. The structure analysis is done according to the Eurocode and National Annex of the country in question. Robot Structural Analysis Professional is the product of Autodesk, an American software corporation specializing in software products for different purposes ranging from engineering and construction to education and entertainment.

3.1 FEM

Autodesk Robot is based on the finite element method, which is the numerical technique for describing any physical phenomenon. In the case of Autodesk Robot, FEM is primarily used for numerically solving structural mechanics problems. The origin of the finite element method can be traced back to the 1950s when the complexity of mathematical calculations reached a point where it would be almost impossible to do them by hand. Computers made

it possible to analyze a considerable amount of data in a reasonable time. At first, the method was utilized by the aerospace industry, but later on, it was used to do calculations in construction and mechanical engineering.

There are several inputs in the finite element method which are required to generate the model and calculate it. First, node locations are defined. Nodes are certain points in space, and their location is defined by coordinates. They can be used as reference points for structural objects. The support type is assigned to each connection: simple, roller, pinned or fixed. Each support type has a certain number of degrees of freedom which has an impact on the stiffness of the connection. The safest and the most conservative solution is to use pinned supports, if more precise information is not available. Then, structural elements such as columns, beams, walls, and slabs are added and the geometrical and physical properties assigned to them. It is also important to choose the right material and properties for the element.

When geometrical properties are defined, the mesh is created. A more detailed definition of the mesh is given in section 4.1.1. Mesh of the finite element model consists of elements that can have different shapes. They can be quadrilateral (Figure 5 (a) or triangular (Figure 5(b). Quadrilateral elements perform better and give more precise results, while triangular elements are better for modeling complicated shapes. Surface elements are used for two-dimensional objects, and in the case of three-dimensional objects, solid elements like parallelepiped (Figure 5 (c) or pyramid (Figure 5 (d) are used. In the case of simple problems, which do not require accurate calculations, line elements can be utilized (Figure 5(e). The choice of the type of mesh elements depends directly on the accuracy of the result, which needs to be achieved and the type of problem: one-dimensional, two-dimensional, or three-dimensional. In the case of Hopealaakso kindergarten, quadrilateral surface elements have been utilized in the model.

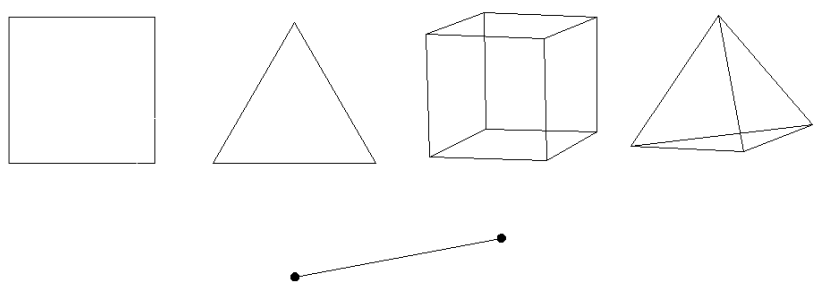

Figure 5. quadrilateral element (a), triangular element (b), parallelepiped (c), pyramid (d), line element (e).

In the next step, the loads are added to the model. They can be acting on a node, member, or surface. In the case of modal analysis, loads must be converted into mass and added to the slab's self-weight. The analysis is done by the direct stiffness method. In this approach, the structure's equations of equilibrium are solved to determine the joint displacements, which are the main unknowns. After displacements are acquired, unknown forces are determined through the force-displacement relations of the member and compatibility considerations. There are several types of analysis in FEM: modal, seismic, spectral, harmonic, footfall, static, etc. Analysis time can be different depending on the power of the calculating machine and the complexity of the model. Finally, the results of the analysis can be viewed. Results can be displayed in diagrams, maps, tables, etc., while the typical outputs which are usually of the main interest are: stresses, forces, bending moments, deflections, support reactions, eigenfrequencies, response factors, etc.

3.1.1 The Mesh

As mentioned in the previous chapter, one of the essential steps needed to run a FEM simulation is defining the mesh. Mesh consists of several elements that make up the overall shape. In other words, the mesh is used to present the 3D object in a group of mathematical points, making the analysis easier. The process of converting the three-dimensional object into mesh is called discretization. The mesh density is chosen depending on how precise the result needs to be or on the power of the calculation machine. The denser the mesh, the more time it takes for the computer to calculate the model.

3.1.2 Basics of FEM calculation process

There are plenty of different outputs which can be extracted from the FEM calculation, though this thesis focus is on the displacements. Each node of a mesh has two main properties of interest: linear displacement and angular displacement. For a node of the 2D object, the number of degrees of freedom is three (displacement along the x-axis,

displacement along the y-axis, and angular rotation about the z-axis), while for the node in 3D, that will be six. The external loads applied to the structure cause nodal forces on the elements. The point of the finite element method is to get the values of nodal displacements due to applied forces, which is usually done by the stiffness method. In this method, the connection between the force and displacement is obtained by the formula below (equation 1).

$$\{F\}=[K]\{U\}$$

Equation 1. Stiffness formula.

Where

- {F} is the nodal force vector that represents the internal force applied to the node.
- [K] is the stiffness matrix of the element. It defines how much each node will displace under certain forces and moments applied.
- {U} is the nodal displacement vector.

To solve the equation, boundary conditions and external loads should also be defined. A boundary condition is a place on a structure where either the external force or the displacement are known at the start of the analysis. The known force or displacement may have some magnitude, or it may be zero. To get the most accurate result possible, input values describing the structure must be correct and precise. These inputs are usually physical properties of the structure, and the typical ones are dimensions, thicknesses, density, modulus of elasticity, Poisson's ratio, coefficient of thermal expansion, and shear modulus. In practice, a calculation of a typical structure contains so many equations that it is only possible to solve it with the modern computer and reliable FEM software like Autodesk Robot.

3.2 Modal analysis

Natural frequency in the modal analysis is found by the Eigen equation, considering the fact that the natural frequency of the damped structure is about the same as the structure is with zero damping. Free vibration happens when the structure can vibrate freely from

external forces. Equation of motion from free vibration of a system without damping is presented below (equation 2)(Pia Johansson, 2009).

$$M\ddot{u} + Ku = 0$$

Equation 2. Equation of motion.

Where:

M – the mass of the structure.

$\ddot{u}$ - the acceleration of the structure.

K – the stiffness of the structure.

u – the displacement of the structure.

However, for the floor structures, which are multi-degree-of-freedom systems - matrices should be used. In that case, the mass and the bending stiffness are replaced with mass and stiffness matrices. Then, the natural frequencies of the structure can be found by the matrix equation (equation 3).

$$(K - \omega^2 M)\Phi = 0$$

Equation 3. Matrix equation.

Where:

K – global stiffness matrix of the structure.

ω – modal circular velocity.

M – global mass matrix of the structure.

ϕ – global displacement vector of the structure.

When a natural circular velocity ω is determined, corresponding eigenmodes can be solved using the global displacement vector.

4 Dimensioning of the floor structure in Autodesk Robot

This part of the thesis describes the steps needed to perform the vibration analysis in Autodesk Robot by presenting the study case example. This research has been done in Robot

Structural Analysis Professional 2022 version. Therefore, there might be some minor changes between different versions, though the procedure and results must be the same. By studying this part of the thesis, the reader will be able to conduct the analysis himself/herself; however, it is recommended to have some basic knowledge of the program and theory of vibration design of floors to perform the analysis.

4.1 Initial data

The first step to do was to choose the type of structure suitable for the project. Structure type depends on the number of available coordinates and nodal degrees of freedom, and it can also be changed during the design process. For example, if 2D frames – structural type is chosen, the coordinate system will consist only of the X and Y axis, while degrees of freedom will be along the X and Y-axis and around the Z-axis. In this project, the vibration analysis of the model should be done in 3D with X, Y, Z coordinate systems and degrees of freedom in all directions and around every direction. Shell–structural type has been chosen as the most suitable one (Figure 6).

Figure 6. Structural types

Another important step to do before starting the modeling was to change the units to the most suitable ones. It is done in the "Job Preferences" window. First of all, it is important to make sure that all the values will be in the metric system, as sometimes the imperial system

is set by default. Then, the precision of the values can be determined, depending on how many decimals are desired to be shown in numbers. For example, a length of 0,64890213 meters can be interpreted as 0,65 m or 0,6 m if better visualization or simplicity is desired; however, if an exactly precise result is needed, it can be left as it is. The materials database was determined depending on the country. In this case, Finland was chosen (Figure 7). Design codes databases were either based on Eurocode or the National Annex of Finland (SFS). The consequence class has been left default – CC2 because, in the case of Hopealaakso kindergarten, the building matches the medium consequence class.

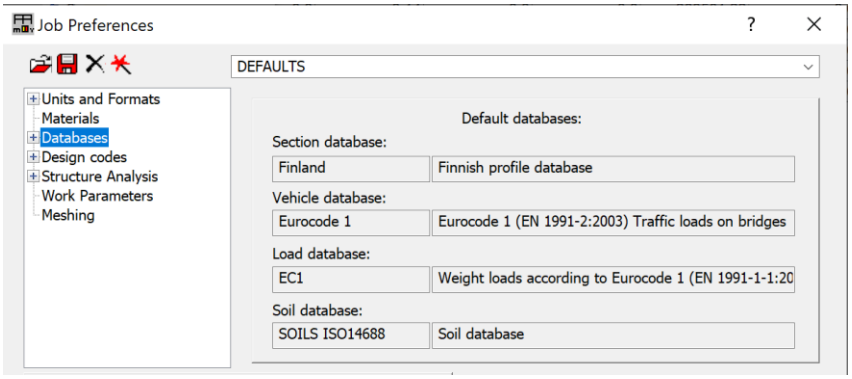

Figure 7. "Job Preferences" window - chosen databases

4.2 Nodes and their coordinates

After all the databases, units, and formats are set as required, the modeling phase can be initiated. The first step needed for modeling the floor structure is a node system, which will make the further design process more convenient. Each node has just two properties: a number and the coordinates (Figure 8).

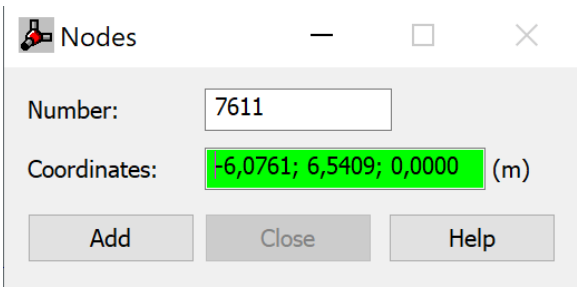

Figure 8. Properties of "Nodes" window

The number is set up automatically, while the coordinates should be typed in. Nodes are added at the coordinates where the edges or beginnings and ends of the beams and the slabs will be located. The same is done for the supports locations. As the point of this project was to model just the floor of the building, which is a plane structure, all the nodes were located on the same level, so only x and y coordinates were altered, while the z level always stayed on zero. The coordinates have been taken from AutoCAD LT 2020 plan drawings and the Peikko Designer Deltabeam model. In general, coordinates don't have to be very precise, as inserting exact values for coordinates consumes much more time, while it will not affect the results that much. The nodes system which has been made for the building can be seen on the screenshot below (Figure 9).

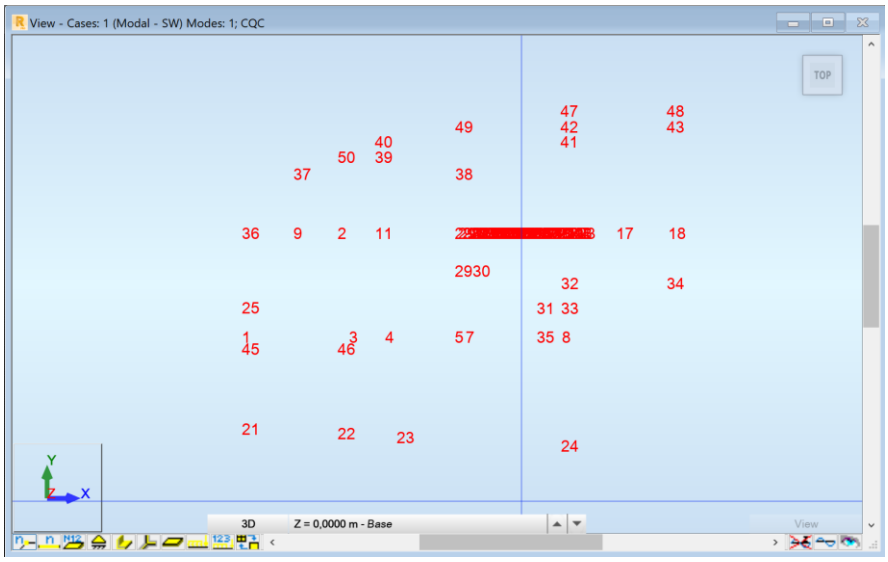

Figure 9. Nodes system

4.3 Support conditions

All the supports were pinned, meaning that movement in all directions is restricted, while the rotations are released in all the directions. That was a quite conservative solution, as in reality, some connections would be much stiffer, like bigger columns or walls connected to columns and beams. In the places where the load-bearing walls support beams or slabs, linear supports were used. There are two ways to model them in Autodesk Robot. The first way is to draw a line and then assign a linear pinned support (Figure 10, a). A line can be drawn using the "Polyline – Contour" menu, which can be found from Geometry > Objects > Polyline – Contour, by assigning the coordinates for the beginning and the end of the line or connecting two nodes. A second way is to copy one node several times in one direction with a little interval. That way, a line of supports located close to one another will be formed (Figure 10, b). It can be done using the "Translation" function, which can be found from Edit > Edit > Move / Copy … . Supports in a line have been added with an interval of 300 mm – same as the mesh element size. Even though the first method sounds easier, it is recommended to use the second method as sometimes linear pinned supports may work wrong, and Autodesk Robot will ignore them. In places where columns support beams, a nodal support is assigned to a node at the connection point (Figure 10, (c). That is quickly done by going to the "Supports" layout in the Layout dialog box and assigning a Nodal Pinned support to each node where it is required just by clicking it. No supports were added in the places where beams are connected with Gerber's joints.

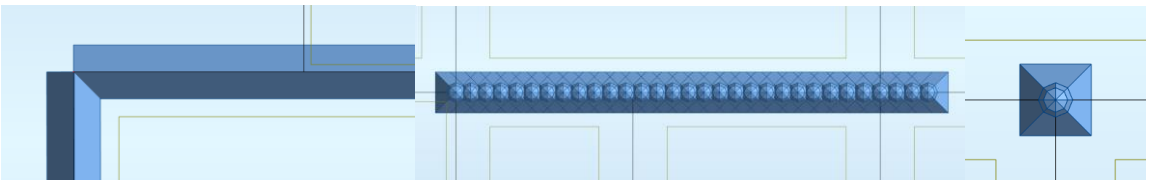

Figure 10. Linear support designed with method 1 (a), linear support designed with method 2 (b), nodal support (c).

In Figure 11, it can be seen how the supports were designed for the whole building. Both methods for making linear supports have been used. However, that doesn't impact the results as both ways work correctly in this case.

