D4.2 Calculation Engine for Floor Vibration Analysis Tool

Report to:	RFCS, as deliverable from the research project: **Stronger Steels in the Built Environment (STROBE)**
Document:	RT1787
Version:	01
Date:	December 2018

SCI (the Steel Construction Institute) has been a trusted, independent source of information and engineering expertise globally for over 30 years, and is one of the leading, independent providers of technical expertise and disseminator of best practice to the steel construction sector.

We support everyone involved in steel construction; from manufacturers, consulting and design engineers, architects, product developers to industry groups.

Version	Issue	Date	Purpose	Author	Reviewer	Approved
01	01	Dec 2018	Final	AQC	NRB	NRB

Contents

Page No

1 INTRODUCTION

Steel framed structures are commonly criticised that they tend to have a problem with vibration of floors. In response to this criticism, the steel industry has sought to develop an objective measurement system for assessing the susceptibility of a floor to adverse comment from users. This work has been conducted as part of the HIVOSS[1] RFCS project, then updated and published as part of SCI P354 Design of Floors for Vibration: A New Approach[2]. Structures designed to P354 have not been reported to have vibration problems, so the assessment method can be considered a success.

While normal strength steel (NSS) floors are perceived to have issues with vibration the perception is even more acute when high strength steel (HSS) is considered. While the lower weight for a given span is economically favourable, HSS floors are intuitively unfavourable for vibration performance.

Although It may be true that floors utilising HSS may be more susceptible to vibration than equivalent solutions in NSS, previous experience and knowledge have shown that the response as a whole is not particularly sensitive to the selection of beams. It could therefore be expected that the difference between an equivalent HSS and NSS floor is either small or negligible.

There are two key issues that will be addressed in this project: (1) lack of availability of analysis tools and (2) a perception that the vibration response of HSS floors are worse than NSS floors.

To address issue (1), a floor vibration analysis (FVA) tool based on finite element methods is being developed in STROBE for calculating the dynamic performance of a given floor design in terms of a response factor (the ratio of the predicted acceleration divided by a baseline value). The user can input information about the floor grid, loading etc. via a simple interface. The beam section sizes can be either user-defined or the FVA tool can determine optimised primary and secondary beam sizes using the optimisation tool developed in Task 4.1 of WP4.

The FVA tool then calculates the critical response factor for the floor system by performing an eigenvalue analysis. The mode shapes, frequencies and modal masses for the floor are obtained and used to calculate the acceleration. In accordance with the assessment method in P354, the steady state response factor is determined from the weighted root mean square (rms) acceleration and the transient state response factor from the weighted peak acceleration. The critical response factors will be compared against regulatory limits in ISO 10137[3].

The analysis tool will be capable of analysing a range of floor systems, including beams (rolled sections in S355/460, homogenous and hybrid plate girders in HSS), floor slabs (solid concrete and various composite decking products) and loading types (normal walking and specialist loading functions relevant to gyms, hospitals and laboratories). The floor system is limited to rectangular grids with capability to define openings for staircase or building core.

The development of the FVA tool is carried out in two stages. Firstly the calculation engine has been developed in Task 4.2 and is presented in this report. The engine then will be used for comparative studies in Task 4.3 to address issue (2) mentioned above.

After that the interface and background documentation will be completed in Task 4.2 as deliverable D4.3. The FVA tool will be made freely available from various public web sites including www.steelconstruction.info.

The calculation engine of the FVA tool consists of a pre-processor to define the floor model based on user input, a finite element method solver for eigenvalue analysis and a post-processor for floor vibration analysis in accordance with P354[2]. The pre- and post-processor were developed using the Python programming language and the open source finite element software CalculiX[4] is used as the solver. Validation of the calculation engine has been carried out by comparing the predicted response factor with that calculated by the SCI floor vibration consultancy service utilising the proprietary FE package ANSYS and commercial CAD software AutoCAD (referred as SCI/ANSYS tool in the following paragraph).

The floor vibration analysis methods presented in design guide P354[2] prefers the use of finite element analysis to determine the natural frequencies, modal masses and mode shapes of the floor under consideration. Finite element modelling is useful to establish a reasonably accurate prediction of the dynamic characteristics of the floor system and will give a better prediction than that given by hand calculation methods. Usually finite element analysis demands special expertise and therefore often needs to be outsourced as it is beyond the capability of many design offices. The FVA tool will be able to make the advantages of FE analysis readily available to design engineers who want to check the vibration response of their floor designs.

This report firstly presents a brief introduction of floor vibration analysis methods in accordance with design guide P354[2] and the free finite element package CalculiX[4]. Then the deployment and validation of the calculation engine of the FVA tool is presented and its performance is evaluated.

2 FLOOR VIBRATION ANALYSIS: FINITE ELEMENT ANALYSIS APPROACH

The dynamic performance can be established through finite element modelling of the floor system. The finite element method is an approximation: it takes a continuous structure and breaks each part of the structure into a number of parts, also known as a finite number of elements. The relationships between these elements are then determined using methods for multi-degree-of-freedom discrete systems. The accuracy of the solution is primarily dependant on the number of elements that the system is broken into, but with increased accuracy comes increased complexity and hence higher computation times.

2.1 FE Implementation suggestions

The following are some recommendations for modelling techniques and assumptions for the FE model. Any improvements will lead to a greater accuracy:

- The dynamic modulus of elasticity of concrete should be taken for normal concrete equal to E_c = 38 kN/mm^2.
- Shell elements are recommended to represent the floor slab.
- All connections should be assumed to be rigid (in vibration the strains are not large enough to overcome the friction and so pinned joints may be treated as fixed).
- Column sections should be provided and pinned at their theoretical inflexion points (located at mid-height between floors for multi-storey construction).
- Continuous cladding provided around façades may be assumed to provide full vertical constraints for perimeter beams. The edge of buildings should normally modelled as pinned.
- The interface at cores should be modelled as fully restrained.
- The mass of the floor should be equivalent to the self-weight and other permanent loads, plus a proportion of the imposed loads which might be reasonably expected to be permanent

One of the most difficult properties to estimate is the level of damping that is present in the floor, due to the fact that it is strongly influenced by finishes and non-structural components. It is recommend that the values for damping give in Table 2.1.

Table 2.1 Critical damping ratios for various floor types

ζ	Floor finishes
0.5%	Fully welded steel structures, e.g. staircases
1.1%	Completely bare floors or floors where only a small amount of furnishings are present.
3.0%	Fully fitted out and furnished floors in normal use.
4.5%	A floor where the designer is confident that partitions will be appropriately located to interrupt the relevant mode(s) of vibration (i.e. the partition lines are perpendicular to the main vibrating elements of the critical mode shape).

Another important issue for properly modelling the floor structure is the offset of the beams from the slab. SCI P354 in section 6.1.2 provides two alternative ways to model the beams, with or without offset. The first option, with offset, was chosen for the modelling of the floor system for the FVA tool.

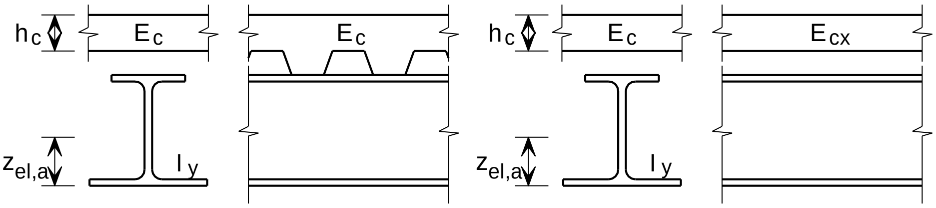


Figure 2.1 Slab and beam modelling for finite element analysis

Figure 2.2 shows the modelling option adopted. The model uses orthotropic shell elements of a depth h_c with elastic modulus E_c along the span and E_{cx} perpendicular to the beam span, with:

$$E_{cx} = E_c \frac{12 I_{c,x}}{{h_c}^3} \tag{2.1}$$

where:

$I_{c,x}$	is the second moment of area of the profiled slab per metre width in the spanning direction
h_c	is the depth of concrete above the profile
E_c	is the dynamic elastic modulus of concrete E_c = 38 kN/mm^2 as defined previously

This option also uses a beam element with the same properties and the same offset as the design. As the slab is modelled using uniform thickness of h_c, the offset, h_s, is:

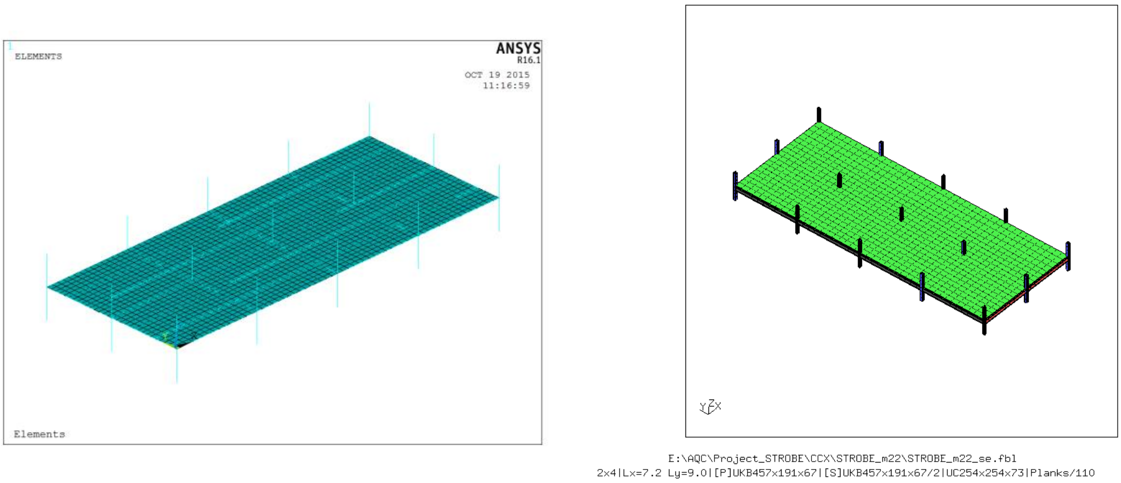
$$h_s = h + h_a - z_{el,a} - \frac{h_c}{2} \tag{2.2}$$

where:

h	is the depth of the slab (including ribs)
h_a	is the depth of the steel beam
$z_{el,a}$	is the height of the neutral axis of the steel beam
h_c	is the depth of concrete above the profile

In this report, it is assumed that $E_c = E_{cx}$ as a plain concrete floor was used in the floor model for validation and evaluation (in Chapter 5). And given that a double symmetric beam are used as well, the offset of the beam would simply be half of the total depth of concrete slab and beam.

