Paper 189

©Civil-Comp Press, 2012
Proceedings of the Eleventh International Conference
on Computational Structures Technology,
B.H.V. Topping, (Editor),
Civil-Comp Press, Stirlingshire, Scotland

A Modular Finite Element Model for Analysis of Vibration Transmission in Multi-Storey Lightweight Buildings

L.V. Andersen[1], P.H. Kirkegaard[1], K. Persson[2], N. Kiel[1] and B. Niu[3]
[1] Department of Civil Engineering, Aalborg University, Denmark
[2] Division of Structural Mechanics, Lund University, Sweden
[3] Department of Production and Mechanical Engineering
Aalborg University, Denmark

Abstract

The paper concerns the development of a modular parametric finite-element model that can be applied to the analysis of vibro-acoustic problems in relation to multi-storey lightweight structures. Floors and walls can be modelled as structural elements, or substructures may be utilised for each type of module. A numerical example shows how the model can be employed to analyse a building with typical wooden panels made of plates on stud or joist frames. Computation times of less than half a second are obtained for a single frequency with a discretisation that is valid up to at least 250 Hz.

Keywords: finite element method, modular, parametric, building, acoustics, vibration, dynamics.

1 Introduction

Transmission of sound and vibration in the built environment is a nuisance to people working and living in buildings. Noise may come from external sources such as road and railway traffic or construction sites, e.g. from pile driving. Furthermore, noise may propagate from one location internally in a building to another room within the same building from lifts, unbalanced washing machines, home cinema audio systems, or footsteps on floors and in staircases.

Assessment of noise propagation in the design of new buildings demands a method for analysis of vibration and sound transmission. For heavy structures built of concrete or masonry, methods based on statistical energy analysis (SEA) have been found to provide results of adequate accuracy within the high-frequency range [1]. Along this line of methods, the European code EN 12354 [2] is agreed to provide acceptable result regarding the overall sound pressure levels in a building made of heavy construction materials. Thus, airborne and structure borne sound are predicted efficiently and with adequate accuracy.

However, at low to medium frequencies and for lightweight structures made of, for example, wooden panels on timber frames, SEA and EN 12354 have several shortcomings. In general, the modal density is low, and especially for lightweight structures the periodicity of ribs, studs and beams leads to the presence of pass bands and stop bands in the frequency ranges relevant to audible sound as well as vibration recognizable by the human body. This is in contradiction to the assumption behind SEA that the Eigen modes of the structure should be distributed uniformly over the frequency range.

Henceforth, for analysis and design of lightweight building structures regarding their dynamic and acoustic performance in the low to medium frequency range, SEA-based methods cannot be employed. Instead this paper proposes the use of finite-element analysis (FEA). A model is constructed in the commercial FEA software package ABAQUS [3]. The model is modularized to allow easy implementation of new panels, materials etc. into the building. Further, the model is parameterized such that parameter studies can easily be performed. To obtain this, the model is constructed with an artificial skeleton of beams having (almost) no structural mass and stiffness. The wall and floor panels are coupled to this skeleton which provides a simple setup of the global finite-element model. For simple, preliminary analyses, the model allows the use of isotropic or orthotropic homogeneous shells representing the individual walls and floors. More detailed analysis can be carried out by use of macro finite elements obtained by system reduction of rigorous models.

The overall idea and specification of the model, and an illustrative example of its use, are presented in this paper. Firstly, Section 2 contains a detailed discussion of the formulation of the modular parametric model with focus on the choices that must be made regarding model complexity, connectivity of the global model, post processing capabilities et cetera. Secondly, Section 3 provides a numerical example, in which the model has been utilised for analysis of a four-storey wooden building with walls and floors modelled as panels with plates on joists or studs. Finally, Section 4 lists the main conclusions.

A number of issues related to modelling of individual wall and floor panels, joints between panels and frames, as well as accuracy of the formulation and various modelling techniques are discussed in the companion papers by the authors and their co-workers. Thus, Kiel et al. [4] perform a comparative study of different approaches to the coupling of panel modules. Flodén et al. [5] examine a structure consisting of a few wooden panels, made as double-leaf plates with joist or stud ribs, using different system reduction techniques with the aim of determining a proper substructure technique. Finally, Niu et al. [6] conduct a parameter study of a three-dimensional building based on the modular finite-element model developed in the present paper.

2 Concept of a modular finite-element building model

2.1 Abstractions and modelling considerations

A modular finite-element (FE) model consists of a number of modules. These modules form the building blocks of the FE model, and a choice has to be made regarding the level of detail and adaptability that is required in the modelling process. A single module can be a structural member, such as a stud or plate, or it can be an entire section of a building, e.g. a flat with two rooms including all walls, floors, ceilings and, possibly, the internal air. In any case, the modules must be connected to each other to form the global building model, and a number of simplifications may be required to allow parameter studies and analysis within the medium-to-high-frequency range. In the following subsections, a number of qualified choices will be made in relation to the formulation of a modular parametric FE model.

2.1.1 Definition of a module

The terminology "module" must be defined. In the context of a lightweight building structure, the following possible definition are identified:

- Each individual stud, joist, beam, plate or similar structural member is considered a module;
- horizontal and vertical divisions (i.e. floors and walls) are defined as modules;
- a module consists of a room, or of an entire flat.

The proper choice of module definition may depend on the purpose of the model. However, if the first definition leads to a high number of modules, resulting in a high complexity of the global model. On the other hand, if a module represents an entire room, assembly of the global model may be difficult. For example, a wall panel may belong to either of the adjacent rooms; but a modularisation requires that all divisions are defined as belonging to a single room, i.e. a single module. Hence, the second approach is suggested. Using each wall or floor panel as a module provides enough freedom to model buildings with a complex geometry, and at the same time the number of (different) modules is manageable in a global model.

The chosen module definition has the disadvantage that air is not simply included within the rooms of a building. However, recent research by the authors shows that air in a building provides additional damping of the structural vibrations, but there is no indication that resonance in the air will lead to increased response. Further, the added mass of the air may only lead to a small decrease in the Eigen frequencies of the structure, and only a small increase of the Eigen frequencies may occur due to an air-cushion effect, where the air inside a room acts like a spring. Thus, it is proposed to model the building without the air in the analysis phase. The acoustic pressure in a room can be found by post processing, assuming a weak coupling between the air and the structure as suggested by, among others, Fiala et al. [11, 12].

The geometrical and material properties of a module are defined by parameters, and further parameters define the overall geometry, connectivity and boundary conditions of the building. On the local level, parameters may define:

- the length, width and height of each individual panel;
- the thickness of plates and the position of any cutaways, e.g. door openings;
- the length, width, thickness and position of studs and joists;
- material properties defining the stiffness, mass and material damping.

Typically, the stiffeners internally in a wall, floor or roof structure are periodic, and their position will be defined in terms of a centre-to-centre distance.

2.1.2 Finite-element model of a single module

The individual members constituting a module can be modelled as structural elements, i.e. beams and shells, or a three-dimensional solid model can be utilised. The main advantage of a module based on structural elements lies in the smaller number of degrees of freedom compared to a solid model. However, the specification of cross-sectional properties and materials is not straightforward for such models, given that a close resemblance to the behaviour of a more advanced three-dimensional solid model or measurements is required. Flodén et al. [5] discuss the shortcomings related to the use of structural elements for dynamic analysis of lightweight floors and walls. An attempt to match an orthotropic shell model to a solid three-dimensional model is presented in the accompanying papers by Niu et al. [6], assuming that adequate results may be achieved at low frequencies when the first modes of resonance for each panel are represented with sufficient accuracy. However, with regard to the global response of a building, that individual panels may act as discs or shear panels. Here the response is mainly dependent on the in-plane stiffness of the panel and only to a lesser extent on the flexural stiffness. On the other hand, the first local modes are primarily influenced by the bending stiffness, which should be taken into consideration when, for example, a shell model is calibrated to a three-dimensional solid model of a floor panel.