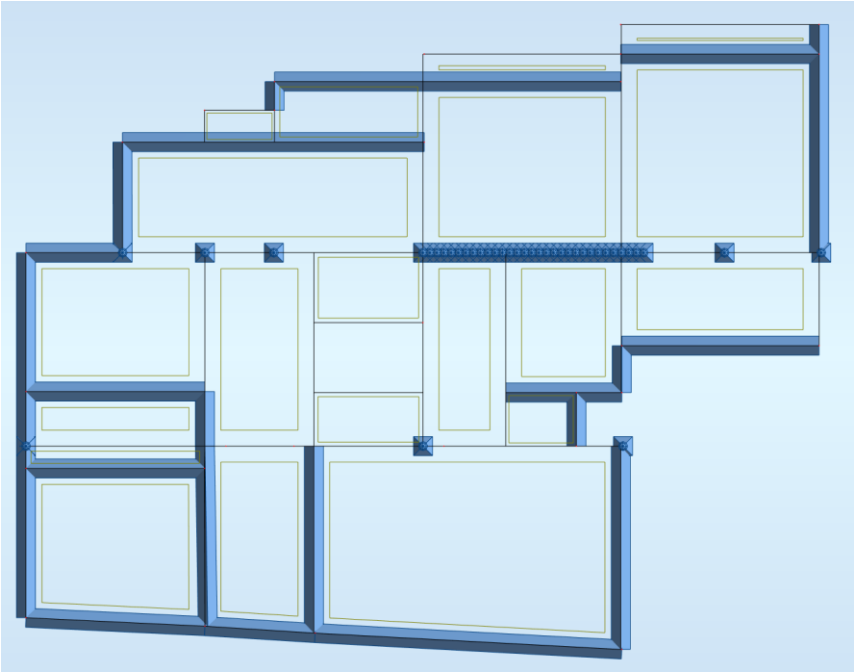

Figure 11. Support conditions for the whole building.

4.4 Dimensioning of slabs and beams of the building

After the nodes have been added, it is time to dimension slabs and beams. Nodes have been added at the special coordinates where corners of the slabs and beginning and ends of the beams would be located, so the dimensioning goes faster. "Beam" dimensioning window can be found from Geometry > Beams … > Beam. It is done either by connecting two nodes by clicking on them or inserting needed coordinates into the "Beginning" and" End" fields (Figure 12). Section type, material, and properties are explained in section 6.5.

"Floor" dimensioning window can be found from Geometry > Floors … > Floor. The thickness of the floor is explained in section 6.6. The model type is chosen to be the shell. As far as all the slabs have a rectangular shape, the definition method is chosen as "Rectangle." It simplifies the modeling process; as for this method, only 3 points should be chosen. Same as in beam dimensioning, slabs can be modeled by connecting three nodes or inserting coordinates into "Point P1", "Point P2", and "Point P3" (Figure 13).

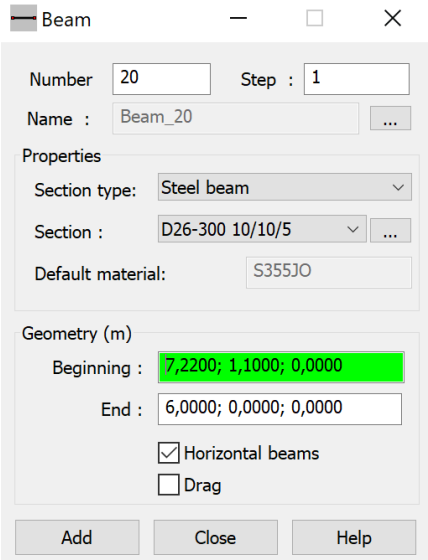

Figure 12. Beam properties

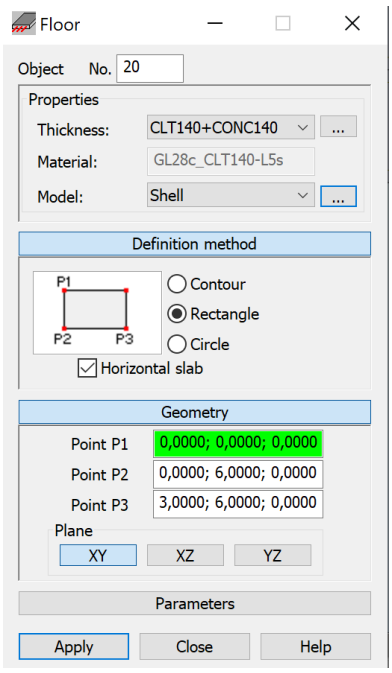

Figure 13. Floor properties

Another essential step about modeling the slabs is to ensure that they don't intersect and that there are no gaps between them if the design needs none. In other words, the slabs should be connected by one same line. Otherwise, that can negatively impact the calculations, and the results might be wrong. The ready floor structure can be seen in Figure 14.

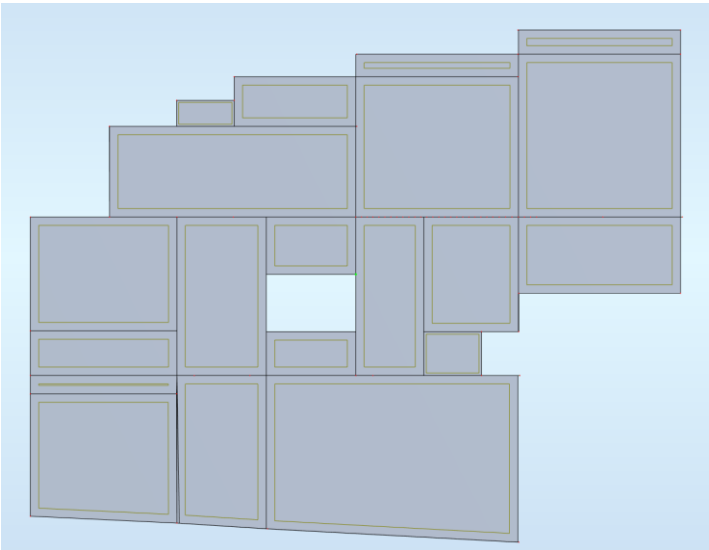

Figure 14. Model of the building after dimensioning the floor structure.

4.5 Design of steel-concrete composite beams profiles

After the beams are dimensioned, they should be assigned a type of section and material. DELTABEAM® composite beams used in this project are not included in the list of standard beam sections in Autodesk Robot, so they should be added manually using the "New section" window. It can be found by pressing the "Sections" button on the left side of the screen and then the "New section." As there is no template for designing a composite beam, it should be added in "User-defined section parameters." In this case, there are just two essential values: Ax – the Cross-section area and Iy – the second moment of an area about the Y-axis. (Figure 15).

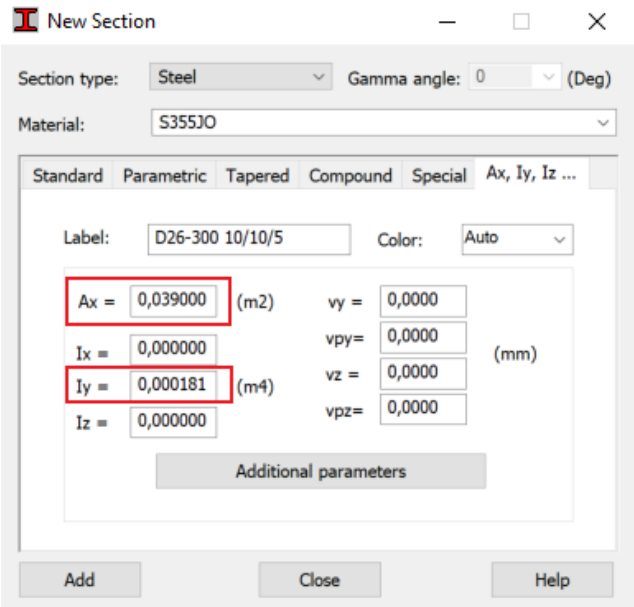

Figure 15. "New Section" window

Before the vibration analysis of Hopealaakso kindergarten, beams had already been dimensioned and calculated, so the values needed for stiffness properties calculation were available. Beam type D26-300 was used in the project, though there was a range of top plates used, so four beam sections and their properties have been calculated. The necessary cross-sectional area of steel can be found by dividing the linear mass of the beam, consisting

of steel and concrete, by the density of steel (Equation 4), as the section material is chosen to be steel S355JO. The beam's mass of steel and concrete is taken from the program Peikko Designer Deltabeam. The density of steel is 77 kN/m^3.

$$Ax := \frac{\left(m_{steel} + m_{concrete}\right)}{\rho_{steel}}$$

Equation 4. The cross-section area of the beam

According to statics laws and publications, the second moment of an area about the Y-axis is found by following steps and equations listed next, according to EN 1994-1-1. Eurocode 4: Design of composite steel and concrete structures. (2005). First, the area of the steel section is calculated by summing up the cross-sectional areas of the top plate, bottom plate, and web plates, excluding web hole gaps. Then, the location of the centroidal axis of the steel cross-section is found (Equation 5).

$$Y_s := \frac{\sum \overrightarrow{\left(y_s \cdot A_s\right)}}{\sum A_s}$$

Equation 5. Location of the centroidal axis of the steel cross-section.

Where:

y_s – is the distance from the bottom to the center of the plate of steel cross-section.

A_s - is the area of the plate of steel cross-section.

After that, a similar procedure is done to find the cross-sectional area of concrete, which is calculated by summing up the area inside the beam, areas between the web and the slab, and two little areas from both sides of the top plate where the concrete is poured. Finally, the location of the centroidal axis for the concrete section is found by the following equation (equation 6).

$$Y_c := \frac{\sum \overrightarrow{\left(y_c \cdot A_c\right)}}{\sum A_c}$$

Equation 6 Location of the centroidal axis of the concrete section

Where:

y_c – is the distance from the bottom to the center of the concrete cast part of the cross-section.

A_c – is the area of the concrete cast part of the cross-section.

The locations of the centroidal axis and section areas were needed to calculate the second moment of area values for steel and concrete sections, as follows (equation 7).

$$I_{X.s} := \sum\left[I_{x.s} + \overrightarrow{\left[A_s \cdot (Y_s - y_s)^2\right]}\right] \quad I_{X.c} := \sum\left[I_{x.c} + \overrightarrow{\left[A_c \cdot (Y_c - y_c)^2\right]}\right]$$

Equation 7. The second moment of an area of steel and concrete sections.

Where:

$I_{x.s}$ – is the second moment of inertia of each steel plate.

$I_{x.c}$ – is the second moment of inertia of each part of the concrete section.

Then, after the second moment of area values are known, axial and bending stiffness for steel and concrete can be calculated. This is done by multiplying the area or second moment of an area by Young's modulus of steel and concrete for the steel section and concrete section (equation 8).

$$EI_s := E_s \cdot I_{X.s} \qquad EI_c := E_c \cdot I_{X.c}$$

$$EA_s := E_s \cdot \sum A_s \qquad EA_c := E_c \cdot \sum A_c$$

Equation 8. Axial and bending stiffnesses of steel section and concrete section.

Where:

E_s – is Young's modulus of steel equal to 210 GPa.

E_c – is Young's modulus of concrete equal to 38 GPa.

From the equation above, the sum of bending stiffness of concrete and steel and the sum of axial stiffness of steel and concrete can be found as follows (equation 9).

$$EI_{sc} := EI_s + EI_c$$

$$EA_{sc} := EA_s + EA_c$$

Equation 9. Steel-concrete bending stiffness and axial stiffness.

After this, the distance between the centroidal axis of the concrete section and steel section is found, which is used to calculate the composite stiffness parameter (equation 10). That factor considers the shear connection of different materials in the composite member.

$$\alpha_i := \frac{e_i^2 \cdot EA_s \cdot EA_c}{EI_{sc} \cdot EA_{sc}} \qquad e_i := \left(Y_c - Y_s\right)$$

Equation 10. Composite stiffness parameter, the distance between concrete and steel section centroids.

Total bending stiffness of the composite beam section (equation 11) is found by the multiplication of sum of 1 and composite stiffness parameter and sum of the bending stiffness of concrete and steel section.

$$EI_{delta} := \left(1 + \alpha_i\right) \cdot EI_{sc}$$

Equation 11. Bending stiffness of composite beam DELTABEAM®

The second moment of area, which is the second input value for section properties, is calculated by dividing the bending stiffness of composite beam by Young's modulus of steel (equation 12).

$$I_y := \frac{EI_{delta}}{E_s}$$

Equation 12. The second moment of area of the composite beam.

In the Hopealaakso kindergarten project only one type of DELTABEAM® was used – D26-300, though the thickness of the top plate was different for some of the beams, to be exact thicknesses were: 10; 15; 25, and 30 millimeters. Due to that, four bending stiffnesses were calculated to be more precise in the calculation. Each beam was given a different name in the design process, so it is possible to identify each one. For example, beam-type name D26-300 15/10/5 means that the thicknesses of the top plate, bottom plate, and the web are 15, 10, and 5 millimeters, respectfully. Results of the calculation are presented in Table 1. They are used as inputs for the analysis in Autodesk Robot.

Table 1. Sectional area and the second moment of an area for beam types

Beam type	A_x [m²]	I_y [m⁴ · 10⁻⁴]
D26-300 10/10/5	0,039	1,808
D26-300 15/10/5	0,04	1,933
D26-300 25/10/5	0,04	2,14
D26-300 30/10/5	0,041	2,224

4.6 Design of CLT-concrete composite slabs

When properties of composite beams are defined, it is time to assign the properties for the slab system. The floor system is chosen to be a composite floor with a 140 mm thick CLT slab and 140 mm thick concrete layer on top. In the main direction, the full contribution of both materials was assumed. However, only 140 mm concrete layer was determined to be load-bearing in the secondary direction. This type of material is called an orthotropic material, meaning that it has different properties along with different directions. As there is no option to choose predefined properties for that floor slab in Autodesk Robot, it has to be added manually using the "New Thickness" window, which can be found from Thicknesses (on the left side of the screen) > New Thickness Definition > New Thickness > Orthotropic (Figure 17). The main direction is defined to be Direction Y, the same as the CLT slabs bearing

direction. In the list of predefined slab geometry types, the "material orthtropy" type is chosen, as it fits the properties of a composite slab. The height of the beam, h, is defined to be 280 millimeters. The first step that shall be done before proceeding to thickness and stiffness calculation is to choose the material. The CLT slab material used in the building is not included in the material library, so it should be added manually. This can be done in the "Material Definition" window, which can be found from Tools (Text menu) > Job Preferences… > Materials > Modification > Timber (Figure 16). Material properties were taken from the publication (Stora Enso Oyj., 2013).

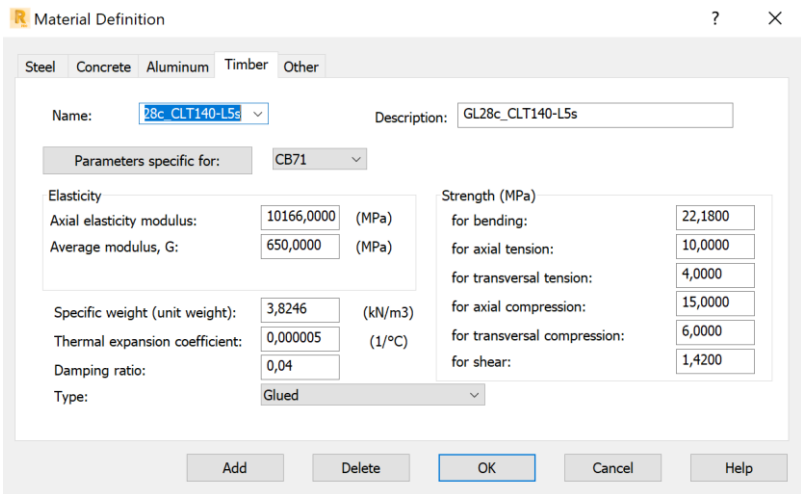

Figure 16. "Material Definition" window

Then, the thickness of the slab is defined. Th is the value of the main interest because it represents the self-weight of the slab. At the same time, Th1 and Th2 are used for calculation when applying thermal gradient loads, so the same input is used for all these values (Autodesk Inc., 2022). Thickness is calculated as follows (equation 13).

$$Th := \frac{\left(\rho_{clt} \cdot 0.14m + \rho_{conc} \cdot 0.14m\right)}{\rho_{clt}}$$

Equation 13. Thickness calculation.

Where:

ρ_{clt} – is the density of the CLT140-L5s.