There are no specific rules for the size of the element (mesh). In general, if the number of elements can be doubled without significantly changing the frequencies then there are sufficient elements. The size of element was determined to be 0.8 m for the FE model used in the FVA tool.



(a) ANSYS model

(b) CalculiX model (FVA tool)

Figure 2.2 Example of a meshed 2x4 floor finite element model

2.2 Modal outputs

The general purpose proprietary finite element software ANSYS is used for modelling the floors in the SCI floor vibration analysis service. For the FVA tool developed in STROBE, the open-source and free FE software CalculiX is used instead. Modal analysis is performed to determine the modal properties, such as frequencies, modal masses and mode shape amplitudes.

The mode shape amplitudes can be ‘unity normalised’ (against peak value of the amplitude) or ‘mass normalised’ (against the modal mass of each mode) in ANSYS. CalculiX only produces mass normalised mode shape amplitudes. It will be shown later that this does not affect the calculation of the floor response.

2.3 Steady-state response

For the steady-state case, the first four harmonic components from a person engaged in walking activity are considered to represent the forcing function applied to the floor (other specialist forcing functions will be included and used in Task 4.3). The appropriate design values of the Fourier coefficients for these harmonic components are obtained from Table 2.2:

Table 2.2 Design Fourier coefficients for walking activities

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

Square root sum of squares (SRSS) is the method used to determine the maximum rms acceleration response $a_{w,rms,e,r}$ at a point on the floor area subject to an excitation force F_h (often at the same point), and it gives the same results as a full time history.

$$a_{w,rms,e,r} = \frac{1}{\sqrt{2}} \sqrt{\sum_{h=1}^{H} \left(\sum_{n=1}^{N} \left(\mu_{e,n} \mu_{r,n} \frac{F_h}{M_n} D_{n,h} W_h \right) \right)^2} \tag{2.3}$$

where:

$\mu_{e,n}$	is the mode shape amplitude, from the unity or mass normalised FE output, at the point on the floor where the excitation force F_h is applied
$\mu_{r,n}$	is the mode shape amplitude, from the unity or mass normalised FE output, at the point where the response is to be calculated
F_h	is the excitation force for the hth harmonic for the Fourier coefficients for walking, where $F_h = \alpha_h Q$ (N)
M_n	is the modal mass of mode n (kg), when the mode shape amplitude is mass normalised M_n = 1 kg
$D_{n,h}$	is the dynamic magnification factor for acceleration
W_h	is the appropriate code-defined weighting factor for human perception

The dynamic magnification factor for acceleration, which is the ratio of the peak amplitude to the static amplitude, is given by the following:

$$D_{n,h} = \frac{\beta^2}{\sqrt{(1-\beta_n^2)^2 + (2\zeta\beta_n)^2}} \tag{2.4}$$

where:

h	is the number of the hth harmonic
β_n	is the frequency ratio (taken as f_p/f_n)
ζ	is the damping ratio

f_p is the frequency corresponding to the first harmonic of the activity (f_p = 2 Hz is used in the current analysis)

f_n is the frequency of the mode under consideration

The aim of vibration analysis is to remove or reduce discomfort. The perception of vibration depends on the frequency. This is because the human body's sensitivity to a given amplitude of vibration changes with the frequency of the vibration, as the body has a variable range of maximum sensitivity. The variation of sensitivity can be taken into account either by attenuating the calculated response (for frequencies where perception is less sensitive) or by enhancing the base value. The degree to which acceleration is attenuated or enhanced is referred to as "frequency weighting".

The most typical case (curve W_b), for residential and office building floors, is presented in Figure 2.3 and, for steady state response, the frequency of the harmonic under consideration hf_p.

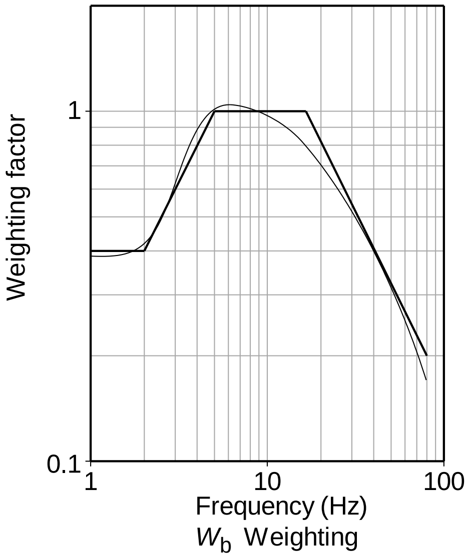


Figure 2.3 W_b frequency weighting curve (BS 6841[5]) for z-axis vibration (room types: residential, office, wards, general laboratories, consultation rooms)

The curve presented in Figure 2.3 can also be expressed as a combination of straight lines, as given by below:

$$\left.\begin{array}{ll} W = 0.4 & \text{for } 1\text{ Hz} < f < 2\text{ Hz} \\ W = \frac{f}{5} & \text{for } 2\text{ Hz} \le f < 5\text{ Hz} \\ W = 1.0 & \text{for } 5\text{ Hz} \le f \le 16\text{ Hz} \\ W = \frac{16}{f} & \text{for } f > 16\text{ Hz} \end{array}\right\} \quad (2.5)$$

2.4 Transient response

For the transient response, an effective impulse is calculated for each footfall of a person engaged in a walking activity; it relates to heel impacts. This force is given by the following formula.

$$F_I = 60\frac{f_p^{1.43}}{f_n^{1.3}}\frac{Q}{700} \tag{2.6}$$

where

f_p is the pace frequency

f_n is the frequency of the model under consideration and

Q is the static force exerted by an "average person" (normally taken as 76 kg × 9.81 m/s^2 = 746 N)

The peak acceleration can be calculated as:

$$a_{w,peak,e,r,n} = 2\pi f_n \sqrt{1-\zeta^2}\ \mu_{e,n}\,\mu_{r,n}\,\frac{F_I}{M_n}W_n \tag{2.7}$$

where

$\mu_{e,n}$ is the mode shape amplitude, from the unity or mass normalised FE output, at the point on the floor where the impulse force F_I is applied

$\mu_{r,n}$ is the mode shape amplitude, from the unity or mass normalised FE output, at the point where the response is to be calculated

F_I is the excitation force given in Equation (2.6)

M_n is the modal mass of mode n (equal to 1 if the mode shapes are mass normalised) (kg)

W_n is the appropriate code-defined weighting factor for human perception determined using the weighting curve presented in Figure 2.3, for residential and office building, and the frequency of the mode under consideration f_n

This force is applied dynamically with a small time step from t = 0 to t = T (= 1/f_p) and a time-history analysis is implemented. The acceleration–time function is given by summing the contribution of each mode n as shown in the following equation.

$$a_{w,e,r}(t) = \sum_{n=1}^{N} a_{w,e,r,n}(t) = a_{w,peak,e,r,n}\sin(2\pi f_n\sqrt{1-\zeta^2}t)\cdot e^{-\zeta 2\pi f_n t} \tag{2.8}$$

where:

t is the time in seconds from the application of the impulse

The root-mean-square (rms) acceleration for transient response needs to be found from the peak acceleration above using the formula shown below:

$$a_{rms} = \sqrt{\frac{1}{T}\int_0^T a(t)^2 dt} \approx \sqrt{\frac{1}{S}\sum_{s=1}^{S} a_s(t)^2} \tag{2.9}$$

For an average walking frequency of f_p = 2 Hz the period is T = 1 / f_p = 0.5 sec; if S = 1000 steps, the time difference between every step is *Δt* = 0.5 / 1000 = 0.0005 sec. A small time step is required to give accurate results through approximation.

According to Section 6.3.3 of SCI P354[2], it is recommended that all modes with natural frequencies up to twice the frequency of the first mode should be taken into account. All modes with a frequency higher than that will have an insignificant impact on the result due to their low weighting factor and hence, can be ignored. However, the first 50 modes were considered in the present study.

2.5 Response factor

The "response factor" of a floor is the ratio between the calculated weighted rms acceleration, from either the steady-state or transient methods, and the "base value" given in BS 6472[6] as 0.005 m/s^2 for vertical vibration in the frequency range of interest for the analysis. The response factor is therefore given by:

$$R = \frac{a_{w,rms}}{0.005} \qquad (2.10)$$

2.6 Acceptance criteria

The vibration response is considered to be satisfactory for continuous vibration when the calculated response does not exceed a limiting value appropriate for the environment (which is expressed in BS 6472[6] and ISO 10137[7] as a multiplying factor). Table 2.3 provides multiplying factors to the base curves for continuous vibrations, which correspond to a "low probability of adverse comment"

Table 2.3 Multiplying factors specified in BS 6472 for "low probability of adverse comment"

Place	Time	Multiplying factor for exposure to continuous vibration 16 h day 8 h night	Impulsive vibration excitation with up to 3 occurrences
Residential	Day	2 to 4	60 to 90
	Night	1.4	20
Office	Day	4	128
	Night	4	128
Workshops	Day	8	128
	Night	8	128

In practice, these multiplying factors are used as limiting values of the calculated response factors,

In 1989, SCI proposed a series of multiplying factors that, in some environment, are larger than those presented in Table 2.3. These supplementary values are presented in Table 2.4, and may be used for design.