A module based on a rigorous modelling by means of three-dimensional elements has the great advantage that a very detailed analysis can be made, obtained results in close agreement with measured vibration levels for a wooden panel. However, such a module FE model will have thousands of degrees of freedom, making it unfit for construction of a global model by simple assembly. For example, the building analysed in Section 3 would have about 2,000,000 degrees of freedom if a model were employed with the given discretisation into solid finite elements. The model only contains eight rooms, and it is only valid up to about 250 Hz. Instead, as proposed by Flodén et al. [5] and implemented into the numerical example in Section 3, a substructure approach should be considered. Hence, for each rigorous model, a macro finite element, or substructure, is formed based on model reduction. The component mode synthesis approach [9, 10] available in ABAQUS has been found to provide highly accurate results for a lightweight wooden panel [5]—even with few Eigen modes augmenting the results obtained by standard Guyan reduction [8].

A similar model reduction is possible for modules that are originally modelled by structural elements. However, this is not recommended. Firstly, the calibration of a simpler model to a rigorous model is superfluous if a model reduction is performed in any case. Secondly, the main advantage of the structural-element approach is that all elements can be kept in the global model, which allows post processing and presentation of results in a simple manner without back calculation of the response in each panel from the retained degrees of freedom in the corresponding substructure. The relatively cumbersome post processing of models based on a substructure approach is actually one of its main drawback. This is further discussed in Section 3.

2.1.3 Model connectivity

Assembly of modules into a global building can be done in different ways. As discussed by Kiel et al. [4], the following methods may be suggested:

- Modules are assembled such that nodes with the same position in the global model have shared degrees of freedom. Duplicate nodes are only present if some degrees of freedom, e.g. the rotations, are untied at the interface between two modules.

- Modules are tied together by introduction of a set of auxiliary equations that bind the degrees of freedom in one module to the surface, or nodes, of another module along their common interface.

- All modules are connected to a common skeleton that may represent a real structure, e.g. a solid timber frame, or it may be artificial with very small mass and stiffness compared to the panels.

The first approach requires that the position of nodes along interfaces is the same for all adjacent modules, whereas the two other methods may allow more freedom in the placement of nodes along common edges. However, as outlined by Kiel et al. [4], alignment of the nodes is a requirement in any case to avoid inaccurate results.

Regarding the implementation of the first method into ABAQUS, a number of “parts”, each representing a module, can be assembled into one common part by use of the so-called “merge” technique [3]. However, in this case duplicate nodes are removed in the global model, i.e. all degrees of freedom are shared on interfaces between modules. Hence, the freedom to untie rotations at an interface does not exist by this approach.

In ABAQUS, the second kind of coupling is achieved by introduction of so-called "tie constraints" between a "master" and a "slave" defined in terms of nodes or surfaces. However, a set of nodes or element surfaces, e.g. the edge of a wall panel FE model, can only act as "slave" to one "master", whereas a single "master" can have many "slaves". Hence, at corners with more than two panels meeting, it may be difficult to define a hierarchy of "masters" and "slaves" that permits a coupling between all adjacent panels. This problem is overcome by the third approach, given that the skeleton is defined as the "master" and all wall and floor modules are introduced as "slaves". Hence, this approach is recommended.

When an artificial skeleton is employed, one panel should always be fully coupled to each segment of the skeleton to avoid degrees of freedom with no or very small stiffness and mass leading to a (nearly) singular system of equations. More details regarding the implementation of an artificial skeleton can be found in Ref. [4].

With the simple "tie constraints" in ABAQUS, a choice can be made between coupling of all degrees of freedom or only the translational degrees of freedom. In any case, a given degree of freedom is either fully coupled or completely uncoupled. More realistic behaviour of joints between structural modules can be obtained by:

- Inserting joint elements between the individual modules and the skeleton;
- Incorporating the physics of the joints into the modules, e.g. by means of the substructure approach.

The second method is proposed since it does not lead to a further complication of the global model. Any changes are made at module level, i.e. the global assembly procedure remains the same.

2.2 Analysis and post processing considerations

The main purpose of the modular, parametric finite-element model is to study the vibro-acoustic performance of a building. Regarding the structural response, the following analyses should be made possible by the FE model:

- Extraction of real, undamped as well as complex, damped Eigen frequencies and the corresponding Eigen modes;
- Frequency-domain analysis of the steady state response to forced vibrations;
- Time-domain analysis of the transient response to forced vibrations.

These methods of analysis are all available in ABAQUS. However, the application of forces and post processing requirements provide some limitations.

ABAQUS allows the construction of substructures in a convenient manner. But loads within the global model can only be applied at nodes which are retained in the global model. Thus, for example, if a step load is to be defined at the middle of a floor panel, a node must be retained at this position within the substructure of that panel. For static analysis of a building, loads may be defined at the substructure level, and these loads can be activated later in the global model. However, this approach is not implemented for dynamic analysis in the current version of ABAQUS [3].

Another issue concerns the post processing related to evaluation of the acoustic pressures inside the building—a calculation that requires knowledge of the displacement at all points on the surface of the walls, floors and ceilings. This information can be achieved by back calculating the response of the internal degrees of freedom in each substructure from the retained degrees of freedom. However, such back calculation is not supported with modal analysis in ABAQUS [3]. Hence, a computation based on direct analysis of the full global FE model is suggested. However, as illustrated in the numerical example below, the computation time related to direct steady state analysis is not significantly higher than the computation time related to modal steady state analysis. Finally, the direct approach allows a wider range of material damping models to be implemented into the FE model.

2.3 Concretisation of the finite-element model

As illustrated in Figure 1, a cartesian coordinate space is introduced with the origin at one corner of the building. The modular building finite-element model is constructed with all modules placed parallel to the coordinate planes. The length and width directions of the building are along the x and z-axes, respectively, and with the facades and vertical divisions placed parallel to the x–y and y–z planes, respectively, cf. Figure 1. The height direction is along the y-axis with horizontal divisions (including the roof) placed parallel to the x–z plane. To keep things simple, all modules with a given orientation are identical, but with the possibility of assigning properties to exterior walls of the building. The module lengths along the x, y and z-axes are denoted L, H and W (length, height and width), respectively.

A unique identification of all floor modules is achieved by denoting horizontal divisions as “yFloor l-m-n”. Here l, m and n are integers with l counting along the x-direction, m counting along the y-direction and n counting along the z-direction, all starting from the value 1 at the origin of the coordinate system. In a similar manner, vertical divisions are identified as “xWall l-m-n” and “zWall l-m-n”, respectively. Furthermore, rooms are numbered as “Room l-m-n”. Finally, as indicated in Figure 1, a geographic orientation of the building is chosen such that the length direction, i.e. the x-axis, goes from west to east, whereas the width direction, i.e. the z-axis, goes from north to south.

The modular parametric finite-element model is implemented in ABAQUS via a script, using the Python language. The scripting approach ensures a fast setup of the modules in the desired room configuration, as few bits of code take care of many duplications of modules, boundary conditions and constraints.