ρ_{conc} – is the density of concrete.

Then, the stiffness coefficients are defined. In this case, the slab has different physical properties in different directions, so the difference in stiffness is determined by multiplying the chosen material stiffness by the coefficients n1 and n2. The procedure for finding the stiffness in the X direction and Y direction and stiffness coefficients n1, n2 are explained in the following equations (equations 14-19). The first step is to find the axial stiffness of concrete and CLT parts of the slab, calculated as follows (equation 14). As all the slabs have a constant thickness but different widths, they can be taken as 1 meter.

$$Ea_c := a_c \cdot E_c$$
$$Ea_{clt} := a_{clt} \cdot E_{clt}$$

Equation 14. The axial stiffness of concrete and CLT parts of the slab.

Where:

a_c – is the thickness of concrete which is equal to 140 millimeters.

a_{clt} – is the thickness of CLT, which is equal to 140 millimeters.

E_c – is Young's modulus of concrete equal to 38 GPa.

E_{clt} – is Young's modulus of CLT equal to 10166 MPa.

After that, the location of the centroidal axis of the composite slab section is found as follows (equation 15).

$$z_{com} := \frac{Ea_c \cdot z_c + Ea_{clt} \cdot z_{clt}}{Ea_c + Ea_{clt}}$$

Equation 15. Location of the centroidal axis of the composite slab section.

Where:

z_c – is the location of the centroid of the concrete part, determined concerning the top, which is equal to 70 millimeters.

z_{clt} – is the location of the centroid of the CLT part, determined concerning the top, which is equal to 210 millimeters.

Then, to calculate the stiffness of the composite section, the second moments of inertia of the CLT and concrete parts and the total section should be found as follows (equation 16).

$$i_c := \frac{a_c^3}{12} \quad i_{clt} := \frac{a_{clt}^3}{12} \quad i_{tot} := \frac{h_{tot}^3}{12}$$

Equation 16. The second moments of inertia of the concrete part, CLT part, and the full section of the slab.

Where:

h_{tot} – is the thickness of the composite section equal to 280 millimeters.

The generalized Steiner rule finds the total bending stiffness of the cross-section. This is the sum of bending stiffnesses of both material, plus the sum of multiplications of bending stiffness for each material by the distance from the centroid of the composite section to the centroid of each material which is found as follows (equation 17).

$$Ei_{com} := E_c \cdot i_c + E_{clt} \cdot i_{clt} + Ea_c \cdot (z_{com} - z_c)^2 + Ea_{clt} \cdot (z_{com} - z_{clt})^2$$

Equation 17. Total bending stiffness of the cross-section.

Then, Young's modulus for the primary and secondary direction can be acquired. In the main direction, both materials define the stiffness of the cross-section, so Young's modulus is calculated by dividing total bending stiffness by the second moment of inertia of the whole section(Equation 18). On the other hand, in the secondary direction, only concrete defines the stiffness of the cross-section, so Young's modulus is acquired by diving the bending stiffness of concrete by the second moment of inertia of the whole section (equation 18).

$$E_{com.main} := \frac{Ei_{com}}{i_{tot}} \qquad E_{com.sec} := E_c \cdot \frac{i_c}{i_{tot}}$$

Equation 18. Young's modulus for the main and secondary direction.

From the equation above: $E_{com.main}$ = 18050 MPa; $E_{com.sec}$=4750 MPa. Young's modulus for the main and secondary direction should be divided by Young's modulus of CLT, to get the stiffness coefficients n1 and n2 (equation 19).

$$n_1 := \frac{E_{com.main}}{E_{clt}} \qquad n_2 := \frac{E_{com.sec}}{E_{clt}}$$

Equation 19. Stiffness coefficients for main and secondary directions.

From the equation above: n1=1,776; n2=0,467. These input coefficients are used in Autodesk Robot to get the values of Young's modulus for the main and secondary directions (Figure 17).

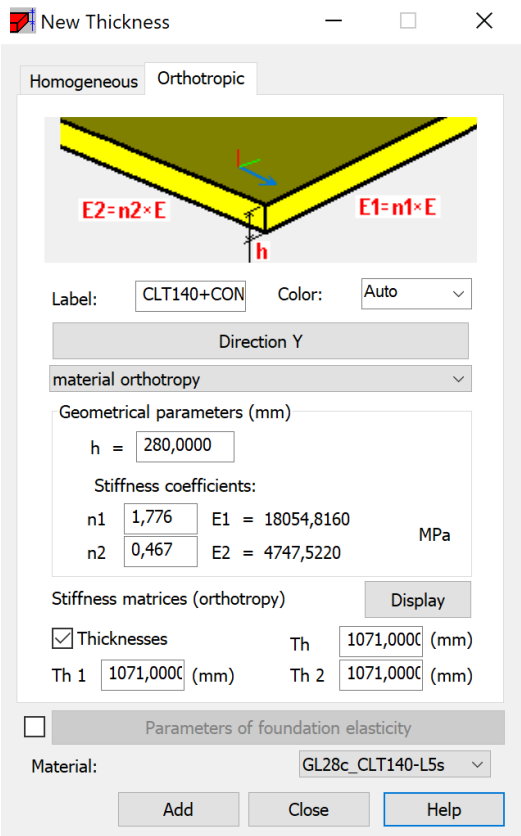

Figure 17. "New Thicknesses" window.

4.7 Mesh generation

After the floor elements of the building have been modeled and properties assigned to them, it is time to make a mesh. The precision of the results and the time it takes for the computer to process the model and analyze it are the key factors when choosing the mesh properties. The denser and more complicated the mesh, the more precise results will be acquired, though it will take longer for the computer to process it. In the case of this project, it took approximately 1-2 minutes for the program to generate the model, analyze it and present the results. Options for the mesh were chosen so that the analysis process doesn't take much time, as it had to be done several times. "Meshing Options" window can be found from the Toolbar > Options of FE Mesh Generation > Meshing options. Meshing methods and parameters were chosen as follows (Figure 18).

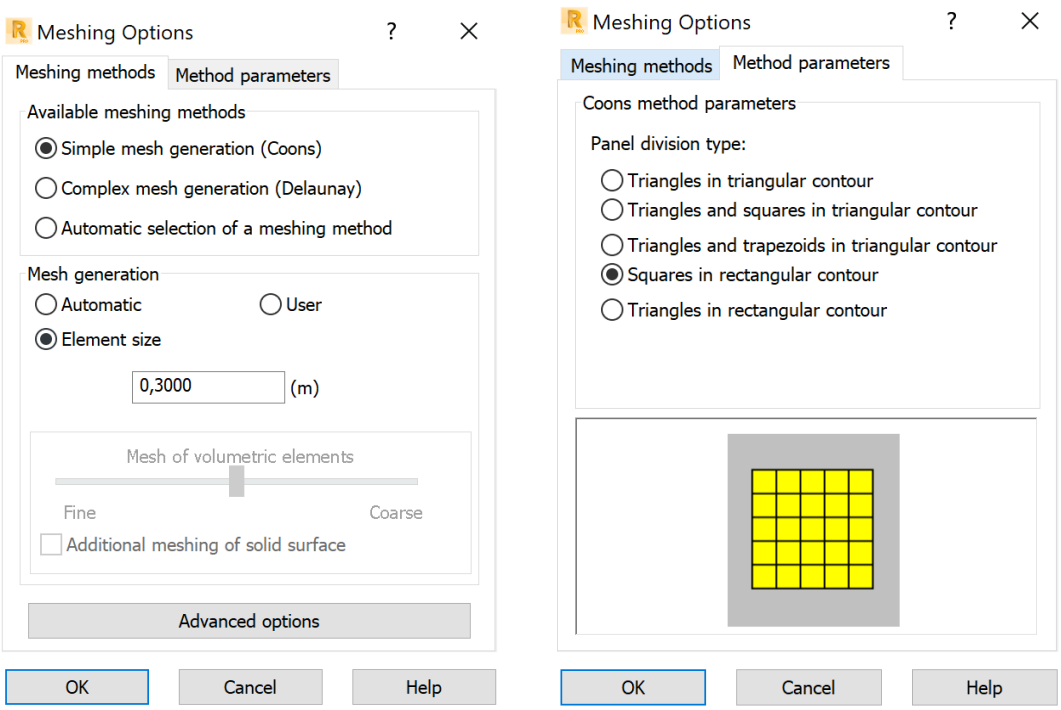

Figure 18. Meshing options and parameters.

Simple mesh generation (Coons) was chosen between all available meshing methods, as it takes the least amount of time to process, though the precision level is enough for this

project. Elemen size was chosen to be 0,3 m, as the most optimal one. However before it was chosen as 0,1, it took about one hour for the program to process the results. In the "Method parameters" window, the mesh shape was chosen as squares in rectangular contour. Quadrilateral mesh elements such as squares tend to perform better and give more precise results. After the mesh generation and nodes that have been put before, the total amount of nodes is 7021.

4.8 Analysis types

In this part, calculations, analysis types of the model, and the procedures for running them are explained. The structure has been checked for the capability to withstand the self-weight, permanent and variable loads before, so in this case behavior of the structure under the influence of the vibration is investigated. There is no need to consider any loads, as self-weight and permanent loads were already considered during the Thickness calculation (section 4.6). There are two types of analysis that are considered in this project: the modal analysis and the footfall analysis. The modal analysis should always be carried out first. It calculates the Eigenmodes and frequencies of the system for every vibration mode. Footfall analysis is carried out to get the response of the floor structure. "Analysis Type" window can be found from Analysis (Text menu) > Analysis Type (Figure 19).

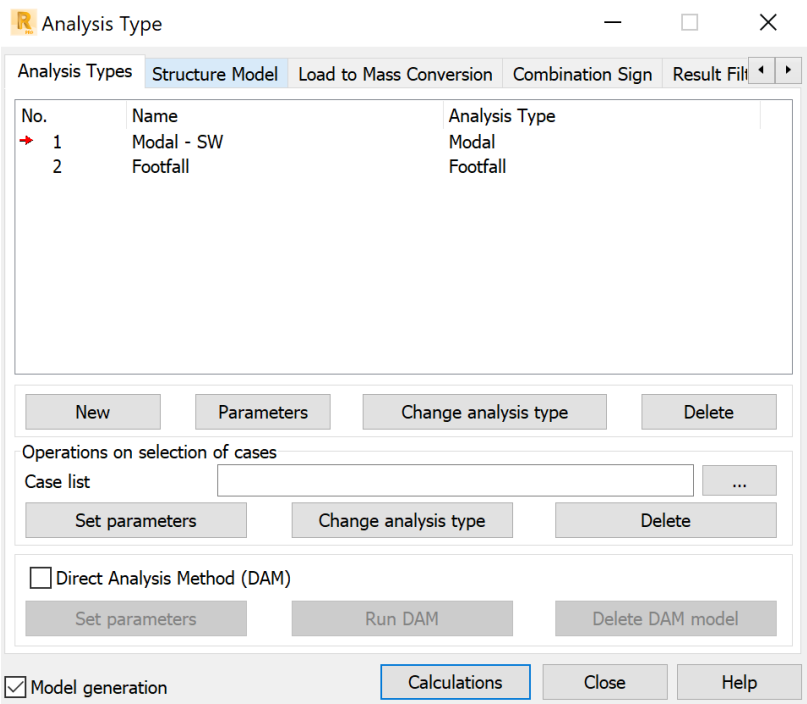

Figure 19. "Analysis Type "window.

4.8.1 Modal analysis

In this section, the parameters needed for performing the modal analysis are defined. Modal analysis type can be created by pressing New in the "Analysis Type" window and choosing Modal in the "New Case Definition" window (Figure 20). Parameters as the number of modes, tolerance, and the number of iterations are left as default. They can be changed in case more modes are needed to be considered or tolerance changed if a more precise result is demanded. There are three options for the mass matrix, and the Consistent one is chosen. Consistent mass matrix regards the rotational degrees of freedom. It is a more time-consuming option, as it needs more memory to run the analysis. In the case of lumped mass matrix, elements of the mass matrix are located on the diagonal positions of the matrix with or without consideration of rotations (Autodesk Inc., 2022). Active mass direction is Z, as it is the direction of interest where the most deformation will happen. Sturm check is carried out for detecting the skipped eigen vibrations. When the skipped mode is detected, an iteration is run again. The damping ratio was estimated before, and it is equal to 0,02. After the properties are chosen as needed, the software can calculate Eigenmodes and frequencies.

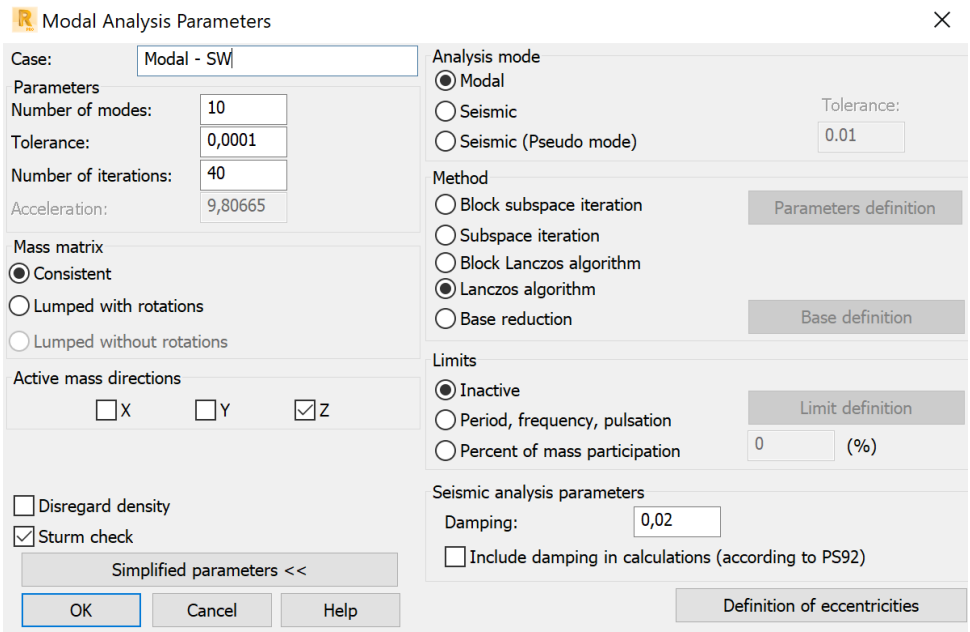

Figure 20. "Modal Analysis Parameters" window.

4.8.2 Footfall analysis

After the properties of the modal analysis are defined, the next step is to adjust the properties of the footfall analysis. It should be added as a new analysis type by pressing New in the "Analysis Type" window and choosing footfall. When conducting the vibration analysis in Autodesk Robot, both footfall and modal analyzes should be taken into account because the Eigenmodes and frequencies acquired from the modal analysis are needed to run the footfall analysis. The excitation method was chosen as Self-excitation. In this way, the response is analyzed in the same node where the force is applied (Autodesk Inc., 2022). In the case of Full excitation, the response is analyzed in any node which is affected by force applied on the other node. Analysis can be done according to four different design guides. SCI Publication P354 Design of Floors for Vibration: A New Approach (A.L.Smith, 2009) is chosen as the research is done according to that publication. Frequency weight is determined from the same publication, and it is factor Wb, explained in section 7.1. Walker's weight – in the case of Hopealaakso kindergarten that was the weight of a person who was walking during the tests, whose weight was approximately 95 kilograms. Footfall analysis has been carried out twice because there were two walking frequency ranges considered: from 1,6 Hz to 2 Hz (Figure 21, a) and from 2,7 Hz to 2,8 Hz (Figure 21, b). The full walking range can be viewed in Table 9. After all, properties are adjusted as needed, and the model is ready, the software can calculate accelerations.