Table 2.4 Recommended multiplying factors based on single person excitation

Place	Multiplying factor for exposure to continuous vibration 16 h day 8 h night
Office	8
Shopping mall	4
Dealing floor	4
Stairs – Light use (e.g. offices)	32
Stairs – Heavy use (e.g. public buildings, stadia)	24

3 OPEN-SOURCE FE PACKAGE CalculiX

CalculiX[4] is an open-source and free package designed to solve field problems using finite element methods. It has a pre- and post-processor to build, mesh, solve and post-process finite element models. The pre- and post-processor is called CalculiX Graphix[8] (CGX). The solver (CalculiX CrunchiX[9], CCX) has both linear and non-linear capabilities and is capable of solving static, dynamic and thermal problems.

As Calculix uses the same input file format of commercial FE package ABAQUS, a conversion tool has been developed in the project to convert CalculiX into an ABAQUS model so that results can be checked using ABAQUS for any floor model created by the FVA tool.

3.1 Beam and shell elements

FE models are created using CalculiX in accordance with the recommendation given in Section 2.1. Eight-node shell element (S8R) with reduced integration are used to represent the floor slab. In CalculiX, quadratic shell elements are automatically expanded into 20-node brick element. The expansion is shown in Figure 3.1. For each shell node three new nodes are generated according to the scheme shown on the right hand side of Figure 3.1. With these nodes a new 20-node brick element is generated, i.e. a C3D20R element for a S8R element.

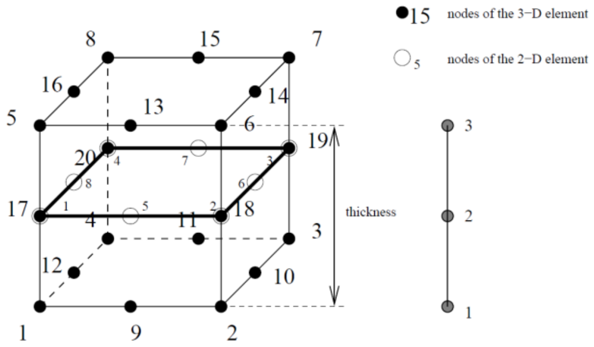


Figure 3.1 Expansion of an 8-node shell element into a 3D brick element

Three-node 3D beam element B32 is used to model the steel beam and columns in the floor model. Similar to shell elements, the B32 beam element is expanded into a C3D20 brick element according to Figure 3.2.

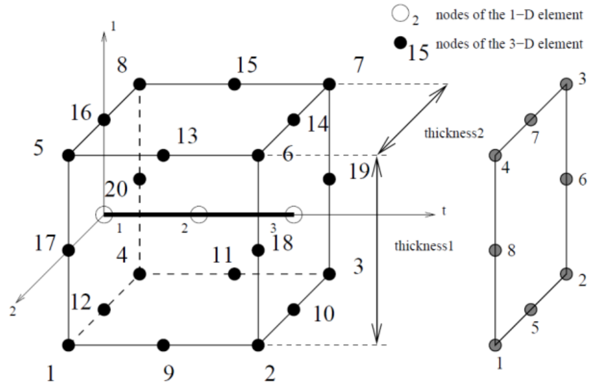


Figure 3.2 Expansion of a beam element

For each node of the beam element 8 new nodes are generated according to the scheme shown on the right of Figure 3.2. These new nodes are used in the definition of the brick element, and their position is defined by the local directions (1 and 2 as shown in Figure 3.2) together with the thickness and offset in these directions.

The section of the beam must be specified. Unfortunately, I-section beams and columns used in the floor system cannot be specified directly in CalculiX. Instead, rectangular sections with an offset is used. The offset can take any real value and enables construction of a beam of nearly arbitrary cross section and the definition of composite beams. In practice, the top flange, web and bottom flange of a beam or column are modelled using three individual beam elements of rectangular cross section with appropriate offsets from the neutral plane of the floor slab (shell element). These three beam elements share the same nodes.

Beam and shell elements are in fact implemented using 3D brick elements (C3D20) in CalculiX, whereas in commercial FE packages they are usually formulated using Euler-Bernoulli beam theory and Kirchhoff-Love plate theory (e.g. used in SCI/ANSYS tool). A few simple verification examples are presented in this chapter to show that using brick elements for beam, column and floor slab does not impinge on the accuracy of the results by.

3.2 Frequency analysis

CalculiX can perform eigenvalue extraction to calculate the natural frequencies and the corresponding mode shapes of the floor system. The model shape amplitudes are mass normalised in CalculiX so that the modal mass M_n is to be taken as 1 kg in the response analysis as previously discussed.

3.3 Simple verification examples

A frequency analysis was performed for simple beam and floor components to verify the accuracy of using brick elements to model beams and floor slabs.

3.3.1 Simply supported beam

A simply supported steel beam with length of 4.8 m is considered here. The beam section is UKB 457 × 191 × 67. The fundamental frequency (the first mode) of a simply beam can be calculated as shown in Equation (3.1) below.

$$f_1 = \frac{1.57}{L^2}\sqrt{\frac{EI}{\rho A}} \tag{3.1}$$

where:

E	is the Young's modulus, $E = 210\ N/mm^2$
ρA	is the mass per unite length of the beam in consideration, $\rho A = 67.1\ kg/m$
I	is the second moment of area of the beam in consideration, $I = 29400\ cm^4$

According to theory, the fundamental frequency ($f_{1,\ ana}$) is 65.36 Hz. The frequency of the first model predicted by CalculiX ($f_{1,\ FE}$) is 73.59 Hz and the mode shape is presented in Figure 3.3.

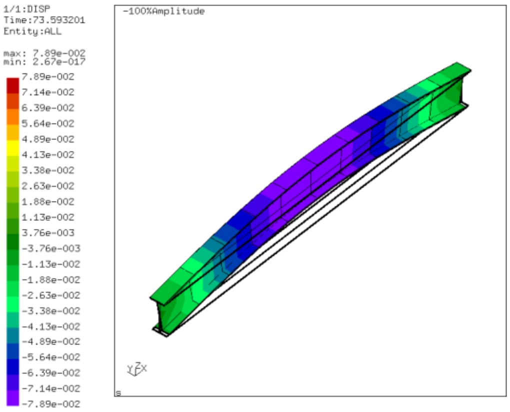


Figure 3.3 Fundamental mode shape of a simply supported beam

The FE value compares reasonably well (12.5% discrepancy) considering that the beam is modelled using brick elements with moderately coarse element size and only three beam elements are used for modelling the cross-section.

3.3.2 Simply supported floor slab

A plain concrete floor simply supported around four edges (SSSS) is considered here. The slab is 7.2 m wide in the X-direction (L_x = 7.2 m) and 9.0 m long in the Y-direction (L_y = 9.0 m). The thickness *h* of the plate is assumed to be 110 mm. The density of concrete is taken as 2400 kg/m^3. For an isotropic slab, the fundamental frequency can be calculated as shown below:

$$f_1 = 1.57\left(\frac{1}{a^2}+\frac{1}{b^2}\right)\sqrt{\frac{D}{\rho h}}\,, D = \frac{Eh^3}{12(1-v^2)} \tag{3.2}$$

where

v	is the Poisson's ration, $v = 0.2$
a, b	is the dimension of the floor, $a = 7.2\ m; b = 9.0\ m$
E	is the Young' modulus of concrete, $E = 38\ N/mm^2$
ρ	is the density of concrete, $\rho = 2400\ kg/m^3$

Knowing the material and dimensional properties of the plate, the theoretical frequency of the first mode of the plate can be estimated to be $f_{1,\text{ ana}}$ = 6.41 Hz, which is effectively the same as the frequency predicted by CalculiX for the first mode ($f_{1,\text{ FE}}$ = 6.40 Hz). The mode shape of the floor slab is shown in Figure 3.4 below.

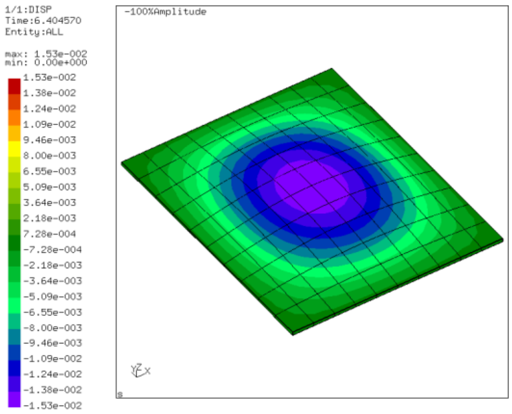


Figure 3.4 Simply supported floor slab

3.3.3 Orthotropic floor slab

Composite floor decks are not isotropic. To accurately estimate the stiffness of a composite floor using profiled steel decking, orthotropic properties of the composite slab have to be used. It is assumed here and in all future work that the properties of shell elements are assigned with the direction of the ribs parallel to the X-axis (i.e. the elastic modulus in the X-direction is greater than in the Y-direction, $E_x > E_y = E_c$ and E_c is the dynamic elastic modulus of concrete $E_c = 38\ N/mm^2$).