The module concept triggers the opportunity to analyse altering module or room configurations as only two or three modules have to be changed for a new setup in any

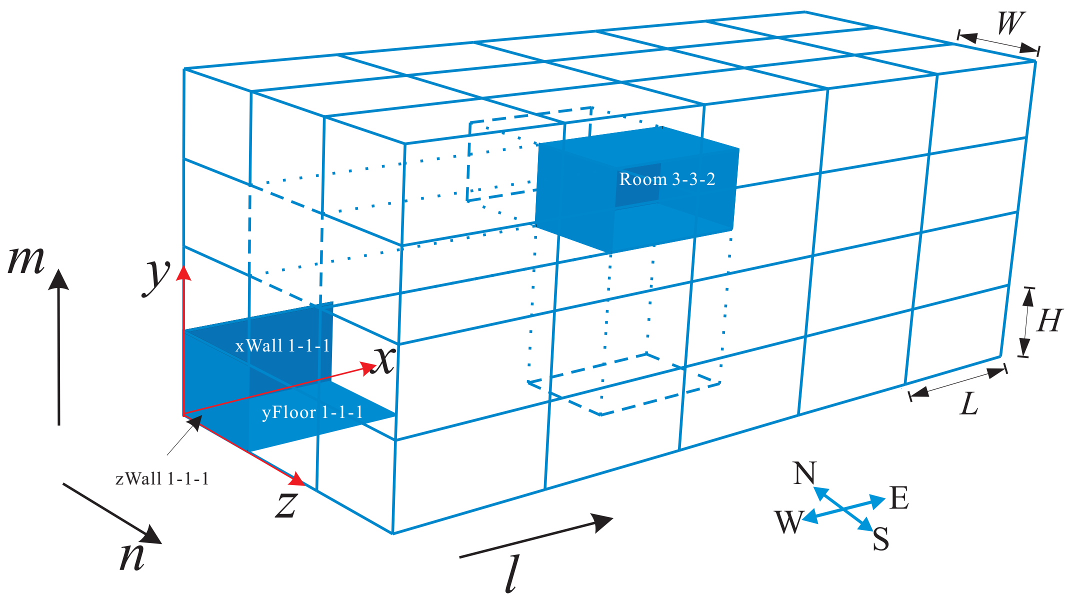

Figure 1: Global layout of modular parametric building model.

given room configuration. And as long as simple shells are used for the wall and floor panels, this can be handled by the script for the entire model by it self.

As discussed above, a skeleton is introduced. It is constructed as a number of beams and columns which are merged together to form a single part that is used as “master” for tie constraints with all wall and floor modules. In a modular FE model based on the substructure approach, the following degrees of freedom will remain:

- the degrees of freedom (translation and rotation) of the nodes in the skeleton;
- degrees of freedom corresponding to the retained internal modes of resonance in each substructure;
- any retained translational or rotational degrees of freedom in the substructures that are not tied to the skeleton, e.g. at an internal node where a concentrated force is applied.

For modules based on structural elements, the number of degrees of freedom in the global model will be higher, but as discussed above the post processing may be easier. In the particularly simple case, where easily scalable modules, i.e. homogeneous isotropic panels with fixed thicknesses, the model can be setup solely based on a few parameters. In addition to the parameters L, H, W, l, m, and n these are the thicknesses of the wall and the floor panels, the mass density and material properties defining the stiffness, e.g. Young’s modulus and Poisson’s ratio, and material dissipation, e.g. a loss factor. If substructures are employed instead of the simplified shells, separate ABAQUS models based on Python scripts are utilised to configure the modules prior to the definition of the global model as demonstrated below.

3 Example modular building

To demonstrate the capabilities of the proposed modular, parametric finite-element model, a simple four-storey building is analysed. With reference to Figure 1, the following parameters are employed to specify the dimensions of the modules:

- Length of modules in the length direction, i.e. the x-direction: $L = 4.8$ m;
- length of modules in the height direction, i.e. the y-direction: $H = 2.7$ m;
- length of modules in the width direction, i.e. the z-direction: $W = 3.6$ m.

Further, the numbers of modules in each coordinate direction are given as:

- Number of modules in the length direction, i.e. the x-direction: $l = 1, 2$;
- number of modules in the height direction, i.e. the y-direction: $m = 1, 2, 3, 4$;
- number of modules in the width direction, i.e. the z-direction: $n = 1$.

All horizontal divisions except for the ground floor are modelled and all vertical divisions in the north–south direction are included, assuming identical properties of the external and internal walls. Only the west–east walls along the facade identified by $n = 1$ are present, i.e. there are no vertical divisions along the west–east facade at $n = 2$. The individual modules are modelled by the substructure approach. Further details on the geometrical and material properties of each module are given below, and the substructure generation and its accuracy are described.

3.1 Substructure for horizontal divisions

The floor module has the dimensions $(L \times W) = 4.8 \times 3.6$ m^2. As illustrated in Figure 2, it is constructed as a panel with two plates connected to a frame. The thickness of each plate is 20 mm, and the total height of the deck is 200 mm. The studs have a width of 50 mm, a height of 160 mm and are placed parallel to the x-axis, i.e. along the length direction of the building. The centre-to-centre distance of the studs is 600 mm with the exception that the first two studs in either side of the panel have a distance of 575 mm such that the exterior edges of the outmost studs coalesce with the exterior edges of the plates. The panel is made of construction wood, idealized as a homogenous isotropic linear viscoelastic material with mass density 550 kg/m^3, Young's modulus 14 GPa, Poisson's ratio 0.35, and structural damping in terms of a frequency independent loss factor of 1%.

A solid finite-element model is created in ABAQUS by a Python script, employing 20-node brick elements with second-order spatial interpolation of the displacement field and full integration. The mesh size is chosen as 100 mm, ensuring accurate results at a frequency of 250 Hz which is the maximum frequency of interest.

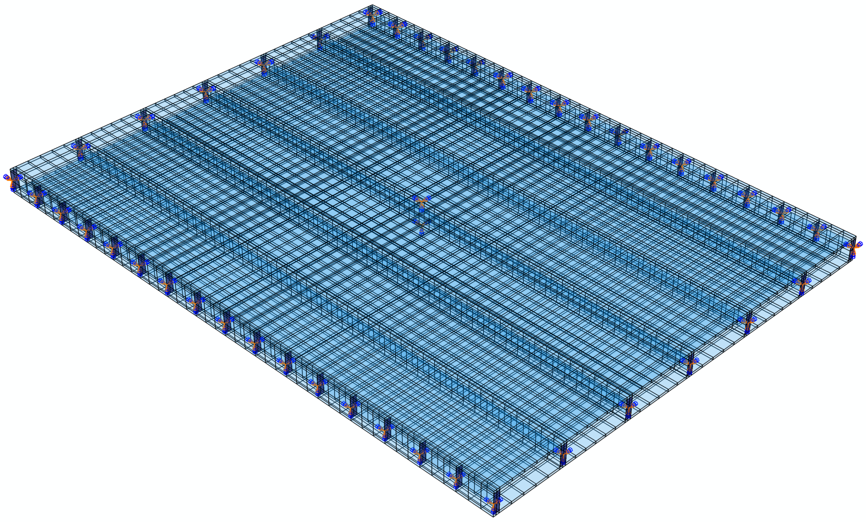

Figure 2: Geometry of the 4.8×3.6 m^2 floor panel model. The orange and blue arrow heads indicate fixed (retained) translational and rotational degrees of freedom.