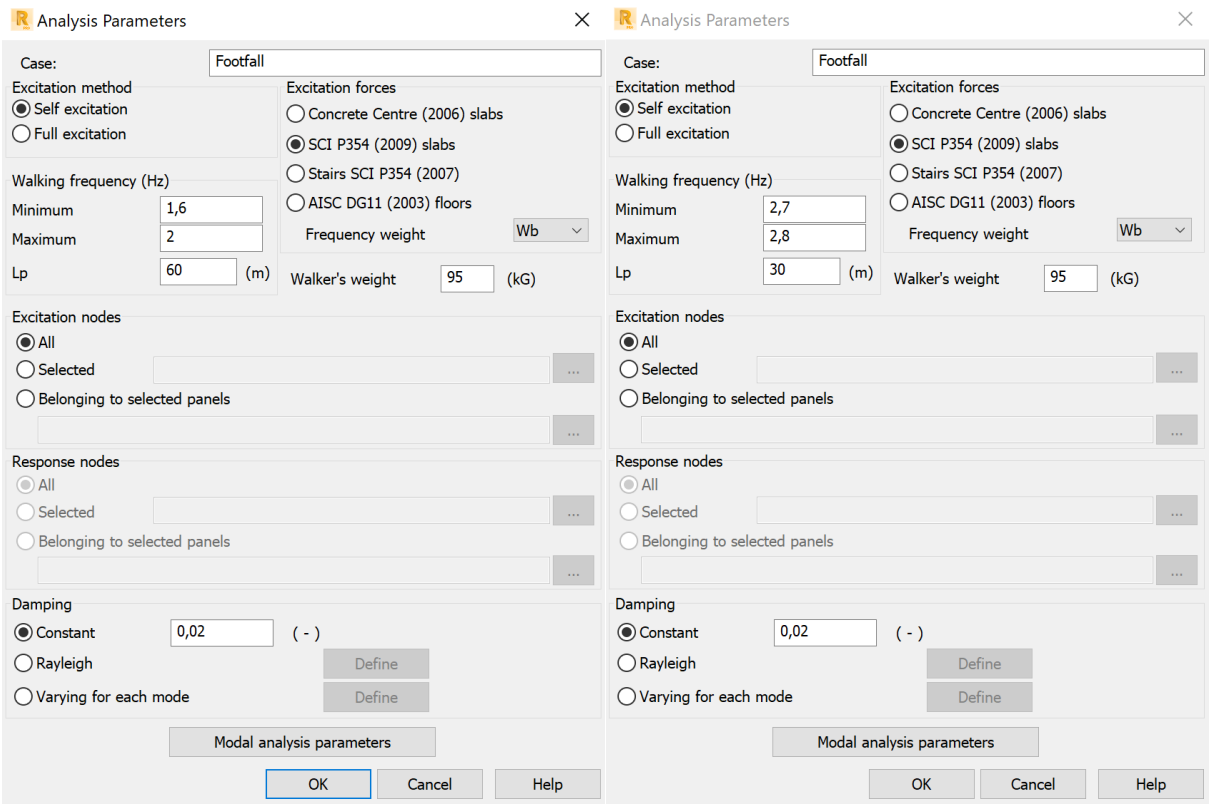

Figure 21. "Analysis parameters" window for the walking frequency interval 1,6-2 Hz (a), 2,7-2,8 Hz (b)

4.9 Calculation and results of FEM model

After the model is ready, it should be checked if the integrity and physical properties of the structural elements look correct, then the model can be calculated. In order to start the analysis, the Calculations button should be pressed in the "Analysis Type" window (Figure 19). All the errors that occur during the calculations should be solved. Instabilities that occur during the calculation should be checked and fixed depending on the impact they have on the results. In sections 4.9.1 and 4.9.2, results from the modal analysis and footfall analysis respectfully are described.

4.9.1 Results from Modal analysis

Calculation of this particular model using a normal office computer took approximately 2 minutes, so depending on the analysis options chosen and computer memory, calculation time can be different. From the modal analysis, there are two main outputs that are of the most interest: lowest natural frequencies and modal masses of the first ten modes. The most convenient way to display these values is in the table format, which can be found from Results (Text menu) > Advanced > Modal Analysis. Then, it is important to choose Modal analysis in the "Select cases" dialog box. The table which is generated automatically doesn't include the modal mass, so modal masses and other data which can be displayed in the table can be added or removed by clicking with the right button of the mouse on the table and going to "Table Columns...". The lowest natural frequencies and modal masses for the first ten modes can be seen in (Figure 22).

Case/Mode	Frequency (Hz)	Period (sec)	Cur.mas.UZ (%)	Rel.mas.UZ (%)	Total mass UZ (kg)	mZ (kg)
1/ 1	9,13	0,11	6,44	6,44	263501,22	9919,47
1/ 2	11,67	0,09	5,86	12,30	263501,22	7899,94
1/ 3	12,53	0,08	8,06	20,36	263501,22	5051,44
1/ 4	13,72	0,07	0,63	20,99	263501,22	9771,28
1/ 5	14,02	0,07	2,12	23,10	263501,22	9570,72
1/ 6	15,82	0,06	3,14	26,25	263501,22	5973,69
1/ 7	16,36	0,06	2,25	28,50	263501,22	10446,80
1/ 8	17,62	0,06	0,21	28,71	263501,22	14081,21
1/ 9	18,16	0,06	0,01	28,72	263501,22	14553,23
1/ 10	18,87	0,05	0,82	29,54	263501,22	12370,50

Figure 22. Natural frequencies and modal masses for modes 1-10

Another way to view the natural frequency of the structure is to display it as a map view. This way, the part of the mass, acting in the first vibration mode, and the amplitudes can be seen. That view can be displayed by pressing the "View" button on the left-bottom corner of the screen. It gives information about the natural frequency of the mode which is chosen and displays the current amplitudes (Figure 23,24). It can be clearly seen that the biggest amplitude in the first mode occurs approximately in the middle of the slab located in the left-bottom part of the floor. In the second mode, the biggest amplitude occurs in the middle of the slab located in the right-top part of the floor; however, it can't be noticed that easily.

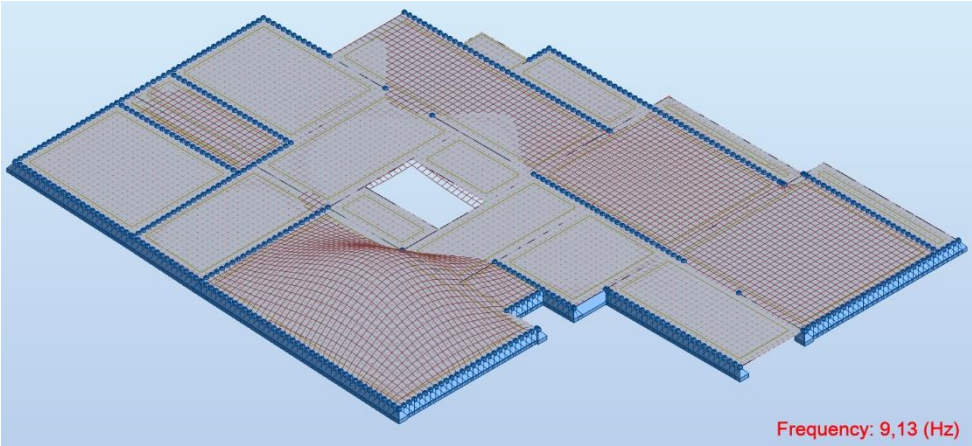

Figure 23. Natural frequency related to the first mode.

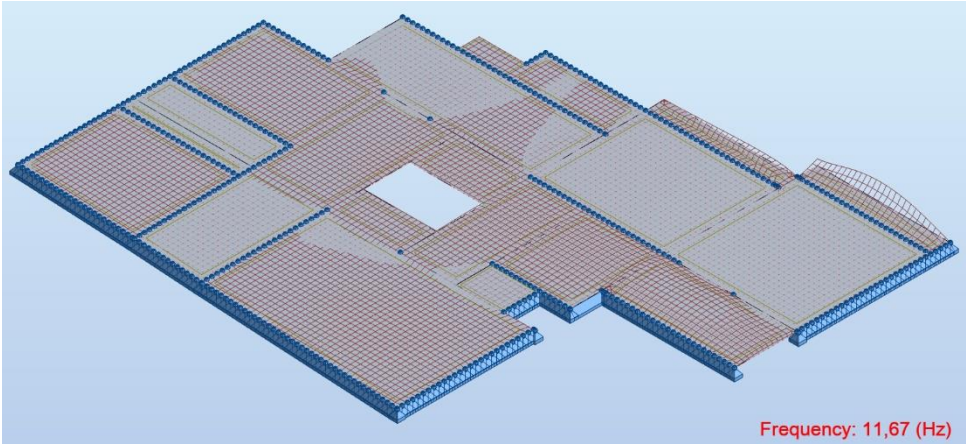

Figure 24. Natural frequency related to the second mode.

4.9.2 Results from Footfall analysis

In this section, the process of acquiring footfall analysis results is explained. In this type of analysis, the root-mean-square acceleration values and response factors can be viewed. In the case of this research, only response factors have been analyzed, as later on, they are compared with hand-calculated response factors to check the reliability of the software calculations. Response values can be displayed from Results (Text menu) > Advanced > Footfall Analysis – Maps… . Then, in order to get the right view, it is important to make sure that Footfall analysis is selected in the "Select Cases" dialog box and Results – diagrams are selected in the "Layouts" dialog box. In the "Footfall" window, it can be chosen what kind of

data is displayed in the model, so Resonant or Transient Maximum response factors are chosen to be displayed. In the case of transient analysis, the response is dominated by the impulsive part of the human walking, which corresponds to the heel impact of the walker. For the resonant response, continuous parts of human walking with the low frequency are crucial ones, as they cause the resonance. Transient response applies to high-frequency floors such as timber floors, while the resonant response is a criterion of low-frequency floors such as floors made of concrete. In the walking frequency interval 1,6-2 Hz, the highest resonant response factor is 2,7, while the highest transient response factor is 9,72 (Figure 25,26). In the walking frequency interval 2,7-2,8 Hz, the highest resonant response factor is 14,97, while the highest transient response factor is 17,53 (Figure 27,28).

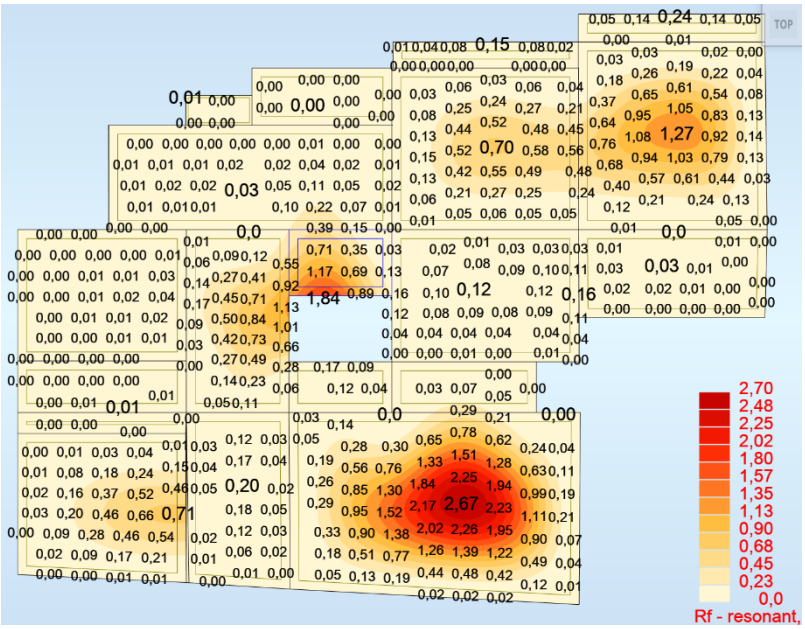

Figure 25. Resonant response factors for the walking frequency interval 1,6-2 Hz.

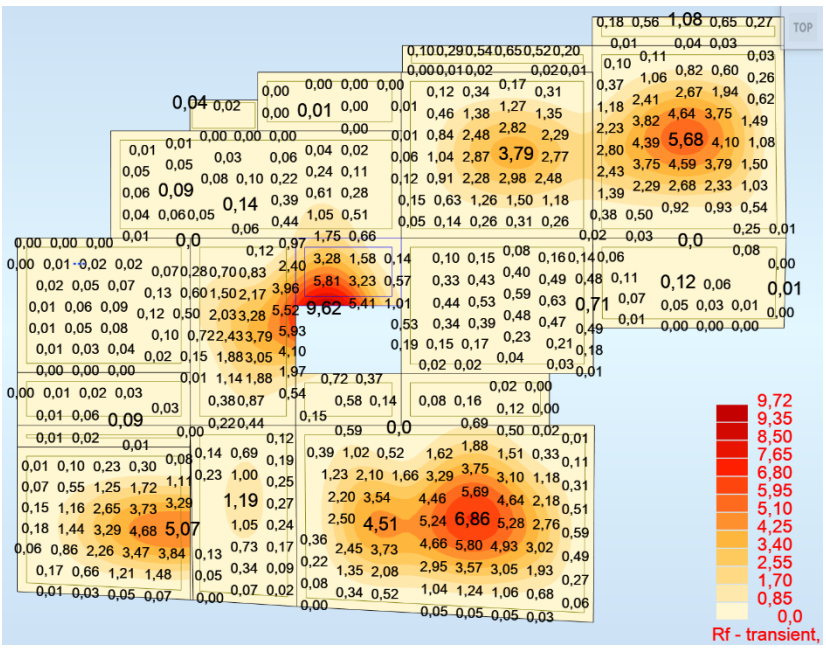

Figure 26. Transient response factors for the walking frequency interval 1,6-2 Hz.

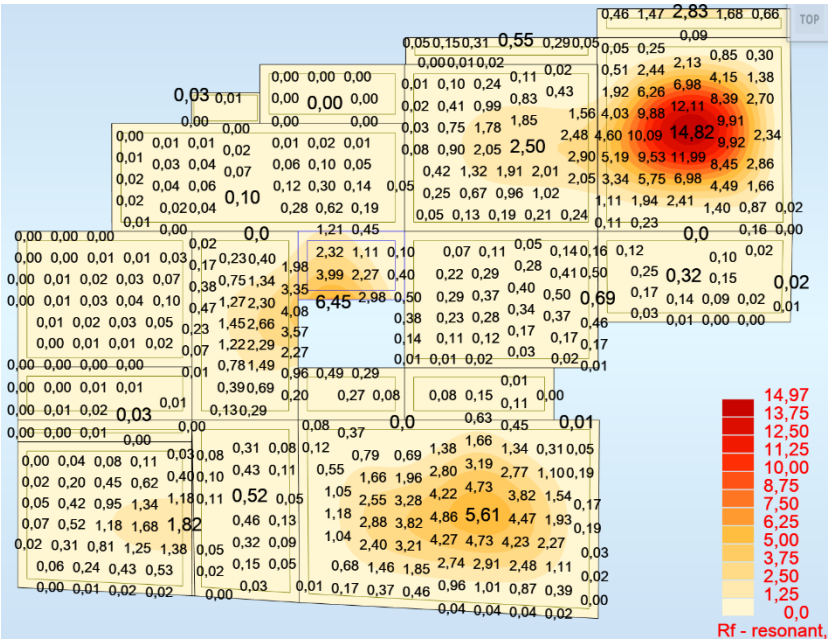

Figure 27. Resonant response factors for the walking frequency interval 2,7-2,8 Hz.