The fundamental frequency ($m = 1, n = 1$) of an orthotropic rectangular plate simply supported around four edges can be calculated using Equation (3.3) below.

$$
\begin{aligned}
f_{mn} &= \frac{1.57}{a^2\sqrt{\rho h}}\sqrt{D_x m^4 + 2D_{xy}m^2n^2\left(\frac{a}{b}\right)^2 + D_y n^4\left(\frac{a}{b}\right)^4} \\
D_x &= \frac{E_x h^3}{12(1 - v_x v_y)} \\
D_y &= \frac{E_y h^3}{12(1 - v_x v_y)} \\
D_k &= \frac{Gh^3}{12}
\end{aligned}
\tag{3.3}
$$

where:

m, n	Model number, for the fundamental model, $m = 1, n = 1$
a	Dimension of the plate in x – direction, $L_x = 7.2\ m$
b	Dimension of the plate in y – direction, $L_y = 9.0\ m$
ρ	Density of the concrete, $\rho = 2400\ kg/m^3$

h	Floor slab thickness, $h = 0.11\,m$ assumed
D_x, D_y, D_{xy}, D_k	Elastic constants
E_x, E_y	Elastic modulus of concrete floor in x and y direction
ν_x, ν_y	Poisson's ratio in x and y direction
G	Shear modulus of concrete floor

The floor model studied in the previous section is used to model the orthotropic plate here, however the material properties have to be modified to account for orthotropic material behaviour. The orthotropic elastic engineering constants are used in CalculiX to define material properties as shown below:

*Elastic, TYPE=ENGINEERING CONSTANTS

E_1, E_2, E_3, ν_{12}, ν_{13}, ν_{23}, G_{12}, G_{13},

G_{23}, T

The material properties used in the analytical equation and FE models in CalculiX defined as following:

$$E_1 = E_x = E_{cx} = E_c \frac{12 I_{c,x}}{h_c^3} \tag{3.4}$$

where:

$I_{c,x}$	is the second moment of area of the profiled composite slab per meter width in the spanning direction
h_c	is the depth of concrete above the steel decking

Young's modulus in the direction 2 and 3 (Y and Z) are defined as:

$$E_2 = E_3 = E_y = E_c \tag{3.5}$$

Poisson's ratio in every direction are defined as:

$$\nu_{12} = \nu_{13} = \nu_{23} = \nu_x = \nu_y = \nu = 0.2 \tag{3.6}$$

Shear modulus in every direction are defined as:

$$G_{12} = G_{13} = G = \frac{E_{cx}}{2(1+\upsilon)};\ G_{23} = \frac{E_c}{2(1+\upsilon)} \tag{3.7}$$

The temperature T is assumed to be 295 Kelvin. Effective second moment of area in X-direction $I_{c,x}$ and concrete depth above decking h_c are to be determined based on the type of the profiled steel decking used in the design. For the simplicity of comparison herein it is assumed that $\frac{12 \mathrm{I}_{\mathrm{c,x}}}{\mathrm{h}_{\mathrm{c}}^3} = 4.822$ and $h_c = 110\,mm$. Thus for the orthotropic floor, in the X-direction $E_1 = E_x = 4.822 \times 38 = 183\,N/mm^2$, and in Y and Z-direction $E_2 =$

$E_3 = E_y = 38\,N/mm^2$ and $G_{12} = G_{13} = G = 76.4\,N/mm^2$, and finally $G_{23} = 15.83\,N/mm^2$.

The fundamental frequencies of the orthotropic plate are compared in Table 3.1 below. It can be seen that the CalculiX FE model is in good agreement with the theory (6% lower). To further validate the implementation of the orthotropic elastic material model, the elastic modulus in the X and Y direction are made equal and the frequencies are identical to the isotropic plate considered previously.

Table 3.1 Fundamental frequency of orthotropic concrete floor slab

[Hz]	E_x = 183 GPa E_y = 38 GPa	E_x = 38 GPa E_y = 38 GPa	E = 38 GPa (Isotropic)
$f_{11,ana}$	13.19	6.41	6.41
$f_{11,FE}$	12.43	6.40	6.40

The frequency of structural components such as beams and plates were calculated using the analytical method and CalculiX FE model. Fundamental frequencies were compared and orthotropic material properties were used for composite concrete floor slab. The results showed good agreement between the CalculiX FE model and the theory of free vibration of simply supported beams and plates.

4 DEVELOPMENT OF CALCULATION ENGINE

Although CalculiX is capable of performing modal analysis accurately, it does not have a unified analysis environment which allows users to carry out floor vibration analysis easily. Specialist knowledge of using CalculiX is required for extraction of frequencies and mode shapes, and then the users will have to use SCI P354[2] to check the vibration performance of a floor system. The FVA tool is developed based on the idea that users will be able to do everything within a single user interface, from defining the floor layout to calculating response factors for the floor system. The development of the FVA tool is divided into three parts:

- Part 1 is the web interface, where the user inputs a range of parameters to define the floor system and vibration response analysis. It produces a group of files (mainly csv files) to store the information of the floor system, loading and input parameters for vibration analysis, such as damping ratio. This is deliverable D4.2b.
- Part 2 is the pre-processing unit of the FVA tool. It uses data files produced by web interface (users) to create and solve an FE model. This unit consists of a Python library developed by SCI and the programme CGX. The Python library is used to setup the FE model, calls CGX to generate the mesh and then completes the input file. Then it submits the input to CCX for modal analysis.
- Part 3 is the post-processing unit of the FVA tool. This unit also consists of a Python library developed by SCI and the programme CGX. The Python library is used to read modal analysis results, carry out vibration analysis in accordance with SCI P354 and then writes the derived results (e.g. response factors) back to the FE results file so that CGX can be called to plot the distribution of steady state and transient response factors over the entire floor area. Pictures of mode shapes are also produced using the Python library by calling CGX. Finally the derived results such as response factors are returned back to the interface for users.

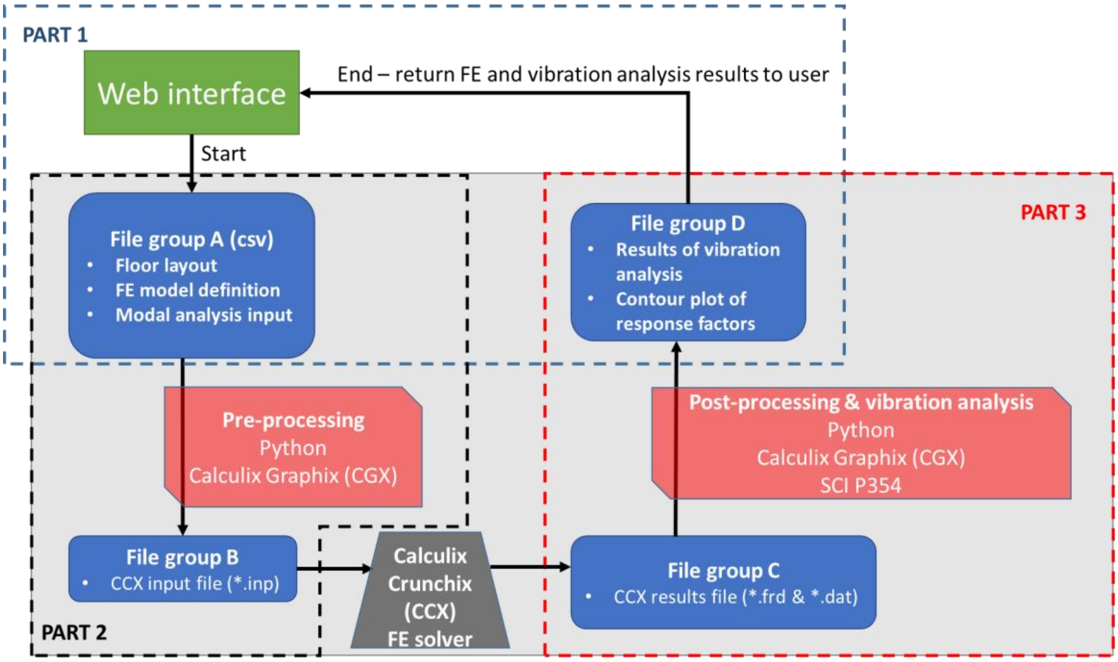


Figure 4.1 Structure and data flow of the floor vibration analysis tool and calculation engine

The structure and data flow of the FVA tool are presented in Figure 4.1. The calculation engine comprises part 2 and 3 of the FVA tool. It has all functions to perform modal analysis using FE methods and carry out vibration response analysis. The calculation engine will be used in a comparative study in Task 4.3 (Deliverable 4.3). After that the web interface (part 1) will be completed as Deliverable D4.2b.

4.1 Input parameters and interface

The complete FVA tool will be able to analyse any rectangular floor systems with openings. The input parameters of the calculation engine presented here and for the comparative study are listed as below:

- Damping ratio (%)
- Imposed floor load (kN/m^2)
- Secondary beam (mid-span or third points)
- Span of primary beam (m)
- Span of secondary beam (m)
- Steel decking profile (including plain concrete slab)
- Bay arrangement ($n_x \times n_y$)
- Slab depth (m)
- Primary beam section
- Secondary beam section
- Column section

A database of sections for hot rolled normal strength steel beams and columns are included in the calculation engine. In addition, custom sections are available for high strength steel beams and columns. The calculation engine also has an interface with the optimisation tool developed in Task 4.1 so that optimised beam sections can be used as well.

4.2 Modal analysis and outputs

The output of the modal analysis used for vibration response analysis are modal frequencies and mass normalised mode shape amplitudes. The mode shapes will be plotted as part of the post-processing procedure.

Although it is recommended that all modes with natural frequencies up to twice the frequency of the first mode should be considered in the response analysis, the first 50 modes are considered in the current analysis regardless of their frequencies.

4.3 Floor vibration response analysis

The vibration response analysis is carried out in accordance to SCI P354[2]. A brief introduction of the analysis methods has been presented in Chapter 2.

4.4 Conversion to ABAQUS model

One of the special advantages of CalculiX is that it makes use of the ABAQUS input format. A Python module has been developed in the post-processing unit to convert the CalculiX model to an equivalent ABAQUS model, so that the models can be checked by ABAQUS for debugging and validation during the development phase.

5 EVALUATION AND VALIDATION OF CALCULATION ENGINE

Validation of the calculation engine is presented in this chapter. The distribution of response factors across the floor area predicted by the calculation engine are compared with that calculated by the SCI vibration analysis tool (using the commercial FE package ANSYS). In addition to the response factor, mode shape amplitudes and modal frequencies are also compared to evaluate the performance of CalculiX FE models.

Two floor arrangements were used in the validation study: a 1 by 1 floor system and a 2 by 4 floor system. The secondary beam is located at mid-span in both cases.