Based on the rigorous three-dimensional solid model, a substructure is extracted. Nodes are retained for every 300 mm along the edges of the panel. The nodes all lie in the midplane of the panel and to allow retainment of translations as well as rotations, small shells with a high stiffness are inserted into the cross-sections of the studs at all positions where the degrees of freedom are to be retained, e.g. the ends of the studs. In addition to the translational and rotational degrees of freedom along the edges of the panel, one node is retained on the middle of the top plate and on the middle of the bottom plate to allow the excitation by concentrated harmonically varying forces at these positions. Finally, the first 20 internal modes of resonance for the panel are kept. These modes are extracted by an Eigen frequency analysis in which the panel is fully fixed at the nodes that are retained in the substructure, cf. Figure 2.

The resulting substructure model has 44 nodes with six degrees of freedom (the retained boundary nodes) and two nodes with three degrees of freedom (the load application nodes) in addition to the twenty internal degrees of freedom related to the first twenty mode shapes of the fully clamped panel. This provides a total of 290 degrees of freedom for the substructure of the 4.8×3.6 m^2 floor panel, whereas the full FE model has 38245 nodes with a total of 117507 degrees of freedom. Nonetheless, a very good accuracy is uphold regarding the dynamic response up to 250 Hz. This is documented by comparison of the Eigen frequencies and their related mode shapes obtained by analyses of the original solid model and the substructure for a floor panel that has been simply supported along the edges. With reference to Figure 3 and Table 1, an almost perfect match is obtained. It is noted that a minimum of 1509 MB of physical memory is required in the full FE model to ensure that the computation stays in core, thus minimising input/output. Only 25 MB is required for the substructure model. Given that these requirements are fulfilled, the total computation time (CPU time) on an Intel i7 CPU 860 @ 2.80 GHz (one core) is 53.30 s for the full FE model and 1.400 s for the model based on the substructure of the floor panel.

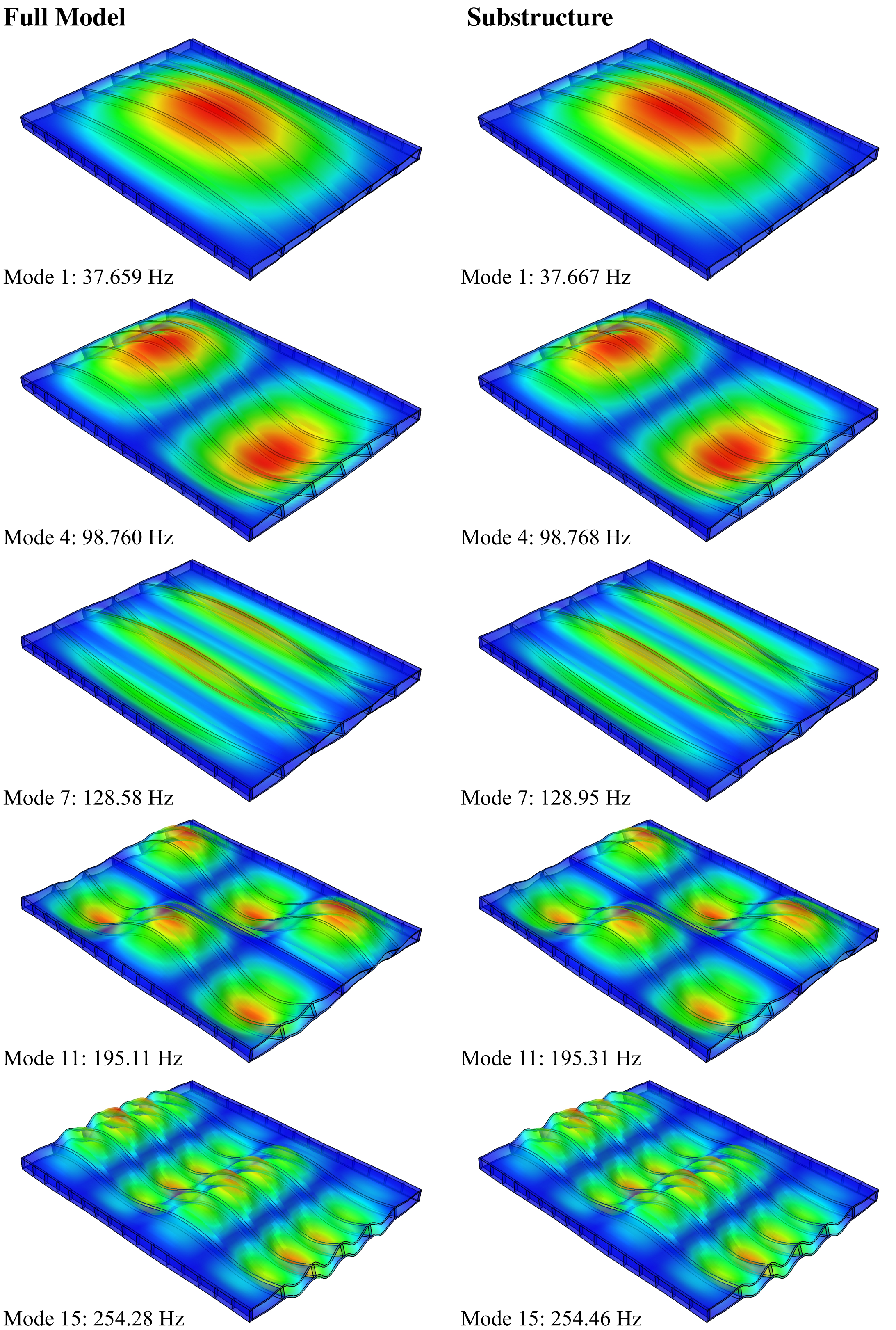

Figure 3: Selected mode shapes and related Eigenfrequencies for the 4.8×3.6 m^2 floor: Results obtained with full model (left) and results obtained with substructure (right). Red and blue colours indicate large and small displacement magnitudes, respectively.

Mode No.	Full model	Substructure	Deviation
Mode 1	37.659 Hz	37.667 Hz	0.0212%
Mode 2	61.751 Hz	61.752 Hz	0.0016%
Mode 3	87.569 Hz	87.676 Hz	0.0011%
Mode 4	98.760 Hz	98.768 Hz	0.0081%
Mode 5	110.98 Hz	110.98 Hz	0.0000%
Mode 6	117.39 Hz	117.40 Hz	0.0085%
Mode 7	128.58 Hz	128.95 Hz	0.2877%
Mode 8	144.89 Hz	144.92 Hz	0.0207%
Mode 9	173.68 Hz	173.81 Hz	0.0749%
Mode 10	181.97 Hz	182.74 Hz	0.4231%
Mode 11	195.11 Hz	195.31 Hz	0.1025%
Mode 12	197.61 Hz	197.73 Hz	0.0607%
Mode 13	219.78 Hz	221.94 Hz	0.9828%
Mode 14	250.44 Hz	250.88 Hz	0.1757%
Mode 15	254.28 Hz	254.46 Hz	0.0708%

Table 1: Eigen frequencies for the 4.8×3.6 m^2 floor.