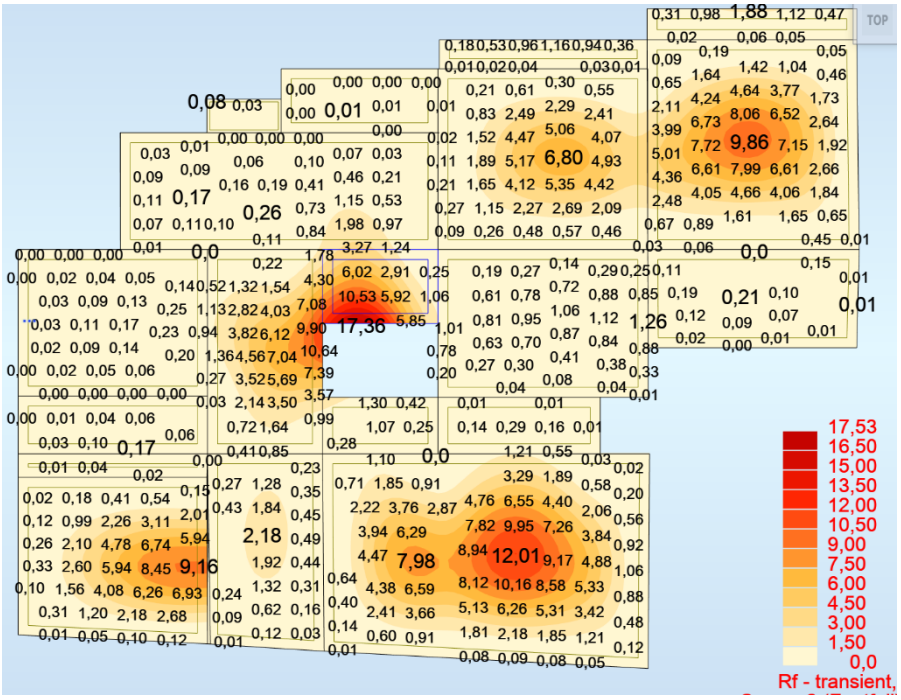

Figure 28. Transient response factors for the walking frequency interval 2,8-2,8 Hz.

5 Measurements acquired from the site

As the building in question was built in 2020, vibration measurement has already been taken in the construction stage. After elements installation and connection, the serviceability check was done, which included measuring specific values for vibration behavior and comparing them to limitations given in SCI P354 and FEM calculations. The first objective of the measurements was to determine the lowest natural frequency of the floor, in other words, the frequency at which the acceleration peak occurs. The second objective was to find the maximum acceleration during the test period. The equipment used in the tests were accelerometers and the computer program "National Instruments Signal Express," which receives the data from accelerometers and represents it graphically. Three accelerometers were used in the testing procedure, connected to the floor structure in the different places where the maximum vibration was predicted to occur (Figure 29).

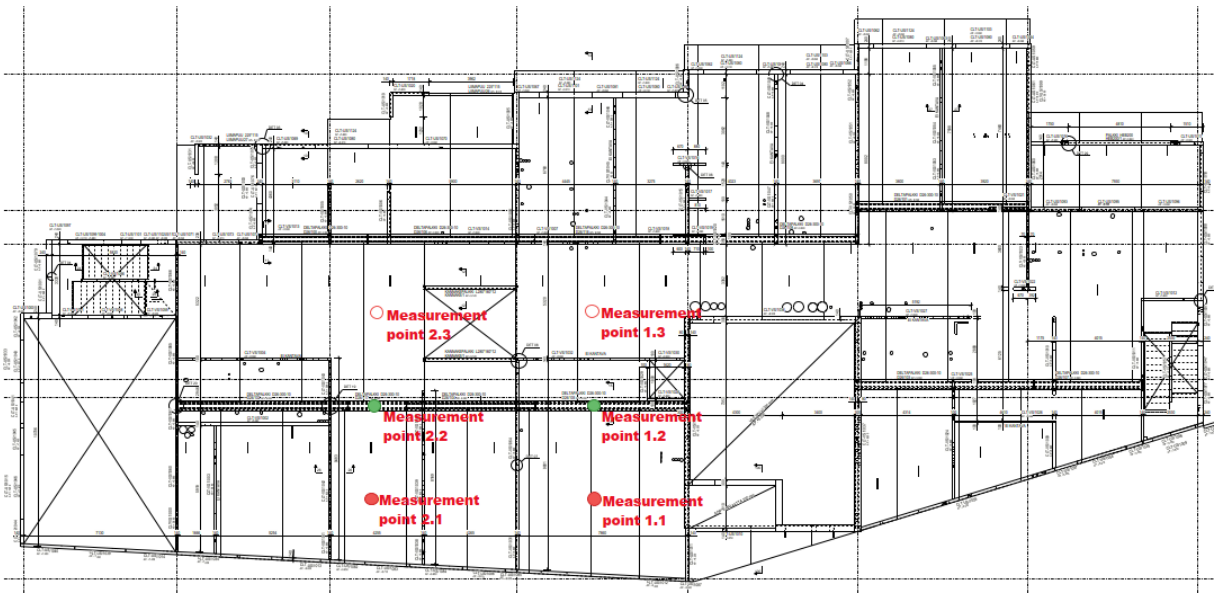

Figure 29. Placement of accelerators and their numbers

5.1 Overview of the results

The data measured with each device will be presented in the graphs with a different color – white, green, or red- to measure points 1, 2, or 3 respectfully. The floor structure in the area of the accelerators is excited by the walking person, whose weight was about 95 kilograms. The walking pace was as smooth and periodic as possible, and the route was from point 1 to point three and then back to point 1, as presented in (Figure 30). There were 15 measurements taken, ten at points 1.1 – 1.3 and 5 at points 2.1 – 2.3. Results from points 1.1 – 1.3 are analyzed in the study as more significant ones. The number of steps, the period of measurement, and the frequency are displayed in Table 2 and Table 3 below.

Table 2 Measurement points 1.1 - 1.3

Measurement	Number of steps	Period	Frequency
1	34	20.6 s	1.650 Hz
2	39	20.8 s	1.875 Hz
3	35	19.2 s	1.823 HZ
4	49	25.9 s	1.892 Hz
5	36	20.5 s	1.756 Hz
6	48	25.2 s	1.905 Hz
7	79	28.3 s	2.792 Hz
8	30	15.7 s	1.911 Hz
9	32	18.2 s	1.758 Hz
10	33	19.9 s	1.658 Hz

Table 3 Measurement points 2.1 - 2.3

Measurement	Number of steps	Period	Frequency
11	32	17.8 s	1.798 Hz
12	34	19.8 s	1.717 Hz
13	32	18.4 s	1.739 HZ
14	36	21.3 s	1.690 Hz
15	32	17.2 s	1.860 Hz

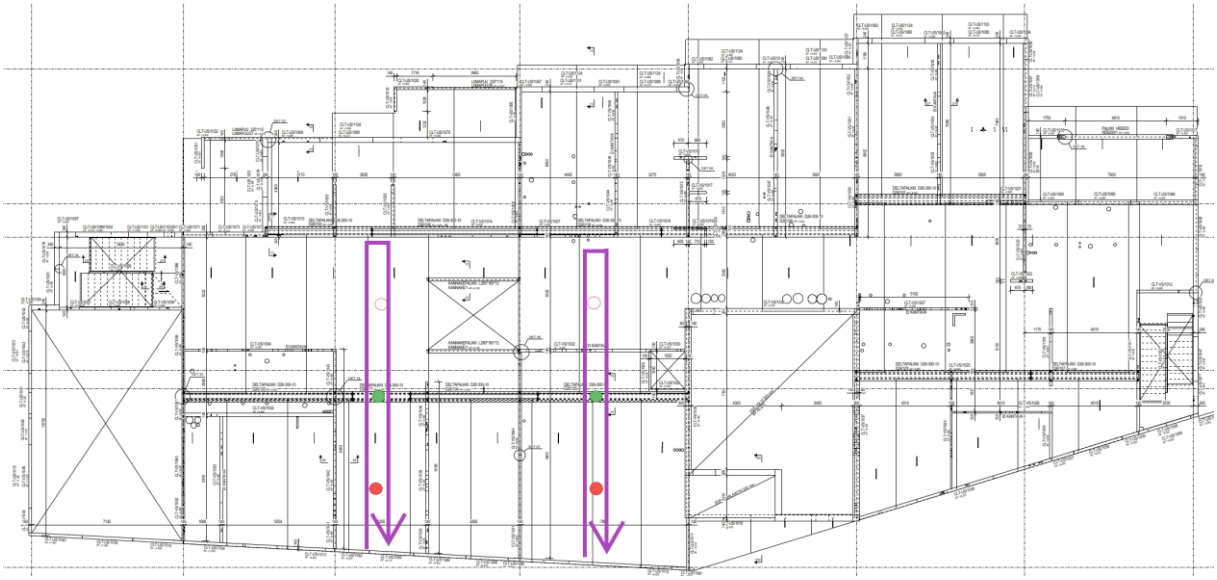

Figure 30. Walking route

As shown in Table 3, the frequency is quite different for every measurement, as the walking pace can not be controlled. The pace can be faster or slower, while the force applied by the step might also be different and not consistent. In total, frequency ranges from around 1,6 Hz to 2,8 Hz, but for further calculation and analysis, it would be more convenient to have two ranges of frequencies: from 1,6 to 2 Hz and from 2,7 to 2,8 Hz.

5.2 Graphical data from the accelerometers

The values of natural frequencies and accelerations have been acquired from the "National Instruments Signal Express" program diagrams. Diagramms are based on the experiments which have been done by Peikko Finland Oy. For each measurement, there will be presented three main diagrams. The first diagram is presented just to show the system's overall behavior, where the y-axis is the acceleration in m/s², and the x-axis is the time in seconds. It

shows how acceleration changes for the whole period of measurement (Annex. Figure 31-40. a). In the second diagram, the value of maximum acceleration is presented, and the frequency at which It happens is the lowest natural frequency of the floor. Y-axis represents the acceleration in m/s^2, while the x-axis represents the frequency in Hz, the period is one second (Annex. Figure 31-40. b). The third diagram shows the relation between acceleration and frequency for the whole measurement period. Y-axis and x-axis are the same as in diagram two, though the period is not one second but the entire measurement time (Annex. Figure 31-40. c).

6 Hand calculations by publication SCI P354

Hand calculations of the response are done for understanding and comparing the calculation process in the numerical analysis software. Hand calculations of weighted accelerations and response factors are based on the publication SCI P354 (A.L.Smith. 2007), which is a guide for the determination of the vibration response of floors with improved accuracy. It is good to mention that the vibration analysis in FEM software was also carried out according to SCI P354. Before performing the hand calculations, modal analysis in FEM should still be done to acquire the values of natural frequencies and modal masses. That can be done manually, though it would take an enormous amount of time for a complicated structure while the results wouldn't be such precise. In sections 6.1 and 6.2, resonant or steady-state response analysis and transient response analysis respectfully are explained.

6.1 Accelerations and steady-state responses of the floor

A steady-state response, also called a resonant response, is applied to low-frequency floors, like concrete floors, for example. It is considered significant when one of the harmonic components of walking excitation is close to the natural frequency of the floor. Steady-state response analysis of the floor is done according to publication SCI P354 section 6.3.2. The weighted root-mean-square acceleration of the steady-state response is calculated as

follows (equation 20). Accelerations are calculated for F=1,6 Hz, F=2 Hz, and F=2,792 Hz in the first four harmonics of the vibration.

$$\alpha_{w.rms.e.r.n.h} := \mu_{en} \cdot \mu_{rn} \cdot \frac{F_h}{M_n} \cdot D_{nh} \cdot W_h$$

Equation 20. Weighted root-mean-square acceleration of the floor.

$\mu_{en} \cdot \mu_{rm}$ – mode shape amplitudes taken as 1, assuming that the acceleration and response are measured at the maximum amplitude.

M_n – modal mass of the nth mode.

F_h – excitation force of the harmonic number h (equation 21), which is calculated by multiplying the static force Q exerted by the person by the Fourier coefficient a_h (Table 4).

$$F_h := \alpha_h \cdot Q$$

Equation 21. Excitation force formula.

Table 4. Fourier coefficients of walking activities according to SCI P354 Table 3.1 where f_p is the pacing frequency.

Harmonic h	Excitation frequency range hf_p (Hz)	Design value of coefficient α_h	Phase angle ϕ_h
1	1.8 to 2.2	$0.436(hf_p - 0.95)$	0
2	3.6 to 4.4	$0.006(hf_p + 12.3)$	$-\pi/2$
3	5.4 to 6.6	$0.007(hf_p + 5.2)$	π
4	7.2 to 8.8	$0.007(hf_p + 2.0)$	$\pi/2$

D_{nh} – dynamic magnification factor of the acceleration, found as follows (equation 22). β is the pacing frequency to natural frequency ratio, and ζ is the damping ratio that is equal to 0,02.

$$D_{nh} := \frac{h^2 \cdot \beta^2}{\sqrt{\left(1 - h^2 \cdot \beta^2\right)^2 + (2 \cdot h \cdot \zeta \cdot \beta)^2}}$$

Equation 22. Dynamic magnification factor.

W_h – weighing factor of human perceptions of vibrations.

It depends on the direction of the vibration, room type, and the impact that it gives: discomfort or the vibration can be perceived. In the case of Hopealaakso kindergarten, vibration acts in Z direction and can cause discomfort, so the weighting curve is W_b (table 5).

Table 5. Weighting factors.

Room Type	Axis of vibration	Category	BS 6841 weighting curve
Critical working areas (e.g. hospital operating theatres, precision laboratories)	z-axis	Vision/Hand control	W_g
	x-, y-axis	Perception	W_d
Residential, offices, wards, general laboratories, consulting rooms	z-axis	Discomfort	W_b
	x-, y-axis	Discomfort	W_d
Workshop and circulation spaces	z-axis	Discomfort	W_b
	x-, y-axis	Discomfort	W_d

As far as the natural frequency of the floor is 9,13 Hz (Figure 23), it fits the frequency interval of 5 – 16 Hz for Z-axis floor vibrations, so the W_h is equal to 1 (table 6).

Table 6. W_b factors.

z-axis vibrations W_b weighting

$W = 0.4$	for $1\ \text{Hz} < f < 2\ \text{Hz}$
$W = \frac{f}{5}$	for $2\ \text{Hz} \leq f < 5\ \text{Hz}$
$W = 1.0$	for $5\ \text{Hz} \leq f \leq 16\ \text{Hz}$
$W = \frac{16}{f}$	for $f > 16\ \text{Hz}$

After weighted root-mean-square accelerations are defined for each harmonic, they are summed together and divided by the base value (equation 23). Response factors are

compared to the limit values to define if the vibration that the person perceives is acceptable or not.

$$R_1 := \frac{a_{w.tot}}{5}$$

Equation 23. Response factor.

Where:

$a_{w.tot}$ – is the sum of root-mean-squre acceleration for the particular pace frequency.

5 – constant reference value, which is equal to 5 mm/s².

In the table below, the calculated value of root-mean-square acceleration for the first four harmonics, total accelerations, and response factors are presented (table 7). It can be concluded that the values of resonant response calculated by hand are similar to those calculated by FEM. Response factor in the interval 2,7 – 2,8 Hz is slightly bigger in Autodesk Robot, though if the model has been a bit more precise, then most likely the results would be closer.

Table 7. Root-mean-square accelerations, total accelerations, and response factors.

Fp (Hz)	a_1 (mm/s²)	a_2 (mm/s²)	a_3 (mm/s²)	a_4 (mm/s²)	$a_{.w.tot}$ (mm/s²)	R
1,6	0,86	1,247	2,56	5,432	6,193	1,086
2	2,21	2,224	5,7	21,916	22,861	4,011
2,792	7,934	6,136	47,123	26,486	54,979	9,65

6.2 Acceleration and transient responses of the floor

Transient response analysis is applied to the high-frequency floors, such as floors made of timber. In this case, the response is dominated by the group of impulses that happen when the heel of the walker hits the ground. Transient response analysis is done according to publication SCI P354 section 6.3.3. . The weighted peak acceleration response is obtained as follows (equation 24).

$$a_{w.peak.e.r.n} := 2 \cdot \pi \cdot f_n \cdot \sqrt{1 - \zeta^2} \cdot \mu_{en} \cdot \mu_m \cdot \frac{F_I}{M_n} \cdot W_h$$

Equation 24. The weighted peak acceleration response of transient vibration.

f_n – natural frequency of the floor.