5.1 Case study of a 1 × 1 floor model

The first case is a floor system with a 1 by 1 bay arrangement. A schematic of the floor system is shown in Figure 5.1. The primary and secondary beams have the same section for simplicity. The floor slab is plain concrete with 110 mm thickness. The floor is assumed to be unloaded.

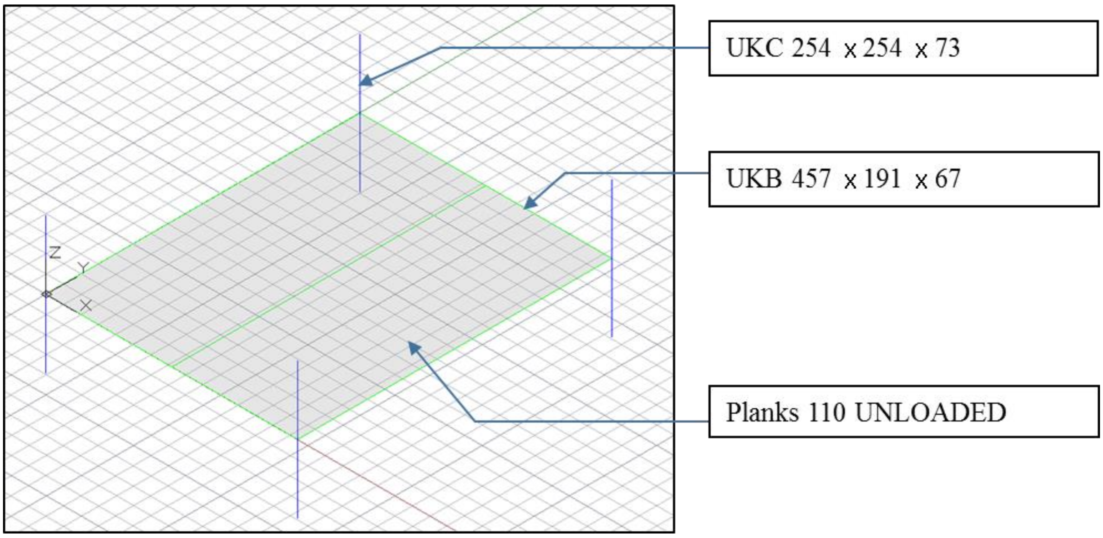
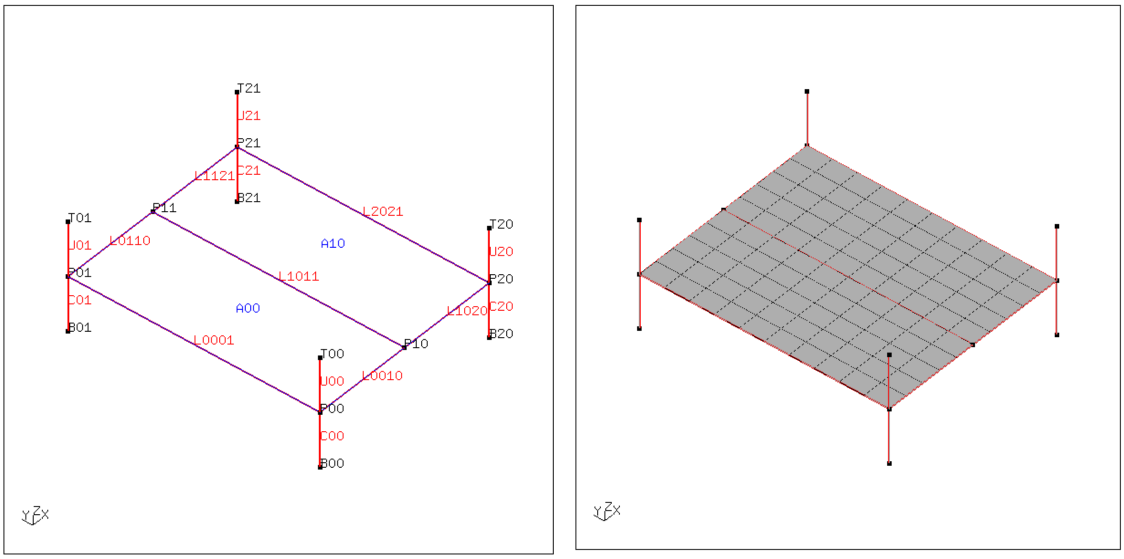


Figure 5.1 A simple 1 bay × 1 bay building floor model

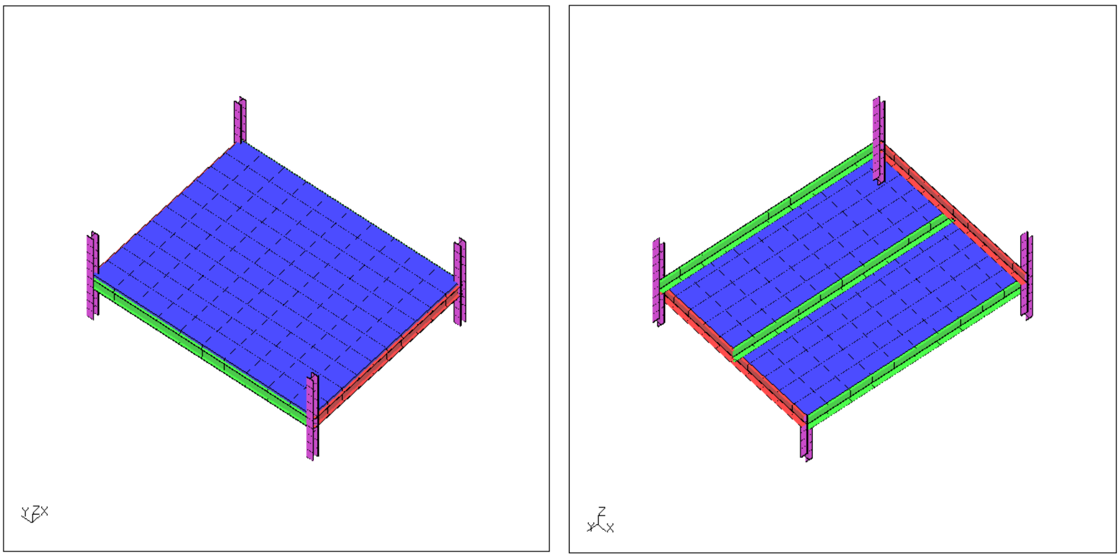
The FE model was created using CalculiX in accordance with recommendations listed in Section 2.1. The CalculiX model of the 1 by 1 floor system is shown in Figure 5.1. Figure 5.1(a) presents geometric entities (i.e. point, line and area) describing the whole model. Entities are systematic numbered for reference in the pre- and post-processing stage. Figure 5.1(b) presents the meshed model (element size of 0.8 m).



(a) Floor model in CalculiX (b) Model mesh

Figure 5.2 Floor FE model in CalculiX (pre-processing)

As discussed in Section 3.1, shell and beam elements are always expanded into brick elements. Figure 5.3 shows the “expanded” finite element model of the floor system, in which the beam and column sections can be clearly identified.



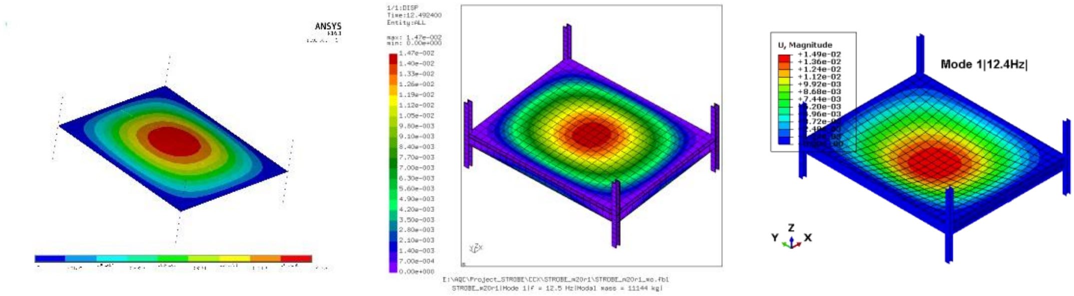
(a) Top ISO view (b) Bottom ISO view

Figure 5.3 Floor FE model in CalculiX (post-processing, expanded into brick elements)

The brick element can only be viewed after the model is solved by CCX (i.e. during the post-processing stage).

5.1.1 Mode shape amplitude, frequency and modal mass

Frequencies, mode shape amplitudes and modal mass are required for floor vibration analysis according to P354[2]. The mode shape and frequency of the fundamental (first) mode predicted by ANSYS, CalculiX (referred as CCX) and ABAQUS are compared in Figure 5.4 below. The ABAQUS model was converted from the input file for CalculiX.



(a) ANSYS, $f_1 = 11.7$ Hz (b) CCX, $f_1 = 12.5$ Hz (c) ABAQUS, $f_1 = 12.4$

Figure 5.4 Mode shape of fundamental mode

The shape of the first mode predicted by the three FE package are the same. The frequency of CCX model is in good agreement with ABAQUS but slightly higher than ANSYS (6.8%).

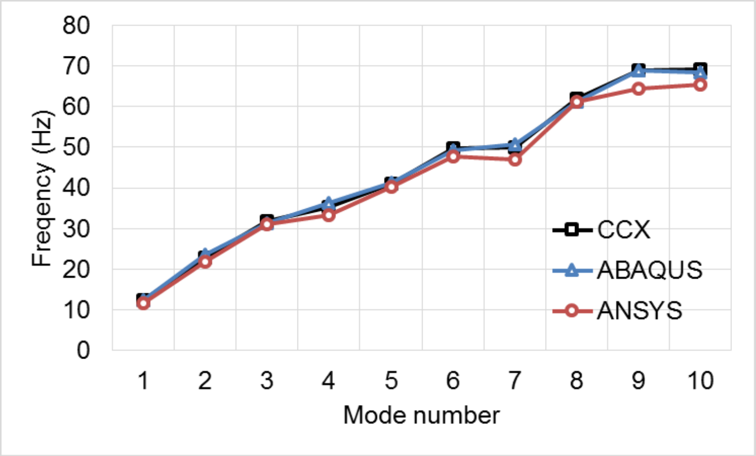


Figure 5.5 Comparison of frequencies of first 10 modes

Frequencies of the first 10 modes are compared in Figure 5.5. It can be observed that the CCX model is in better agreement with ABAQUS than ANSYS, the latter produced slightly lower frequencies for all ten modes. The shape of all first 10 modes are presented and compared in Appendix A.