3.2 Substructures for vertical divisions

The walls in the west–east direction (the "xWalls") have the dimensions $(H \times L) = 2.7 \times 4.8$ m^2, whereas the walls in the north–south direction (the "zWalls") have the dimensions $(H \times W) = 2.7 \times 3.6$ m^2. Both wall modules have a total thickness of 100 mm. Similarly to the floor panel, the walls are constructed as plates connected to a frame. The joists are all vertical and placed with a centre-to-centre distance of 600 mm, again except for the outermost profiles which are aligned with the side edges of the plates. The thickness of the plates is 20 mm for the "xWalls", whereas a smaller thickness of 10 mm is employed for the "zWalls". No difference is made between interior and exterior wall panels. Finally, the material properties of the wall modules are the same as those of the floor module. The finite-element models for the respective modules are visualised in Figures 4 and 5.

Substructures are extracted for the two wall modules, again introducing 20 internal modes and retaining nodes for every 300 mm along the edges of the midplane. Regarding the "xWall" module, the following observations can be made:

- The substructure has 230 degrees of freedom, whereas the full model has 87783 degrees of freedom;
- Minimization of input/output requires a minimum of 1096 MB of RAM for the full model but only 25 MB for the substructure model;
- The CPU time on an Intel i7 CPU 860 @ 2.80 GHz (one core) is 32.00 s for the full model and 1.1000 s for the substructure model.

Likewise, for the 2.7×3.6 m^2 "zWall" module placed along the north–south direction:

- The substructure has 206 degrees of freedom, whereas the full model has 67485 degrees of freedom;
- Minimization of input/output requires a minimum of 839 MB of RAM for the full model but only 25 MB for the substructure model;
- The CPU time on an Intel i7 CPU 860 @ 2.80 GHz (one core) is 30.50 s for the full model and 0.9000 s for the substructure model.

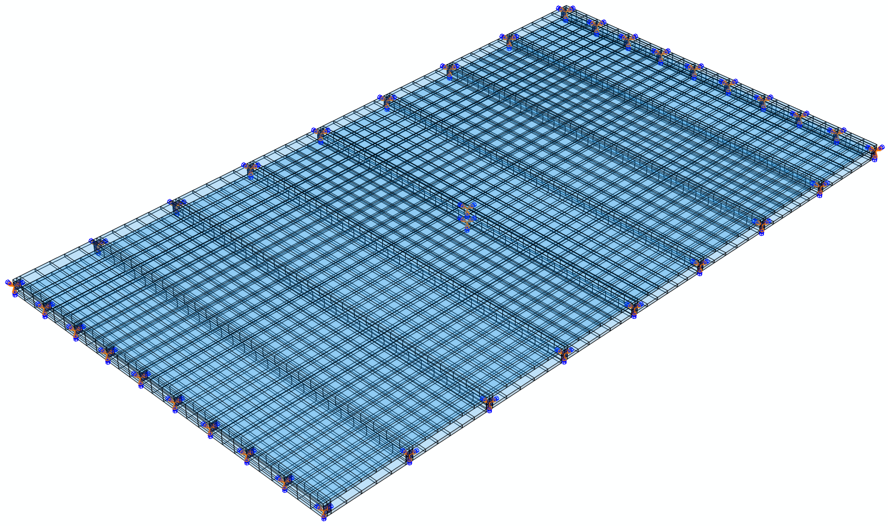

Figure 4: Geometry of the 2.7×4.8 m^2 wall panel model. The orange and blue arrow heads indicate fixed (retained) translational and rotational degrees of freedom.

Mode No.	Full model	Substructure	Deviation
Mode 1	42.712 Hz	42.722 Hz	0.0234%
Mode 2	56.151 Hz	56.151 Hz	0.0000%
Mode 3	74.110 Hz	74.174 Hz	0.0863%
Mode 4	93.584 Hz	93.587 Hz	0.0032%
Mode 5	112.95 Hz	113.20 Hz	0.2213%
Mode 6	128.36 Hz	128.37 Hz	0.0078%
Mode 7	130.62 Hz	130.65 Hz	0.0230%
Mode 8	136.34 Hz	136.35 Hz	0.0073%
Mode 9	143.53 Hz	144.50 Hz	0.6758%
Mode 10	149.12 Hz	149.18 Hz	0.0402%
Mode 11	166.03 Hz	166.12 Hz	0.0542%
Mode 12	185.48 Hz	185.60 Hz	0.0647%
Mode 13	205.24 Hz	205.38 Hz	0.0682%
Mode 14	220.16 Hz	221.94 Hz	0.8085%
Mode 15	222.05 Hz	222.29 Hz	0.1081%

Table 2: Eigen frequencies for the 2.7×4.8 m^2 wall.

The accuracy of the substructure models is analysed by comparing the Eigen frequencies and Eigen modes with those of the full model, assuming that the panels are simply supported along the edges. Based on the results listed in Tables 2 and 3, a very good accuracy has been achieved for the first 15 Eigen frequencies. However, for the 2.7×3.6 m^2 "zWall" module, a mode shift occurs at mode 12. Here, the substructure behaves stiffer than the original solid model, cf. Figure 6.

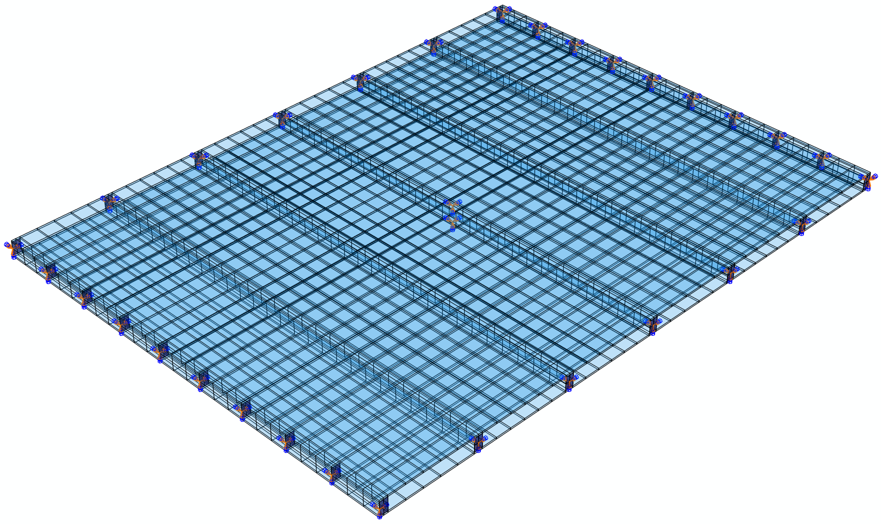

Figure 5: Geometry of the 2.7×3.6 m^2 wall panel model. The orange and blue arrow heads indicate fixed (retained) translational and rotational degrees of freedom.

Mode No.	Full model	Substructure	Deviation
Mode 1	42.384 Hz	42.444 Hz	0.1416.%
Mode 2	52.414 Hz	52.421 Hz	0.0134%
Mode 3	66.997 Hz	67.328 Hz	0.4941%
Mode 4	82.005 Hz	82.046 Hz	0.0500%
Mode 5	94.062 Hz	95.813 Hz	1.8275%
Mode 6	121.66 Hz	121.93 Hz	0.2219%
Mode 7	127.70 Hz	128.00 Hz	0.2349%
Mode 8	139.56 Hz	140.34 Hz	0.5589%
Mode 9	155.51 Hz	157.20 Hz	1.0867%
Mode 10	164.53 Hz	165.61 Hz	0.6564%
Mode 11	166.93 Hz	167.18 Hz	0.1497%
Mode 12 (16)*	170.56 Hz	172.68 Hz	1.2430%
Mode 13 (12)*	171.32 Hz	171.32 Hz	0.0000%
Mode 14 (13)*	171.54 Hz	171.54 Hz	0.0000%
Mode 15 (14)*	171.77 Hz	171.77 Hz	0.0000%

*: Modes 12, 13, 14 and 15 in the full model correspond to modes 16, 12, 13 and 14 in the substructure model.