$\mu_{en} \cdot \mu_{rm}$ – mode shape amplitudes are taken as 1, assuming that the acceleration and response are measured at the maximum amplitude.

F_I – the excitation force equal to the force from one footfall according to SCI P 354 section 3.1.1, which is calculated as follows (equation 25).

$$F_I := 60 \cdot \frac{f_p^{1.43}}{f_n^{1.3}} \cdot \frac{Q}{700} \cdot s$$

Equation 25. The excitation force formula.

When the weighted peak acceleration values are obtained, they are summed together and divided by the base value to get the response factors (equation 23). Accelerations and response factors for the transient response analysis are presented in Table 8.

Table 8. Accelerations and response factors for the transient response analysis.

Fp (Hz)	$a_{.w.peak}$ (mm/s²)	R
1,6	52,013	9,13
2	71,563	12,55
2,792	115,312	20,23

Values of transient response factors calculated manually are not far from the same factors calculated by FEM. The results are satisfactory; however, if the model had been more precise, there would be a greater correlation between the results.

7 Overview of the results

In this section of the thesis, results are summed up. Results are presented in the table format (Table 9), where the values which have been measured on-site are compared with the results calculated by hand according to publication SCI P354.

Table 9. Results from the calculated and measured values.

	Measured values			Calculated values		
Measurement N	Walking frequency (Hz)	Lowest natural frequency of the floor (Hz)	$a_{,max}$ (mm/s²)	Natural frequency (Hz)	$a_{,w,peak}$ (mm/s²)	$a_{,w}$ (mm/s²)
1	1,65	10,68	56	9,13	54	7,1
2	1,875	10,75	50	9,13	65	14,1
3	1,823	10,9	53	9,13	63	11,9
4	1,892	10,96	179	9,13	66	15,0
5	1,756	10,52	35	9,13	59	9,6
6	1,905	10,85	58	9,13	67	15,7
7	2,792	10,85	62	9,13	115	55,0
8	1,911	10,1	75	9,13	67	16,0
9	1,758	10,88	61	9,13	60	9,7
10	1,658	10,6	55	9,13	55	7,3

Measured values are ones obtained from the diagrams in section 6.2. These values are walking frequency, lowest natural frequency of the floor, and maximum acceleration "$a_{,max}$" which have been measured by the accelerometers. The total number of measurements is 10. The calculated values are: the lowest natural frequency of the floor, the weighted peak acceleration "$a_{,w,peak}$" and the total acceleration"$a_{.w}$". The lowest natural frequency of the floor is the one calculated by FEM. The weighted peak accelerations were calculated according to the transient response analysis described in section 7.2 for the walking frequencies measured by the accelerometers, while total accelerations were calculated according to resonant response analysis described in section 7.1.

7.1 Conclusions

After the results have been summed up in Table 9, measured values to calculated values ratios are defined in Table 10. The first column is the number of the measurement. The

second column is the lowest measured natural frequency to the lowest calculated natural frequency ratio. The third column is the calculated weighted peak acceleration to the measured maximum acceleration ratio.

Table 10. Measured and calculated values comparison.

Measurement N	$\frac{Lowest\ measured\ natural\ frequency}{Lowest\ calculated\ natural\ frequency}$	$\frac{\max(a,w,peak;a,w) - calculated}{a,\max - measured}$
1	10,68/9,13 = 1,17	54/56 = 0,96
2	1,18	1,30
3	1,19	1,19
4	1,20	0,37
5	1,15	1,69
6	1,19	1,16
7	1,19	1,85
8	1,11	0,89
9	1,19	0,98
10	1,16	1,00

There are several conclusions that can be made from all the data and results collected:

1. As it can be seen from Table 10, in the measurements 4, 8, and 9, calculated values of acceleration are smaller than the measured ones. The most logical reason for high floor acceleration during measurement 4 is just heavier footsteps of the person who was walking.

2. All fundamental frequencies of the floor measured on the site and calculated by FEM are higher than 9 Hz, which is typical for high-frequency floors. That makes sense because, in the case of Hopealaakso kindergarten floor structure is made of timber.

3. Resonant response factors calculated both by hand and FEM are smaller than the transient response factors. That proves the fact that resonant response analysis is less relative in the vibration analysis of high-frequency floors.

7.2 Acceptability of the vibration

Acceptability of the vibration is checked according to section 5.3.2 of SCI P354 if other acceptability criteria are not given. The response factors calculated previously are compared to the multiplying factors, which are shown in table 11.

Table 11. Recommended multiplying factors.

Place	Multiplying factor for exposure to continuous vibration
Office	8
Shopping mall	4[35]
Dealing floor	4
Stairs – Light use (e.g. offices)	32[58]
Stairs – Heavy use (e.g. public buildings, stadia)	24[58]

Since the real floor accelerations have already been measured on-site, they are taken into account for acceptability checks. The highest acceleration value was 179 mm/s^2, though it is not taken into consideration. Acceleration of 75 mm/s^2 corresponds to the response factor equal to 15. That and most of the response factors are higher than the limit value, so the floor structure is considered as not acceptable according to this criteria.

8 Summary

The main aim of the thesis was achieved: results of vibration analysis from FEM, hand calculations, and measurements have been compared and considered as relatively similar. Natural frequencies measured on-site have good correspondence with the frequencies calculated by FEM, while the accelerations which were calculated according to SCI P354 are about the same as accelerations of the floor on-site. There is a deviation between

accelerations on measurement number 4 due to the fact that the walker's footsteps were heavier, so it is fine. It can be concluded from the research that Robot Autodesk is a good and reliable tool for checking vibration analysis of CLT slabs supported by steel-concrete composite beams DELTABEAM®.

Another objective of the thesis was to make a guide for the vibration analysis of CLT slabs in Autodesk Robot. That objective was also achieved, as this research-based thesis can be used as a guide for vibration analysis in FEM. However, for properly utilizing and understanding the guide, it is recommended that the reader has some knowledge about vibration in slabs and has some experience in the program.

References

Peikko Group. (2021). DELTABEAM® Composite Beams Slim Floor Structure with integrated fireproofing [Technical manual]

Peikko Group. (2020). Performance of DELTABEAM® - CLT floors in human-induced vibration.

Peikko Group. (n.d.-a). DELTABEAM® Slim Floor Structure with timber constructions [Brochure].

EN 1994-1-1. (2005). Eurocode 4: Design of composite steel and concrete structures. Part 1-1: General rules and rules for buildings.

Stora Enso Oyj. (2013). Simplified design method using the gross section.

Autodesk Inc. (2022). Robot Structural Analysis 2022 Help. Retrieved February 10, 2022, from https://help.autodesk.com/view/RSAPRO/2022/ENU/

Pia Johansson. (2009). Vibration of Hollow Core Concrete Elements Induced by Walking.

Annex

Annex includes diagrams from the measurements taken.

Measurement 1 – walking frequency 1,65 Hz.

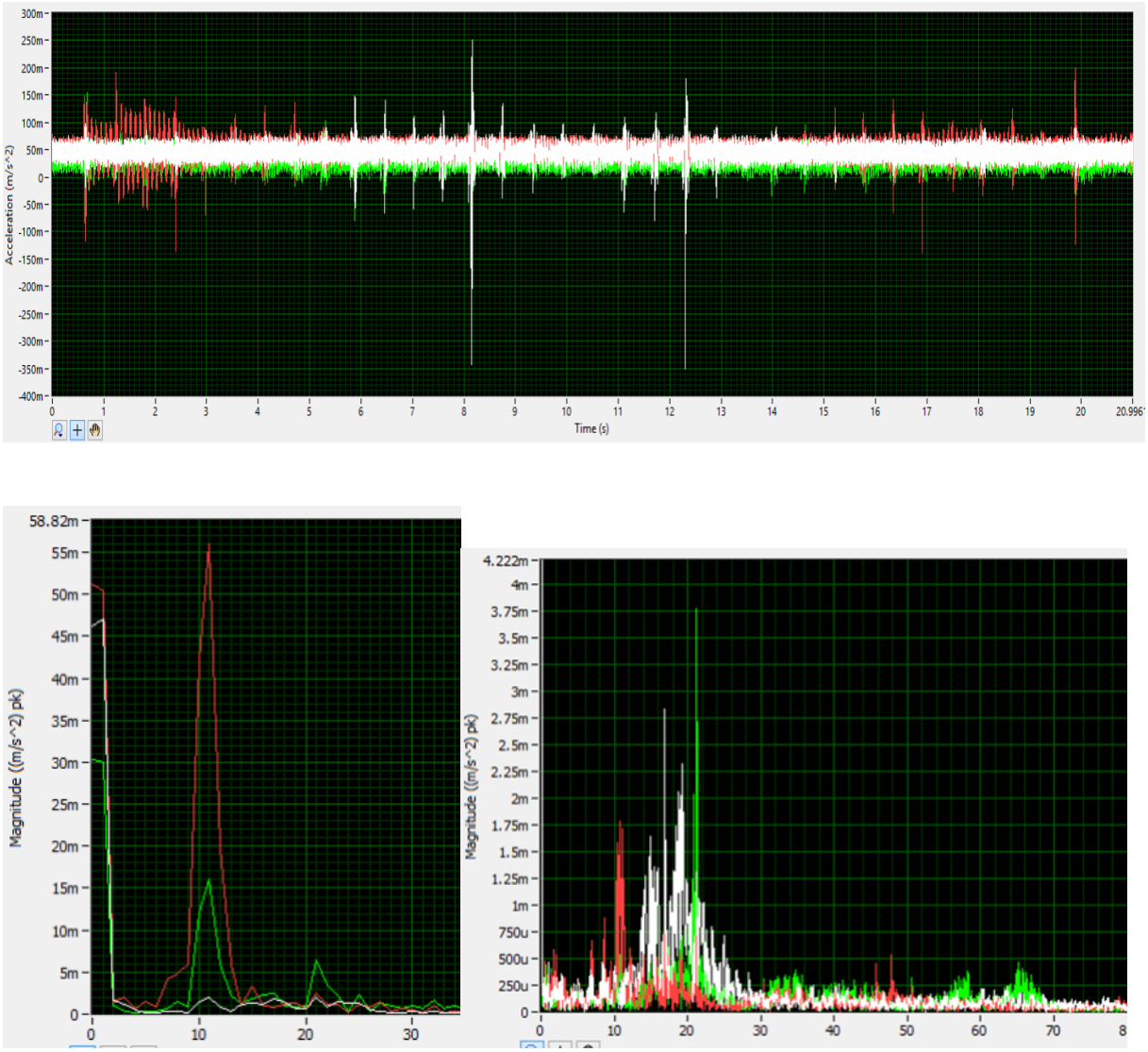

Figure 31. Measured acceleration for the whole period (a), maximum acceleration in the period of 1 second (Hz on the x-axis) (b), and maximum acceleration for the whole measurement period (c).

Measurement 2 – walking frequency 1,875 Hz.

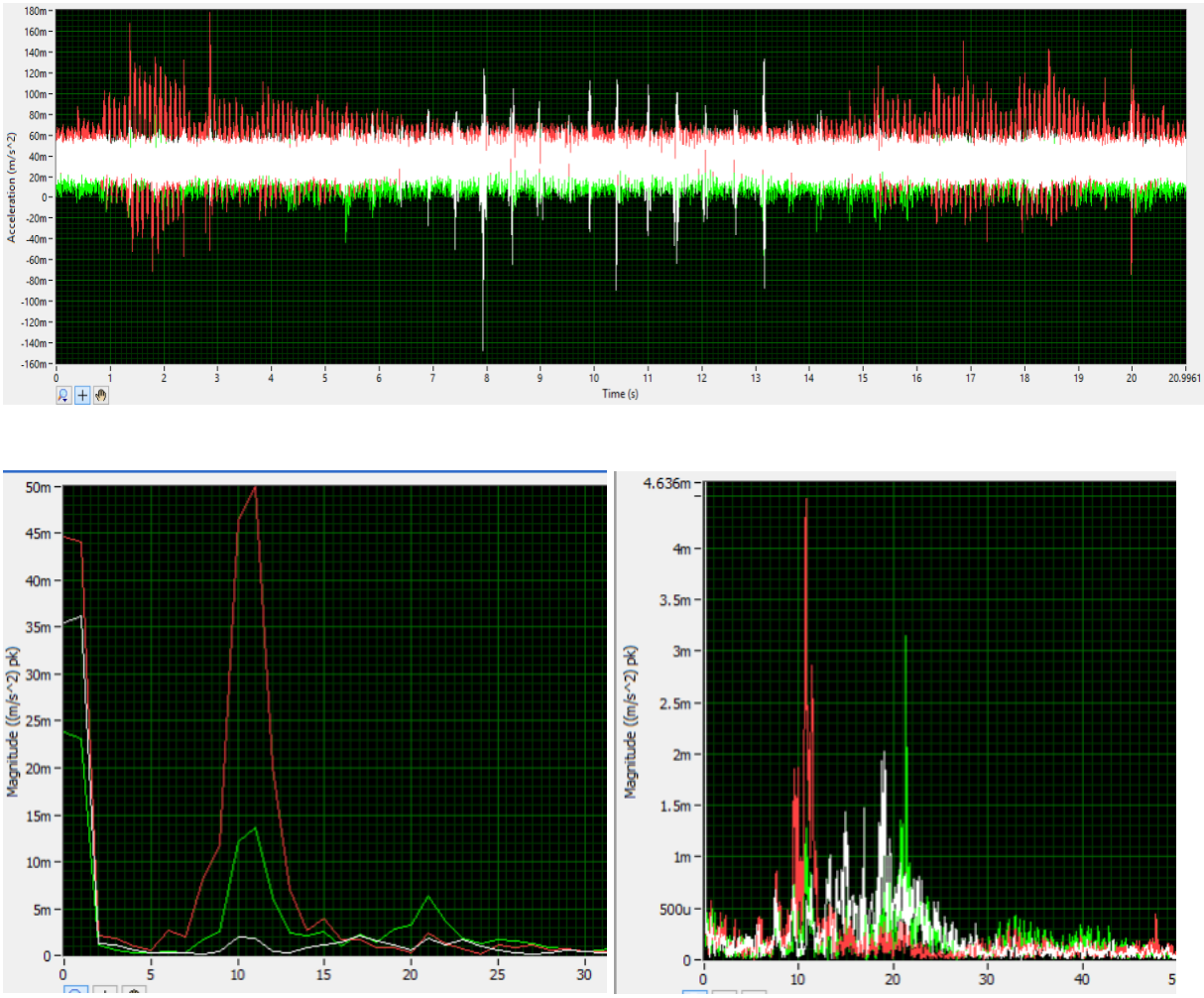

Figure 32. Measured acceleration for the whole period (a), maximum acceleration in the period of 1 second (Hz on the x-axis) (b), and maximum acceleration for the whole measurement period (c).

Measurement 3 – walking frequency 1,823 Hz.