Mode shape amplitudes are used in Equation (2.3) and (2.7) to calculate steady-state and transient acceleration of the floor. As discussed previously, they can be either unit normalised (used in SCI/ANSYS analysis tool) or mass normalised (in CalculiX). In order to provide an insight of the performance of these three FE models, the maximum mass normalised mode shape amplitude of the first 10 modes are compared in Figure 5.6. The dotted lines are the trend lines of the data presented in the figure.

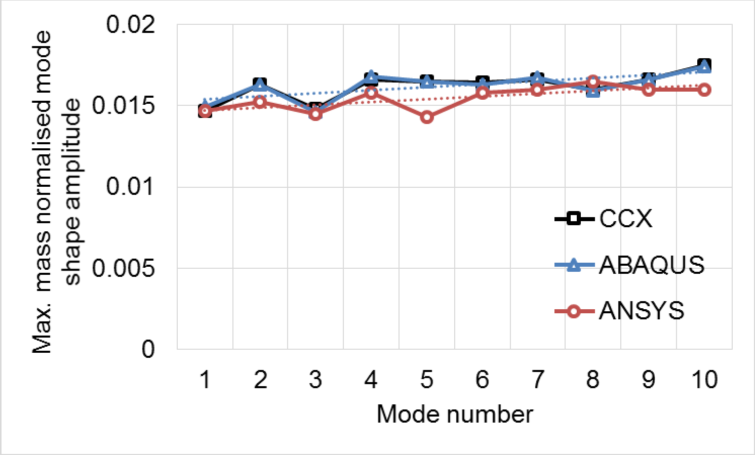


Figure 5.6 **Comparison of maximum mass normalised mode shape amplitude for the first 10 modes**

It can be seen that the CCX model is in good agreement with ABAQUS, whereas ANSYS produces very similar but slightly smaller amplitudes. It should be noted that ABAQUS and ANSYS are capable of producing either unity or mass normalised mode shape amplitudes.

When the mode shape amplitudes are unity normalised, the modal mass for each mode are made available by ANSYS and ABAQUS. Modal masses are therefore compared in Figure 5.7 even though CCX does not produce modal mass as results. The discrepancy between ABAQUS and ANSYS is certainly noticeable, but they share the same trends that higher modes poses smaller modal mass thus have higher influence on the response of the floor. However, it should be noted that the more dominating factor is the frequency. Overall, as the frequency of the higher modes increase, their influence on the response factor decrease. .

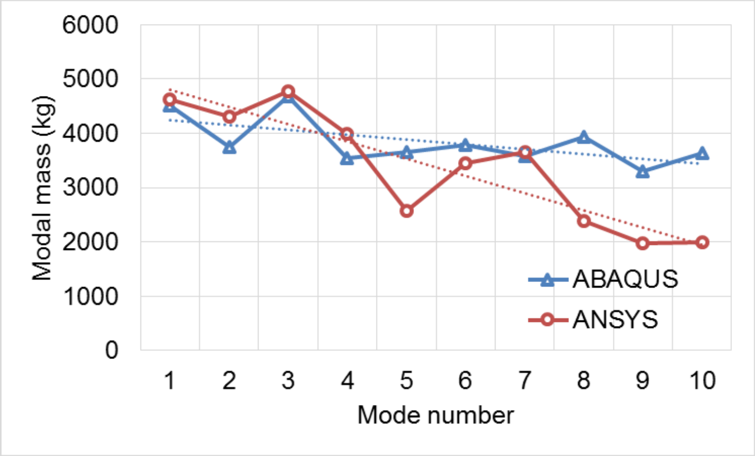


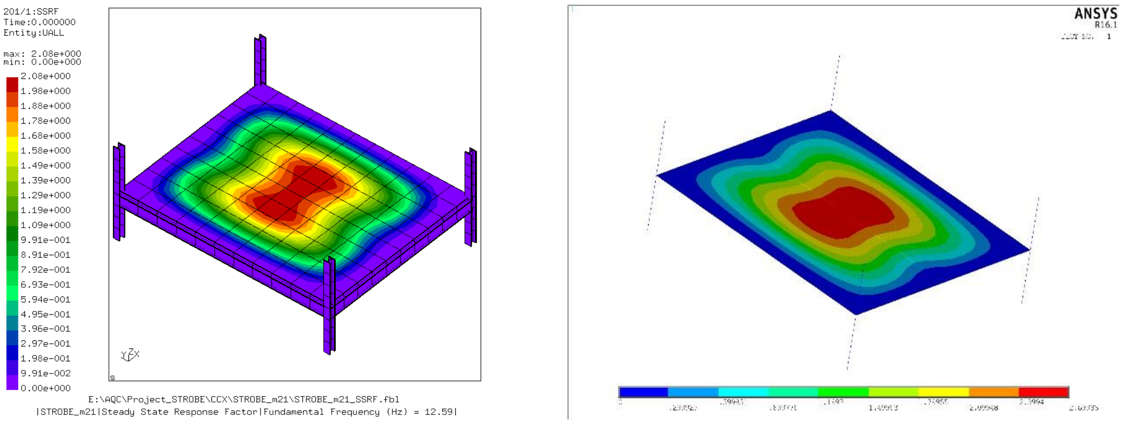
Figure 5.7 **Comparison of modal mass for first 10 modes between ABAQUS and ANSYS models**

In general, CalculiX FE results are in good agreement with ABAQUS and ANSYS.

5.1.2 Steady-state response

Vibration response analysis was carried out in accordance with P354 using the Python post-processing library developed as part of the FVA tool. The analysis methods are presented in Chapter 2. A damping ratio of 1.1 % is used for steady-state and transient response analysis.

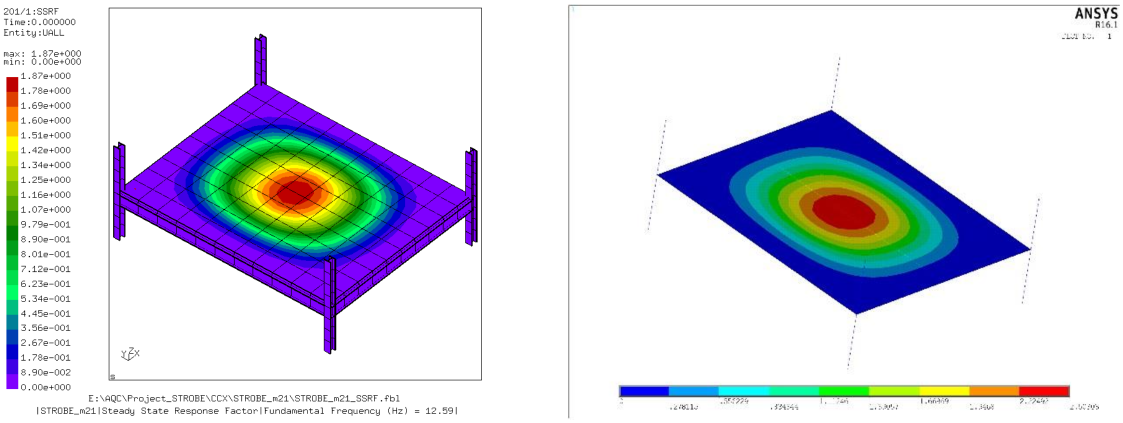
The steady-state response factor predicted by the calculation engine is compared with SCI/ANSYS tool in Figure 5.8. The first 50 modes were taken into account in the analysis. It can be seen that the distribution of the response factors are in good agreement between the two analysis tools. However, the maximum response factor predicted by the calculation engine is 2.08 which is smaller than that from SCI/ANSYS tool (2.69).



(a) Calculation engine, $R_{ss,\ max}$ = 2.08 (b) SCI/ANSYS tool, $R_{ss,max}$ = 2.69

Figure 5.8 Comparison of steady state response factor

Investigation into the discrepancy was carried out. At first, the response factors were re-calculated using only the first mode, as shown in Figure 5.9. This illustrates the weight of contribution of the 1st mode and simplifies the analysis algorithm for investigation. The results show that the maximum response factor calculated by the calculation engine is still smaller than SCI/ANSYS tool.



(a) Calculation engine, $R_{ss,\ max}$ = 1.87 (b) SCI/ANSYS tool, $R_{ss,max}$ = 2.50

Figure 5.9 Comparison of steady state response factor using only 1st mode

Subsequently, the frequency of the 1st mode predicted by ANSYS ($f_{1,\ ANSYS}$ = 11.7 Hz) was used in the Python post-processing library for response analysis, replacing the frequency obtained by CalculiX ($f_{1,CCX}$ = 12.5 Hz). The resulting response factor (2.53), as shown in Figure 5.10, was improved significantly and is almost identical to the SCI/ANSYS tool (2.50).