Table 3: Eigen frequencies for the 2.7×3.6 m^2 wall.

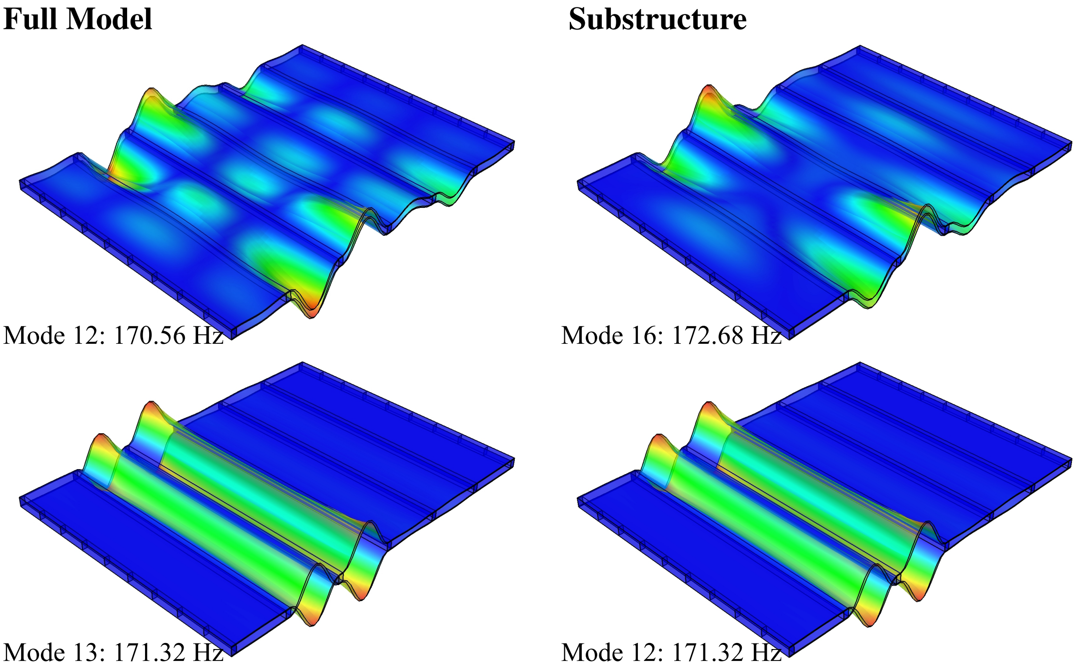

Figure 6: Selected Eigen frequencies and related mode shapes for the 2.7×3.6 m^2 wall: Results obtained with full model (left) and results obtained with substructure (right). Red and blue colours indicate large and small displacement magnitudes, respectively.

3.3 Building model

The three different substructure modules are assembled into a global finite-element model. In accordance with the number of rooms in each coordinate direction, the resulting model is referred to as Building 2-4-1. The room numbering is defined in Figure 7. The artificial skeleton is made of beam elements having a cross-sectional area of 10×10 mm^2. The mass density of the skeleton is 1 kg/m^3, while Young's modulus is 1 GPa, and Poisson's ratio is defined as 0. The low stiffness and mass ensures minimal influence on the behaviour of the real structure while at the same time avoiding spurious modes in the skeleton in the considered frequency range from 0 to 250 Hz. The finite-element model of the entire building contains 4697 degrees of freedom and requires 75 MB of RAM to run without going out of core.

Firstly, 569 undamped Eigen modes up to 250 Hz are extracted using a symmetric Lanczos solver. The Eigenvalue problem is solved in 16.70 s on an Intel i7 CPU 860 @ 2.80 GHz, using a single core. Figure 8 shows the mode count, and example mode shapes are illustrated in Figures 9 to 11. Only 21 modes are present below 50 Hz. In the remaining frequency range up to 250 Hz, the modal density is relatively constant about 2 Hz^{-1}. However, a particularly high modal density is seen near 175 Hz. The first mode of resonance occurs at 10.655 Hz and is identified as a torsional model. The second mode is not illustrated but occurs at 11.389 Hz. In this mode, the entire building sways back and forth in the north–south direction. Further, it is observed in Figure 9 that already mode number 3 is related to local bending in some of the panels.

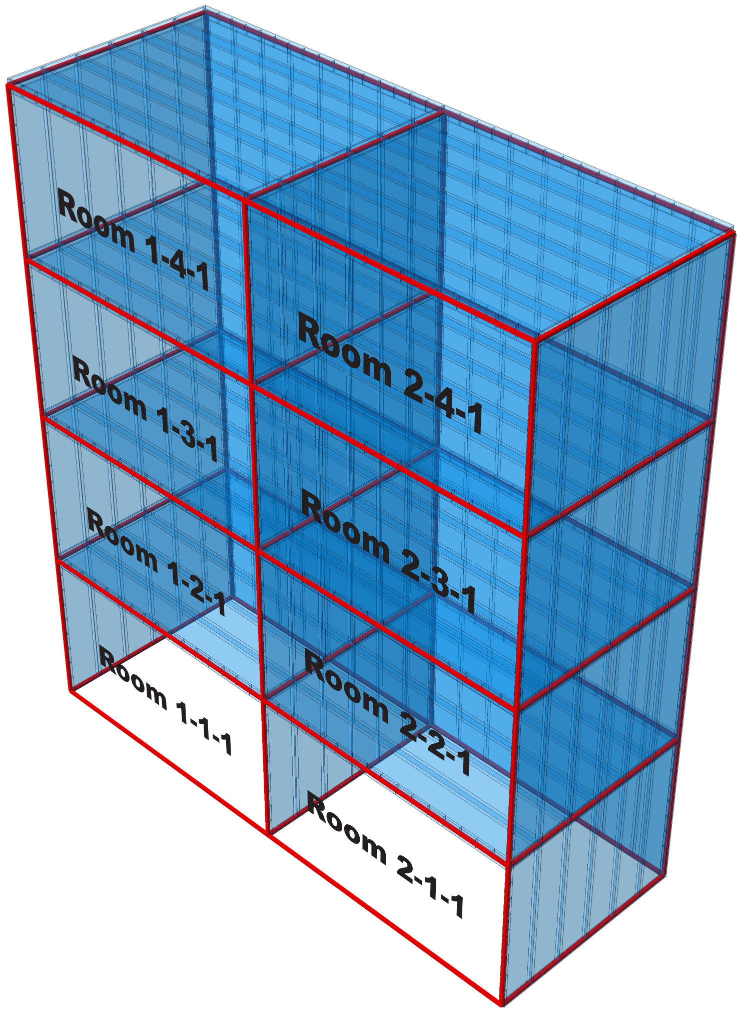

Figure 7: Geometry of Building 2-4-1. The artificial skeleton is plotted in red.