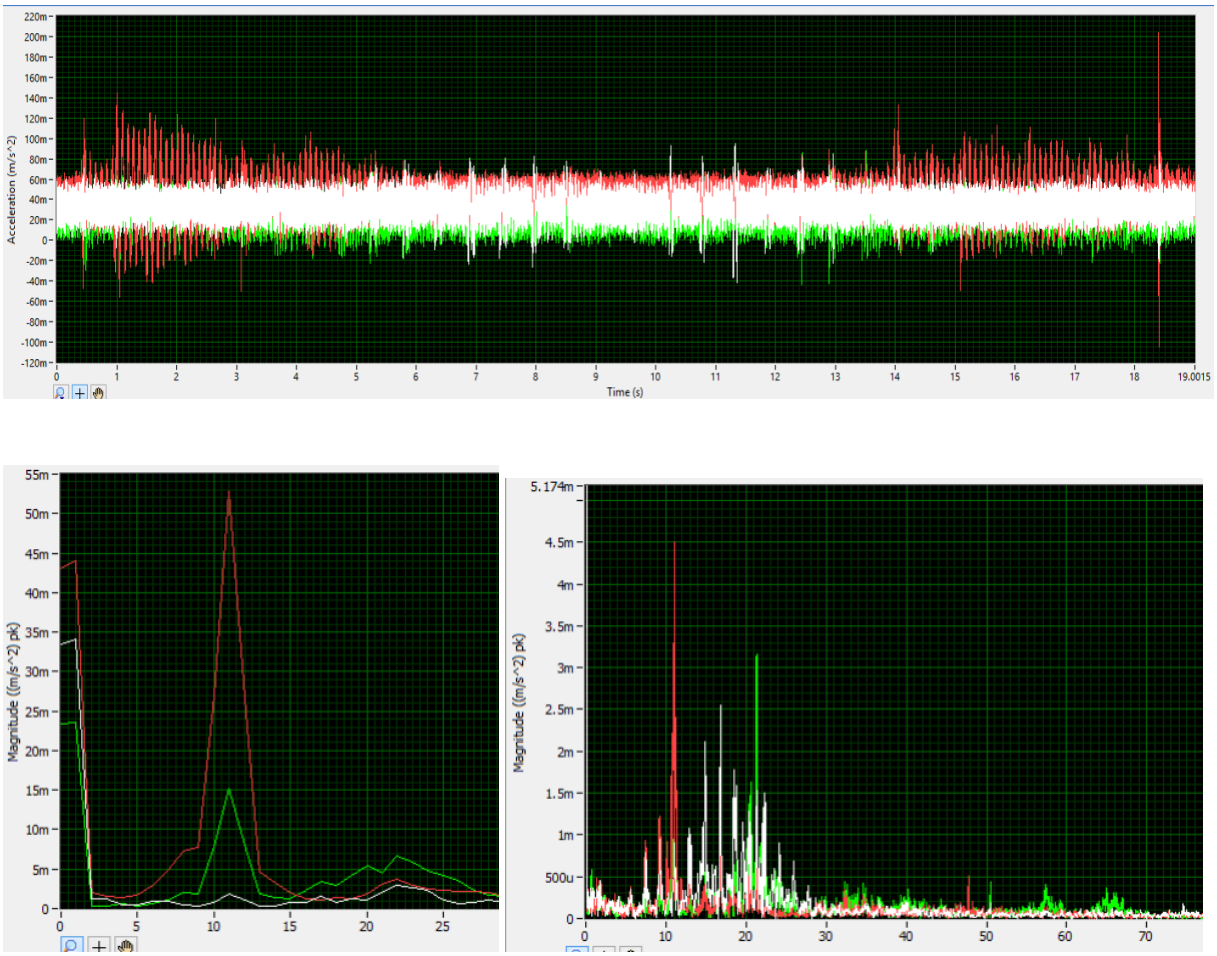

Figure 33. Measured acceleration for the whole period (a), maximum acceleration in the period of 1 second (Hz on the x-axis) (b), and maximum acceleration for the whole measurement period (c).

Measurement 4 – walking frequency 1,892 Hz.

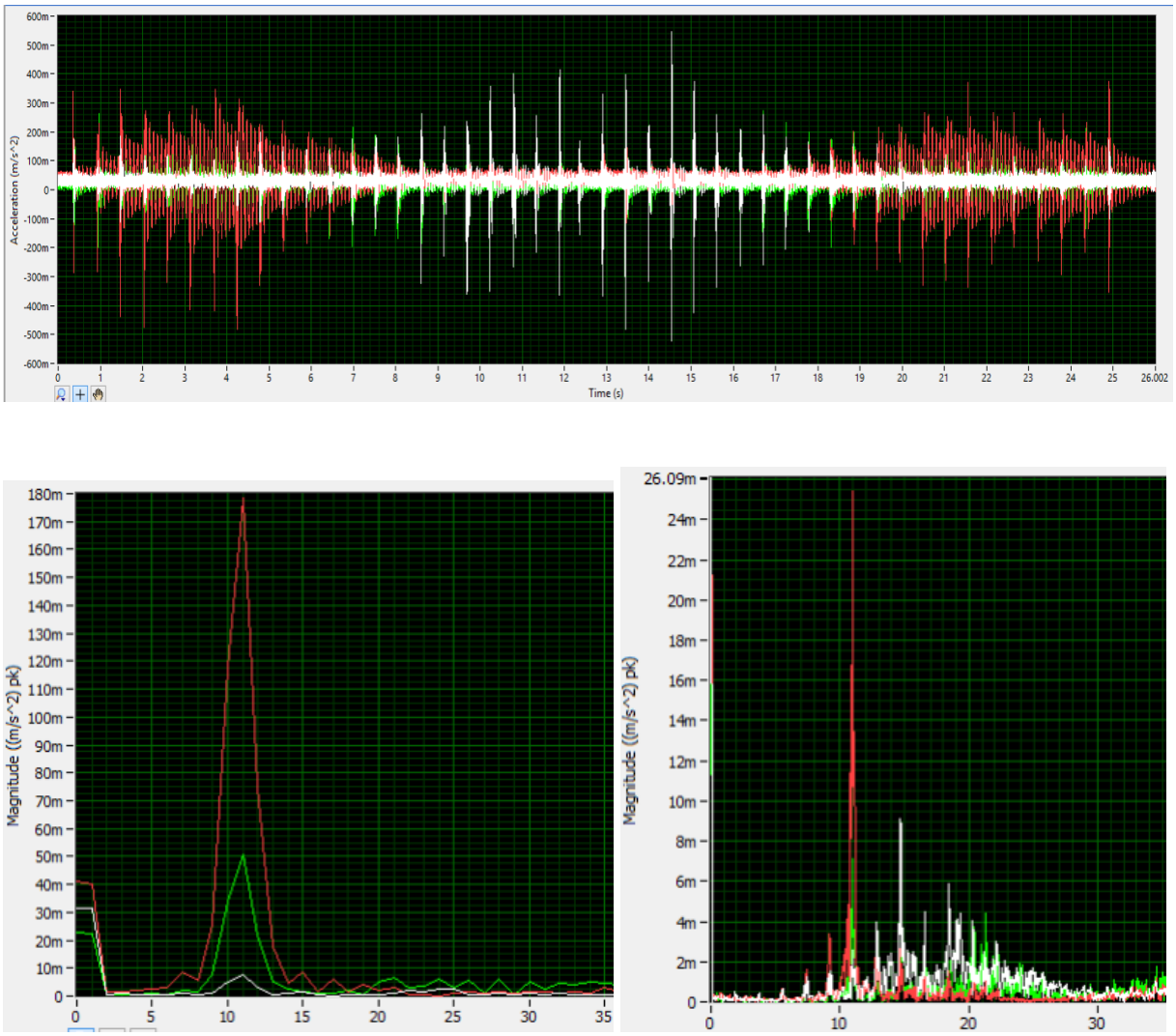

Figure 34. Measured acceleration for the whole period (a), maximum acceleration in the period of 1 second (Hz on the x-axis) (b), and maximum acceleration for the whole measurement period (c).

Measurement 5 – walking frequency 1,756 Hz.

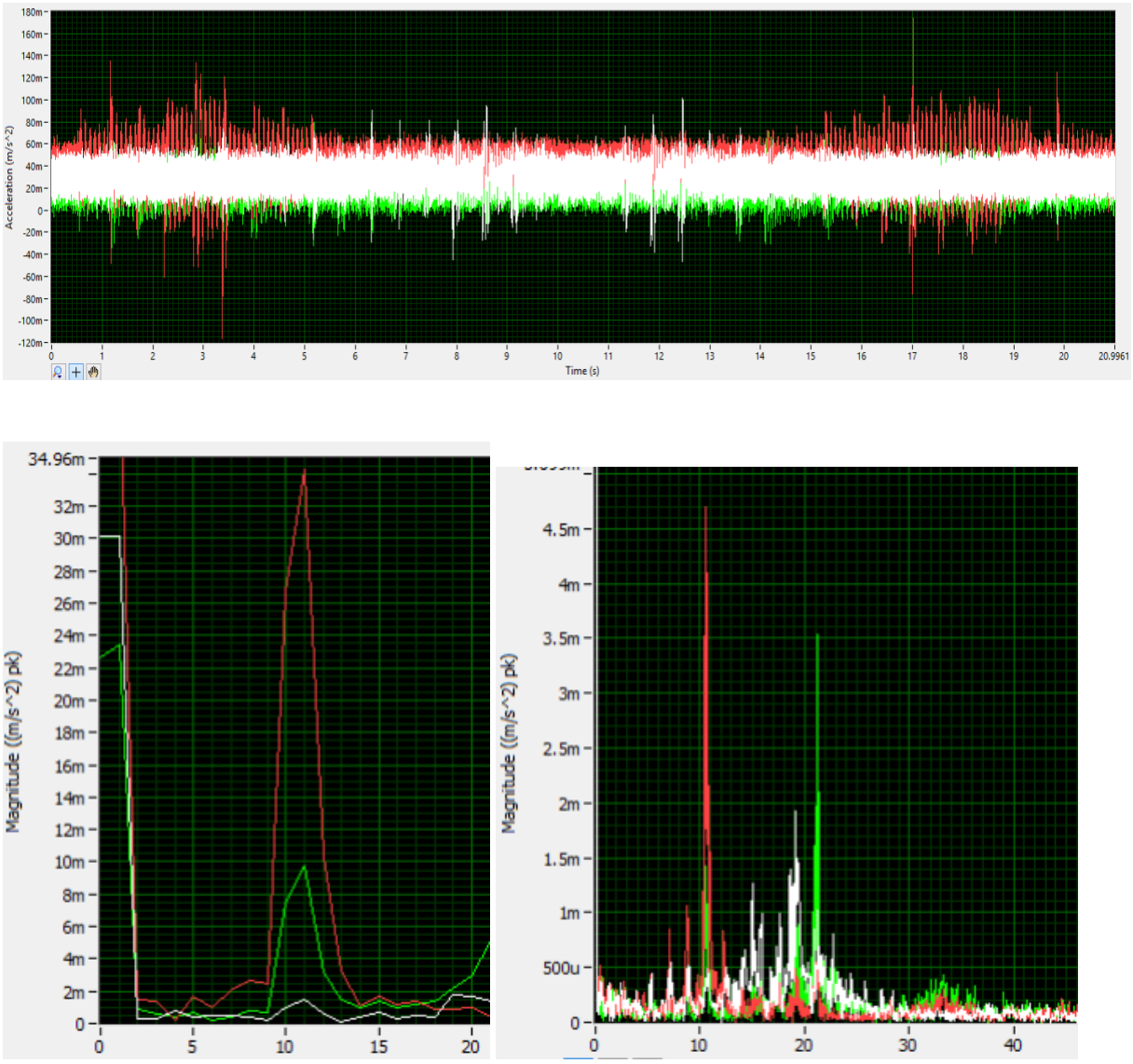

Figure 35. Measured acceleration for the whole period (a), maximum acceleration in the period of 1 second (Hz on the x-axis) (b), and maximum acceleration for the whole measurement period (c).

Measurement 6 – 1,905 Hz.

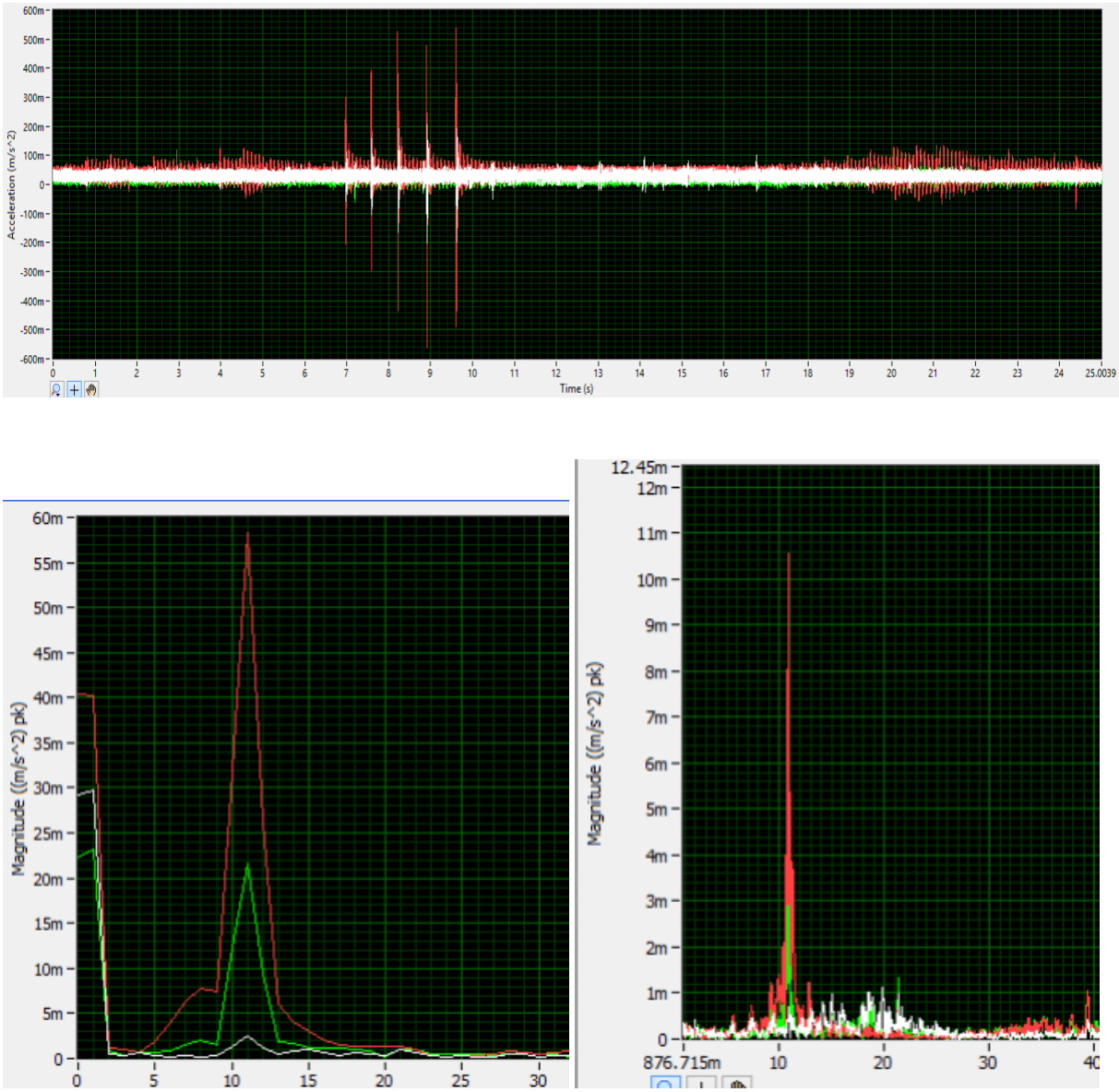

Figure 36. Measured acceleration for the whole period (a), maximum acceleration in the period of 1 second (Hz on the x-axis) (b), and maximum acceleration for the whole measurement period (c).

Measurement 7 – walking frequncy 2,792 Hz.