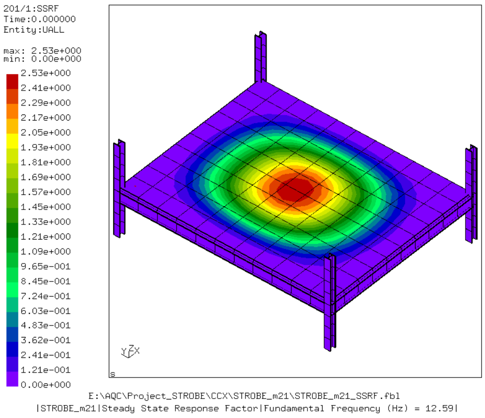
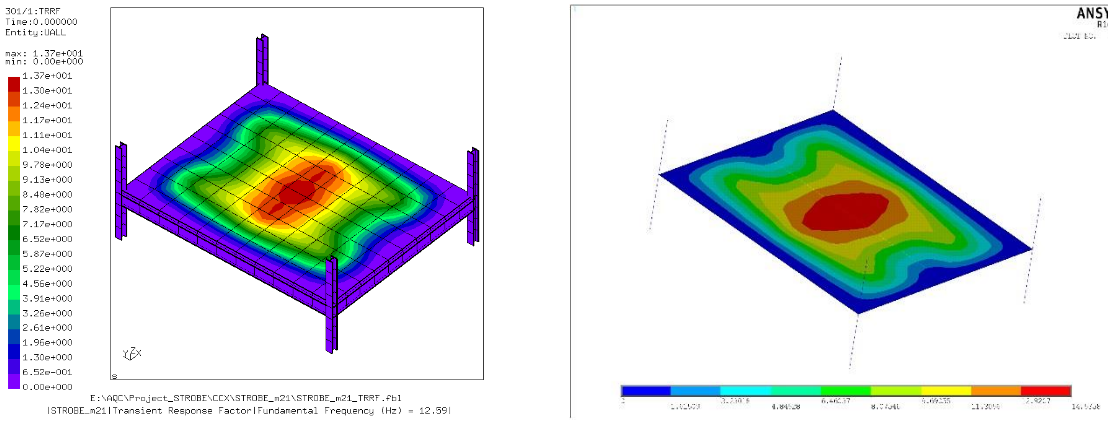


Figure 5.10 Steady state response factor from calculation engine using 1st mode and adjusted frequency ($R_{ss,\ max}$ = 2.53)

5.1.3 Transient response

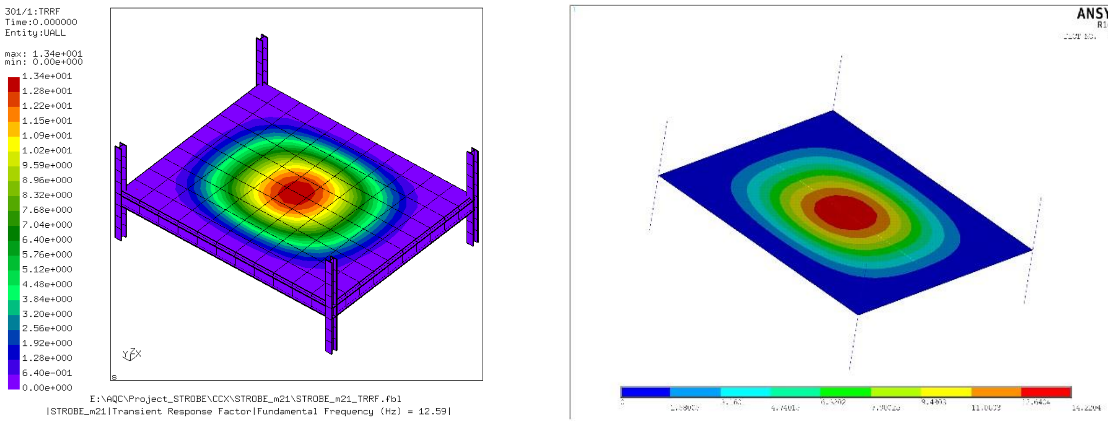
Transient response results are compared in Figure 5.11. Similar to the comparison of steady-state analysis results, the maximum transient response factor predicted by the calculation engine is smaller than the SCI/ANSYS tool results as well.



(a) Calculation engine, $R_{t,\,max}$ = 13.7 (b) SCI/ANSYS tool, $R_{t,max}$ = 14.5

Figure 5.11 Comparison of transient response factor

Recalculation of the transient response factors using only the first mode shows a similar outcome, as presented in Figure 5.12.



(a) Calculation engine, $R_{t,\,max}$ = 13.4 (b) SCI/ANSYS tool, $R_{t,max}$ = 14.2

Figure 5.12 Comparison of transient response factor using only 1st mode

Using the frequency predicted by ANSYS instead of CCX resulted in an almost identical response factor (14.4) to that produced by the SCI/ANSYS tool (14.2).

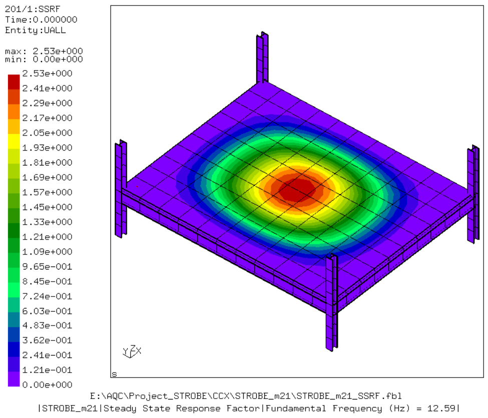


Figure 5.13 Transient response factor from calculation engine using 1st mode and adjusted frequency ($R_{t,\ max}$ = 14.4)

It should be noted that the difference of transient response factors between the calculation engine and SCI/ANSYS tool are much smaller than that of steady-state response. This is mainly attributed to the dynamic magnification factor (as presented by Equation (2.3)) used for evaluation of the steady-state accelerations. It significantly increases the response acceleration (at all four harmonics shown in Table 2.2) when the natural frequency (mainly 1st mode) of the floor is closer to the frequency of excitation source (i.e. $f_p = 2$ Hz).

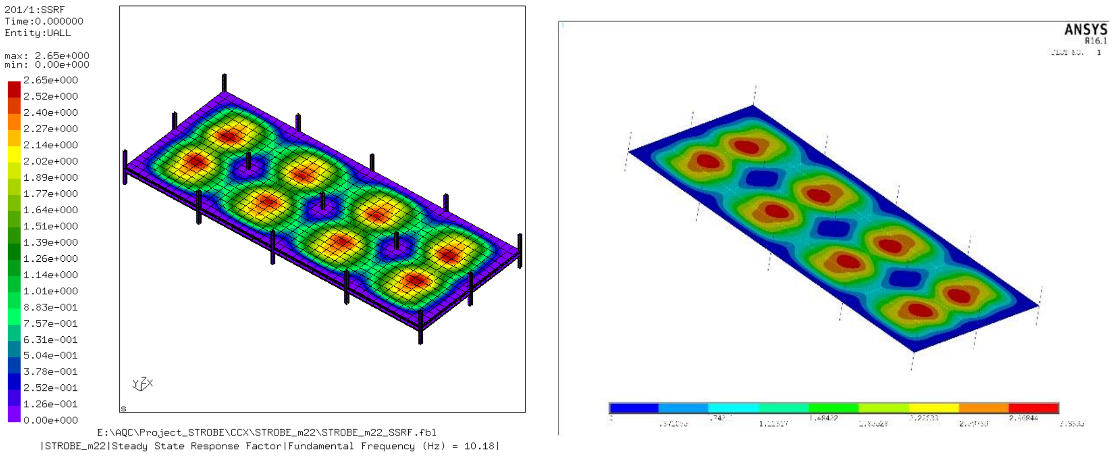
The evaluation and investigation above showed that:

- The response analysis method (in accordance with P354) is implemented correctly in the Python post-processing library
- Modal frequencies of the floor system have a significant influence on the vibration response

5.2 Case study of a 2 × 4 floor model

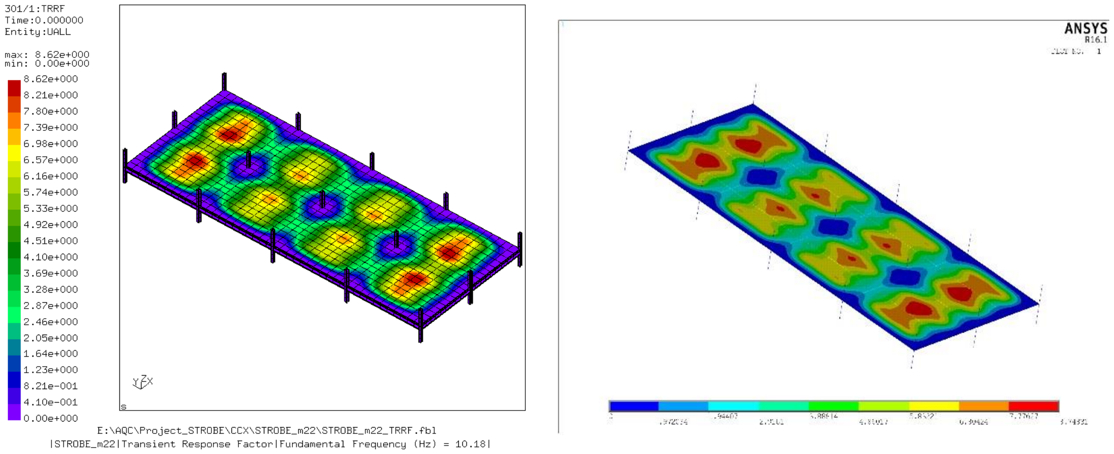
The second case study considers a more generic bay arrangement as shown in Figure 5.14. This 2 by 4 floor system uses the same structural components as the one studied in the previous section.

The frequency of the first mode was calculated by the calculation engine as 10.2 Hz and it is slightly higher than the frequency of 9.7 Hz predicted by SCI/ANSYS tool. A comparison of frequencies and mode shapes for the first 10 modes is presented in Appendix B. Steady-state and transient response factors are compared in Figure 5.14 and Figure 5.15 respectively.