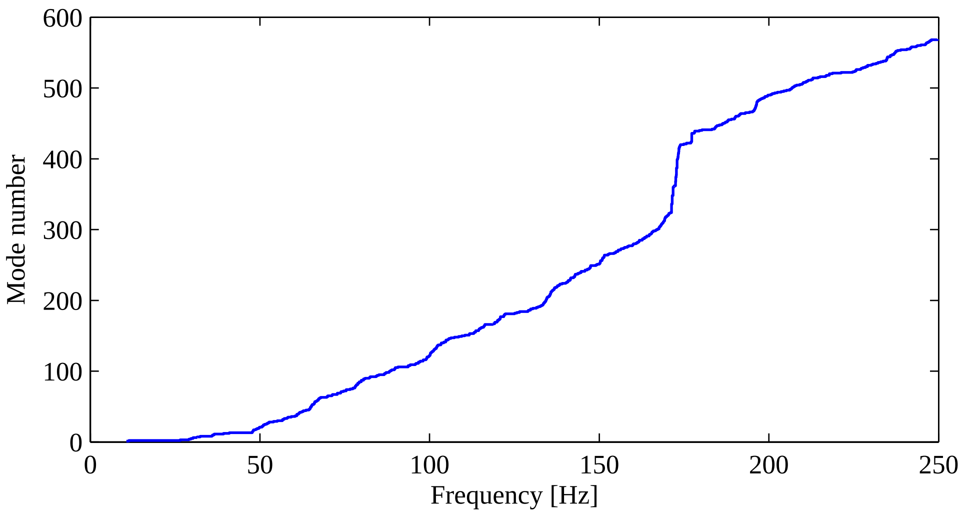

Figure 8: Mode count for Building 2-4-1.

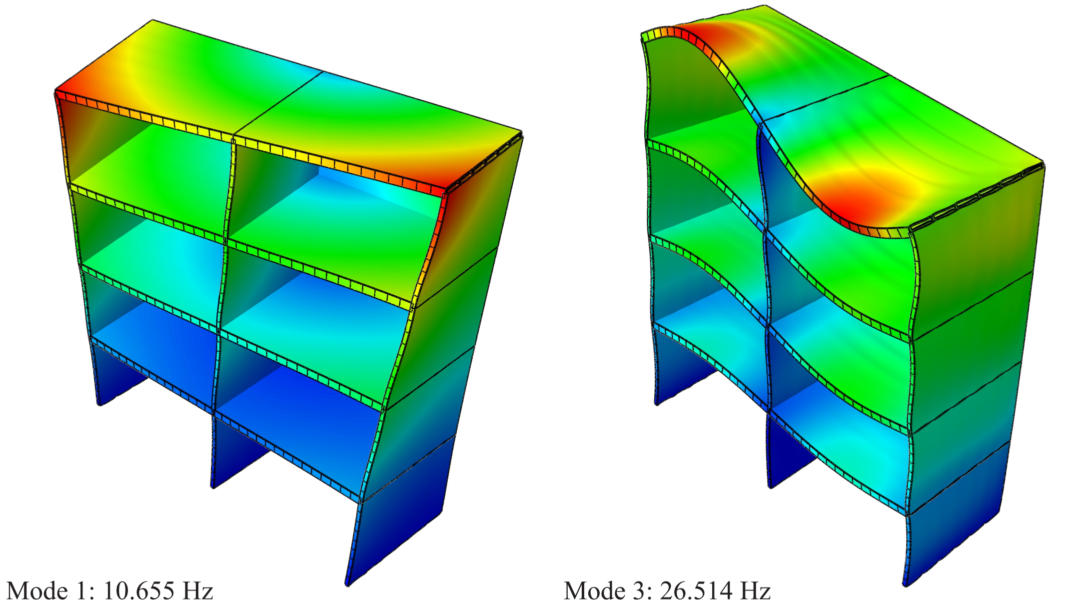

Figure 9: Eigen modes 1 and 3 of Building 2-4-1. Red and blue colours indicate large and small displacement magnitudes, respectively.

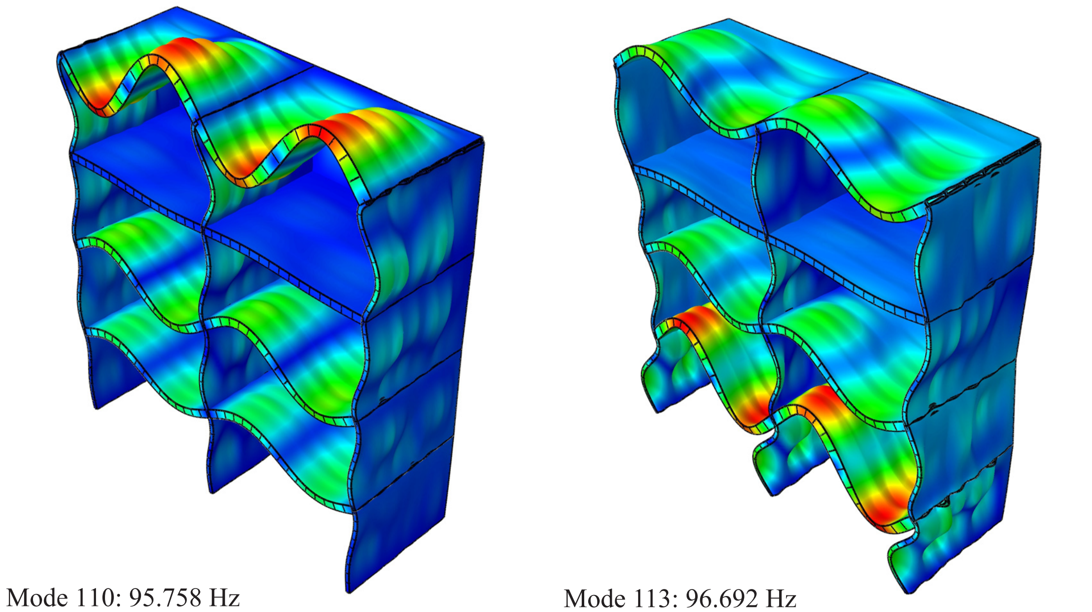

Figure 10: Eigen modes 110 and 111 of Building 2-4-1. Red and blue colours indicate large and small displacement magnitudes, respectively.

The second analysis concerns the steady state response of the building to a harmonic concentrated force applied vertically with unit magnitude on the middle of yFloor 1-2-1, i.e. the floor of Room 1-2-1, cf. Figure 7. As described above and visualized in Figure 2, a node has been retained on the middle of the floor module in the substructure generation to allow the application of forces at this position. Two

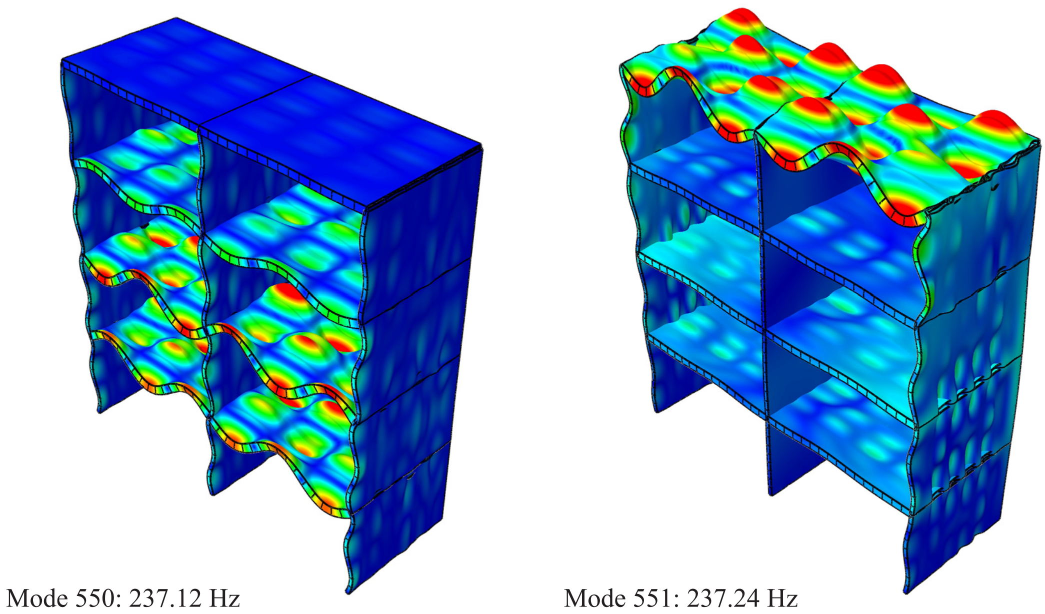

Figure 11: Eigen modes 550 to 551 of Building 2-4-1. Red and blue colours indicate large and small displacement magnitudes, respectively.