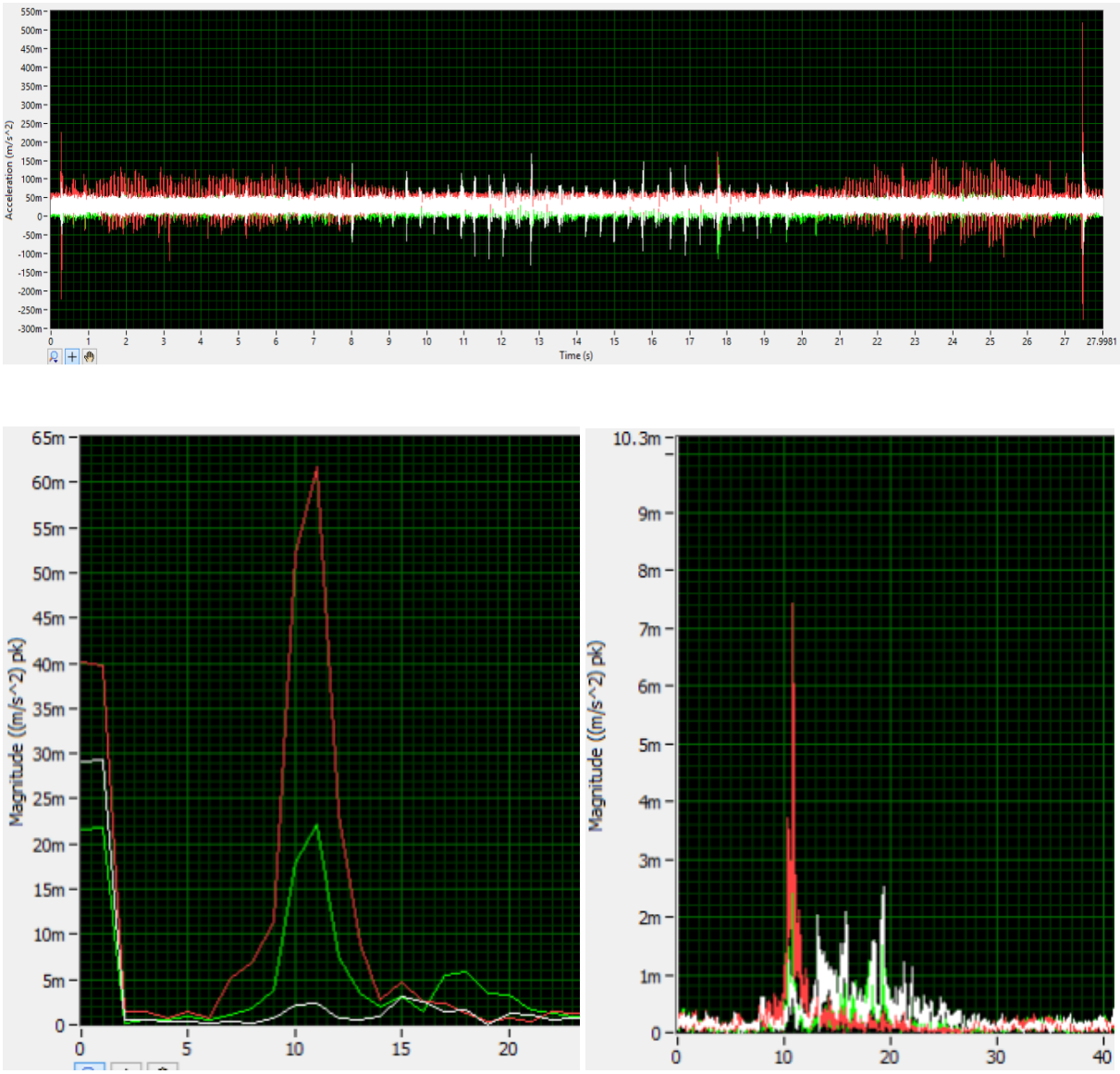

Figure 37. Measured acceleration for the whole period (a), maximum acceleration in the period of 1 second (Hz on the x-axis) (b), and maximum acceleration for the whole measurement period (c).

Measurement 8 – walking frequency 1,911 Hz.

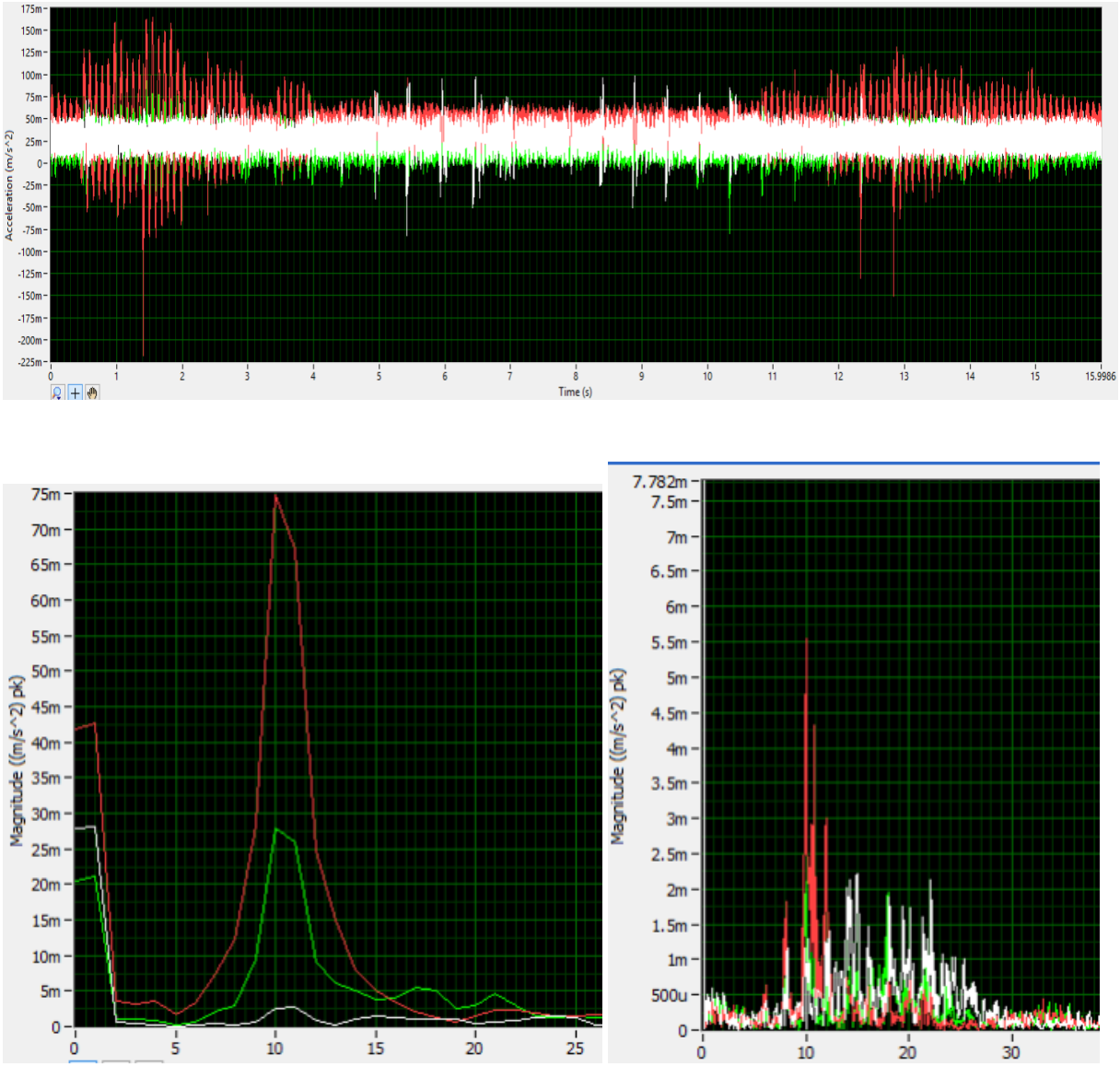

Figure 38. Measured acceleration for the whole period (a), maximum acceleration in the period of 1 second (Hz on the x-axis) (b), and maximum acceleration for the whole measurement period (c).

Measurement 9 – walking frequency 1,758 Hz.

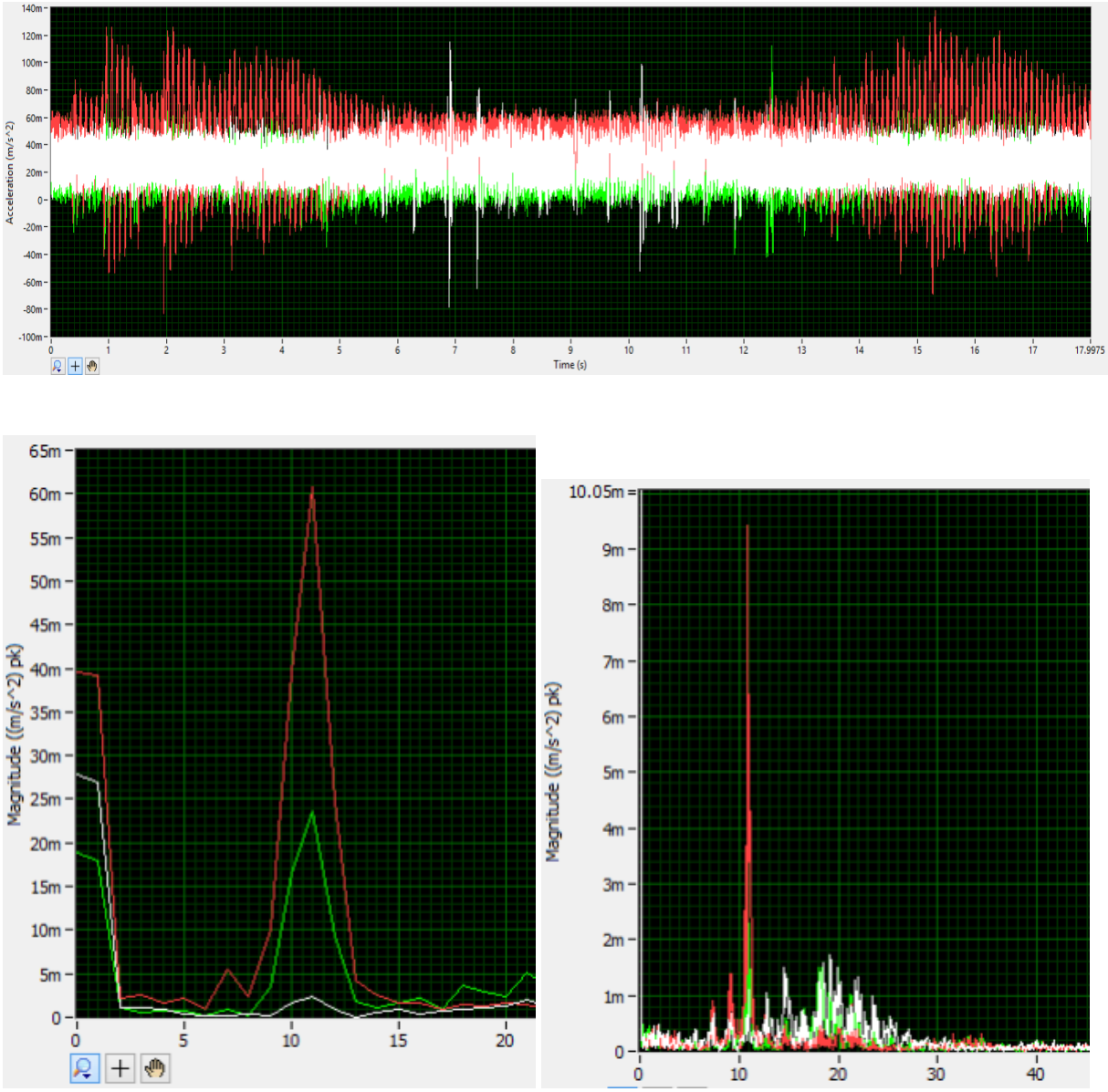

Figure 39. Measured acceleration for the whole period (a), maximum acceleration in the period of 1 second (Hz on the x-axis) (b), and maximum acceleration for the whole measurement period (c).

Measurement 10 – walking frequency 1,658 Hz.

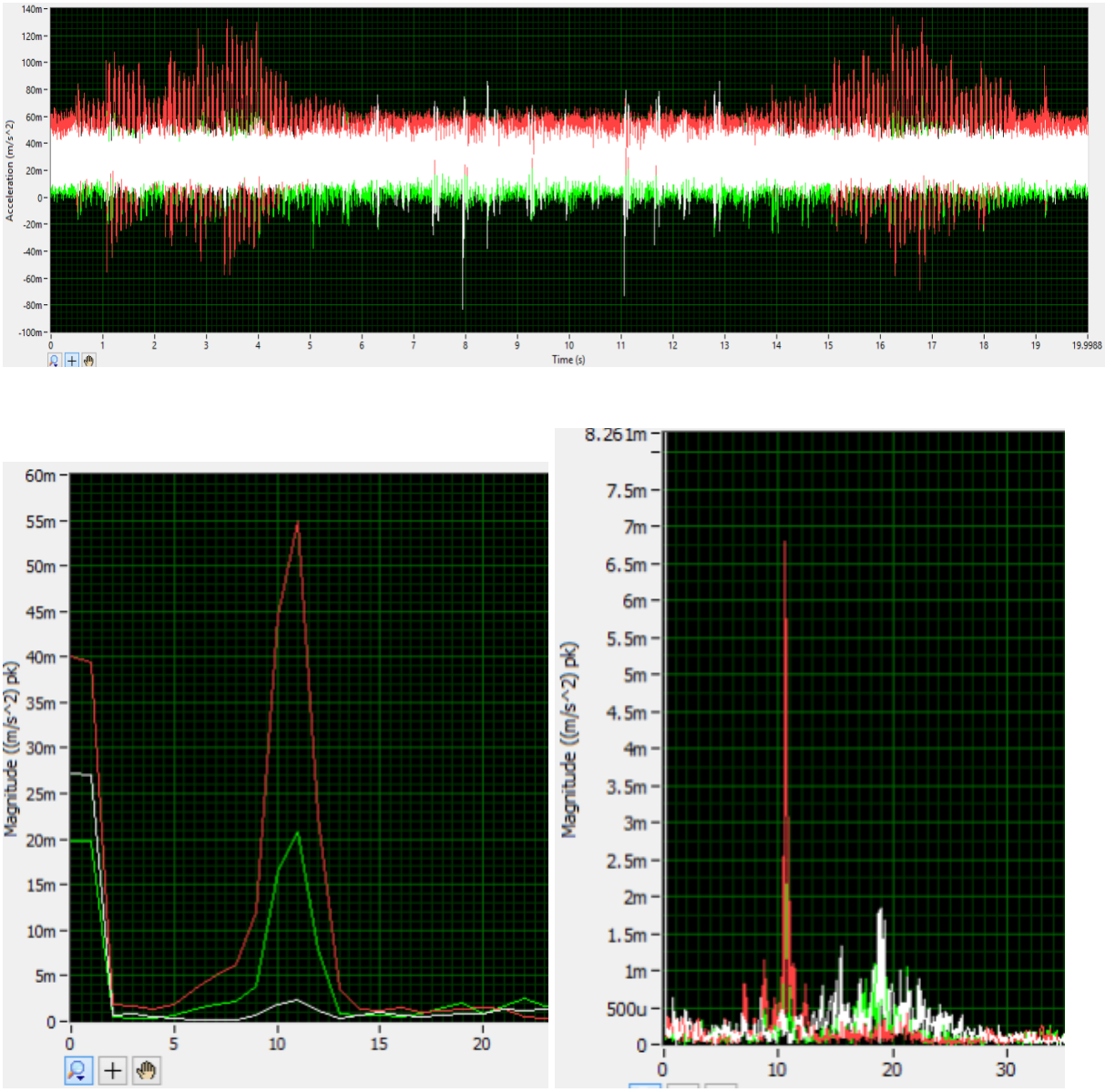

Figure 40. Measured acceleration for the whole period (a), maximum acceleration in the period of 1 second (Hz on the x-axis) (b), and maximum acceleration for the whole measurement period (c).