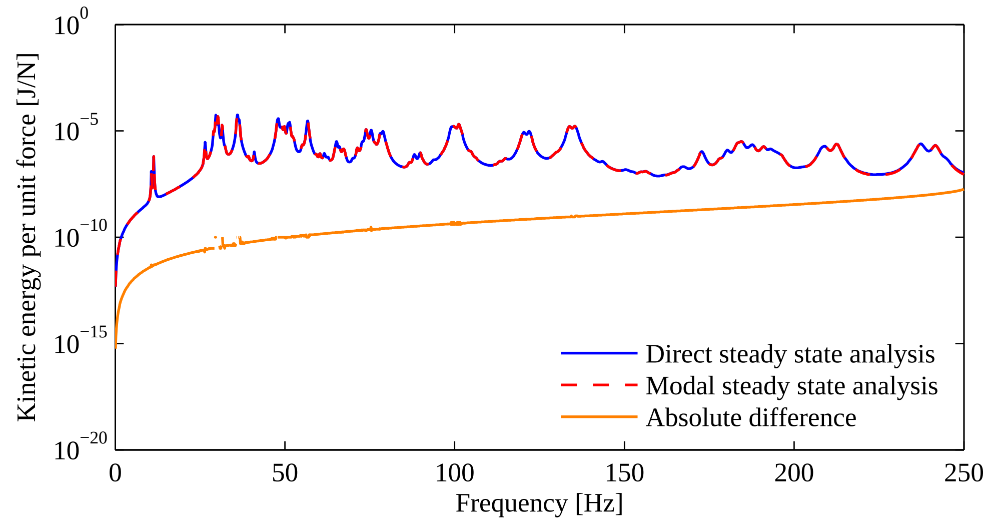

Figure 12: Kinetic energy amplitude in Building 2-4-1 for a unit magnitude harmonically varying point force applied vertically on Floor 1-2-1.

methods of analysis are employed, namely direct and modal steady state analysis. The kinetic energy is calculated for the entire model at the 2001 discrete frequencies 0, 0.125, 0.25, ..., 250 Hz, and results are shown is Figure 12. The direct analysis based on the full finite-element model composed of the panel substructure modules and the skeleton has a computation time of 0.35 s per frequency. The modal analysis requires less RAM than the direct analysis (79 MB instead of 140 MB) and the computation is slightly faster. However, when modes up to 250 Hz are used, the results are only accurate, within 1% deviation from the direct solution, for frequencies below 150 Hz.

4 Conclusions

A modular parametric finite-element model has been developed for vibro-acoustic analysis of lightweight multi-storey buildings, using Python scripting and ABAQUS. The main conclusions are listed in the following.

- Air is not included in the model. The acoustic pressure should instead be found by post processing, since the coupling between the air and the structure is weak.
- Wall and floor panels are considered as modules of the building. In the current version of the model, all floor modules are identical. However, the external walls can be made different from the interior walls in the length direction of the building as well as the width direction.
- The panels are all attached to a skeleton that can be physical in order to model a real structure of, for example, solid timber—or it may be artificial with a very small stiffness and mass to minimize influence on the modular building.
- Modules can be modelled as simple homogenous shells discretized into shell finite elements. This facility is directly integrated into the main Python script.
- Structural elements lead to a global model that allows post processing in a simple manner. However, calibration of such modules to the dynamic behaviour of a real floor or wall panel is not straightforward.
- Alternatively, more complex modules can be constructed in a prior analysis step using a substructure technique. The number of degrees of freedom can in this manner be reduced to less than one percent with no practical loss of accuracy.
- When substructures are used for the various modules, post processing is cumbersome, since surface displacements and other relevant output have to be back calculated from the retained degrees of freedom of the individual substructures.
- Modal analysis is not recommended when substructures are used, since the internal degrees of freedom cannot be back calculated in ABAQUS and computation time is not reduced significantly compared to direct analysis.

Future development of the modular parametric finite-element building model will include implementation of post processing facilities to asses the sound pressure level in rooms. Furthermore, realistic coupling at joints between modules will be developed and implemented into the ABAQUS model.

Acknowledgements

The present research is part of the InterReg project “Silent Spaces”, funded by the European Union. The authors highly appreciate the financial support.

References

[1] R.J.M. Craik, "Sound Transmission Through Buildings using Statistical Energy Analysis,", Gower, 1996.

[2] EN 12354-1:2000, "Building Acoustics. Estimation of Acoustic Performance in Buildings from the Performance of Elements. Airborne Sound Insulation between Rooms", European Committee for Standardization, 3rd edition, 2000.

[3] Dassault Systèmes Simulia Corp., Providence, RI, USA, "ABAQUS Analysis. User's Manual Version 6.11", 2011.

[4] N. Kiel, B. Niu, L.V. Andersen and K. Persson, "Different modelling approaches to coupling wall and floor panels within a dynamic finite-element model of a lightweight building", in Proceedings of the Eleventh International Conference on Computational Structures Technology (CST2012), Civil-Comp Press, 2012.

[5] O. Flodén, K. Persson, A. Sjöström and N. Olhoff, "Model reduction in dynamic finite element analysis of lightweight structures", in Proceedings of the Eleventh International Conference on Computational Structures Technology (CST2012), Civil-Comp Press, 2012.

[6] B. Niu, L. Andersen, N. Kiel, O. Flodén and G. Sandberg: "Vibration transmission in a multi-storey lightweight building: A parameter study", in Proceedings of the Eleventh International Conference on Computational Structures Technology (CST2012), Civil-Comp Press, 2012.

[7] L. Holterman and Å. Petersson, "Vibrations in a seven-storey wood building" Master's Dissertation, Division of Structural Mechanics, Lund University, Sweden, ISSN 0281-6679, 2008.

[8] R.J. Guyan, "Reduction of stiffness and mass matrices", AIAA Journal, 3, 380, 1965.

[9] R.R. Craig, "Structural dynamics: An introduction to comuter methods", John Wiley & Sons Inc., New York, United States, 1981.

[10] R.R. Craig and M. Bampton, "Coupling of substructure in dynamic analysis", AIAA Journal, 6, 1313, 1968.

[11] P. Fiala, G. Degrande and F. Augusztinovicz, "Numerical modelling of ground-borne noise and vibration in buildings due to surface rail traffic", Journal of Sound and Vibration, 301, 718–738, 2007.

[12] P. Fiala, S. Gupta, G. Degrande and F. Augusztinovicz, "A numerical model for re-radiated noise in buildings from underground railways", Notes on Numerical Fluid Mechanics, 99, 115–121, 2008.