(a) Calculation engine, $R_{ss,\ max}$ = 2.65 (b) SCI/ANSYS, $R_{ss,max}$ = 3.34

Figure 5.14 Comparison of steady state response factor of a 2 x 4 floor model



(a) Calculation engine, $R_{t,\ max}$ = 8.62 (b) SCI/ANSYS, $R_{t,max}$ = 8.75

Figure 5.15 Comparison of transient response factor of a 2 x 4 floor model

For the same reasons as discussed previously, the response factors predicted by the calculation engine for the FVA tool are slightly smaller than that from the SCI/ANSYS vibration analysis tool, due to the higher frequency predicted by the CCX model.

6 CONCLUSION

The development of the floor vibration analysis (FVA) tool and validation of its calculation engine have been completed and presented in this report. Floor vibration analysis methods in accordance with P354[2] implemented in the Python libraries were also summarised.

The study showed that the calculation engine developed by SCI can accurately perform vibration response analysis. Modal frequencies and mode shape amplitudes extracted by CalculiX are in reasonably good agreement with the commercial FE package ANSYS and ABAQUS.

The response factors are shown to be sensitive to the fundamental frequency of the floor system and less sensitive to mode shape amplitudes/modal mass.

Owing to the way the beams are modelled (using solid brick elements), the natural frequencies of CalculiX models are slightly higher than that of the ANSYS, which leads to lower steady-state response factors. Transient response factors are in better agreement than steady-state response factors as they are not affected by the dynamic magnification factor.

6.1 Further work

This report (D4.2) concludes the first phase of work in Task 4.2. The calculation engine will be used in Task 4.3 to perform a comparative study of high strength steel floors and normal strength steel floor systems (D4.4). After that, the user interface will be developed by SCI (part 1 in Figure 4.1) with complete background documents (D4.3).

Further improvement to the calculation engine is also identified. The prediction of modal frequencies can be improved by increasing the number of solid elements to model the beam/column cross-section, at the expense of increasing solution time.

As the calculation engine will be used as an online backend application, optimisation of the Python pre- and post-processing libraries will be beneficial as it helps to reduce the run time of the FVA tool.

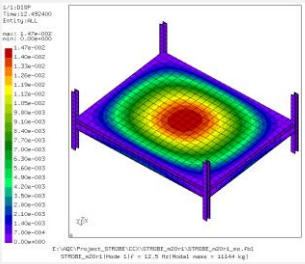
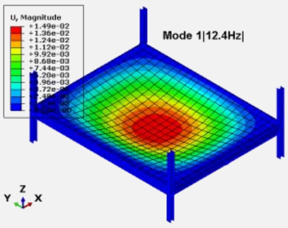
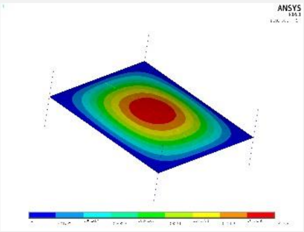
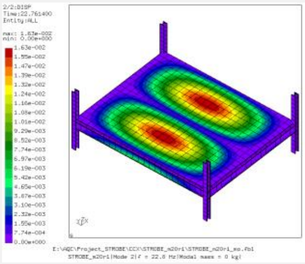
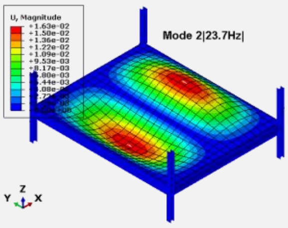
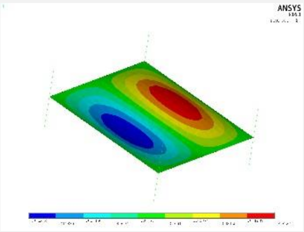
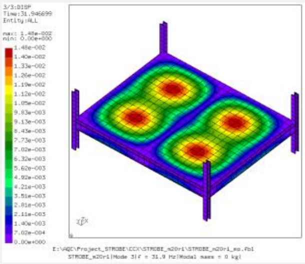
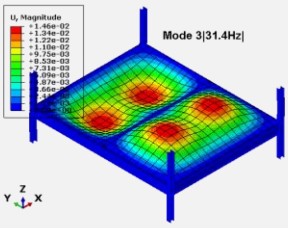
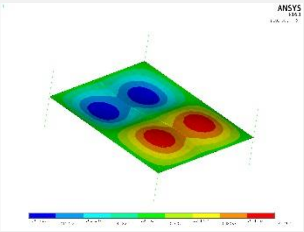
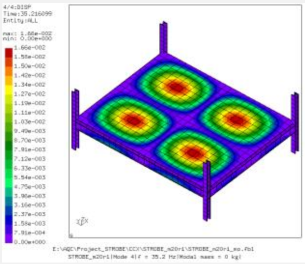
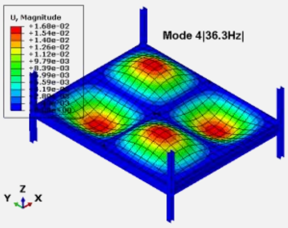
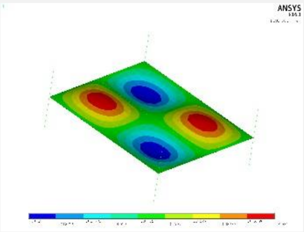
7 REFERENCES

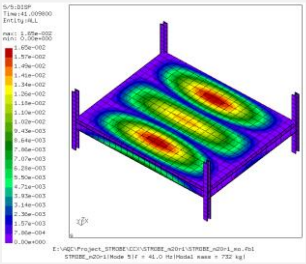
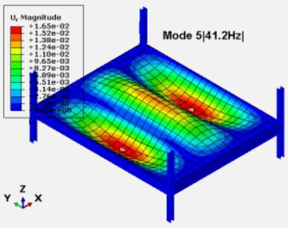
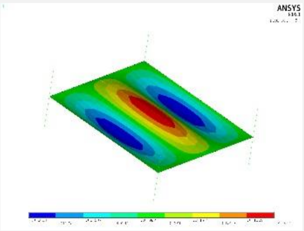
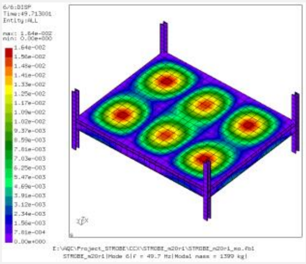
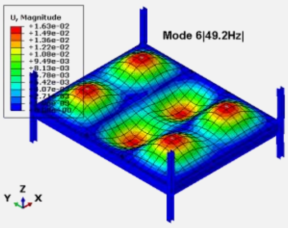
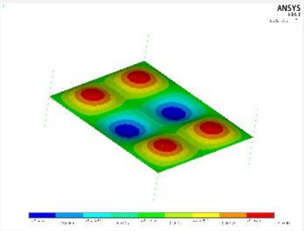
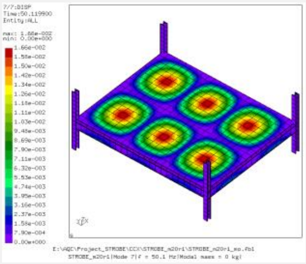
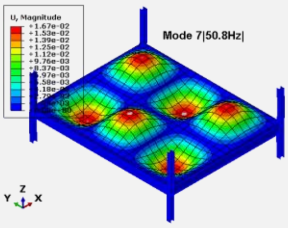
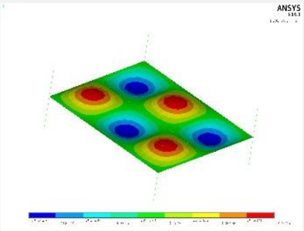
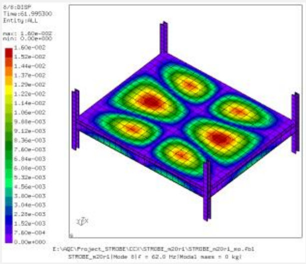
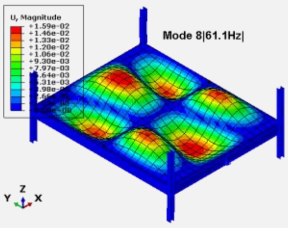
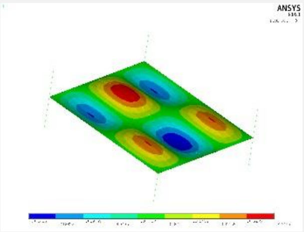
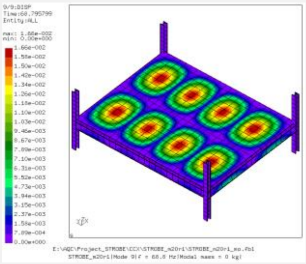
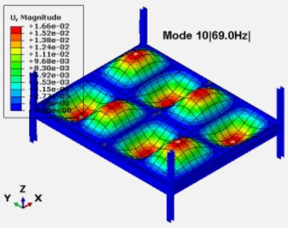
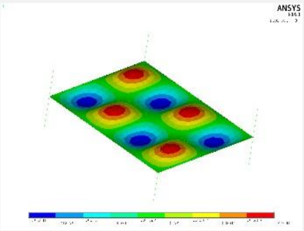
1. Human-induced vibration of steel structures (HIVOSS), RFS2-CT-2007-00033, Research Fund for Coal and Steel, European Commission, 2010
2. Smith, A.L., Hicks, S.J. and Devine P.J., Design of Floors for Vibration, A New Approach P354. Revised Edition, Steel Construction Institute, 2009
3. ISO 10137:2007 Base for design of structures – Serviceability of buildings and walkways against vibrations
4. Dhondt, G. and Witting K., CalculiX: A Three – Dimensional Structural Finite Element Program, Version 2.14, 2018. (http://www.calculix.de/)
5. BS 6841:1987 Guide to measurement and evaluation of human exposure to whole-body mechanical vibration and repeated shock, British Standards Institute, 1987
6. BS 6472:1992 Guide to evaluation of human exposure to vibration in buildings (1 Hz to 80 Hz), British Standards Institute, 1992
7. ISO 10137:2007 Bases for design of structures – Serviceability of buildings and walkaways against vibrations, International Organisation for Standardization, 2007
8. Witting, K., CalculiX GraphiX User's Manual, version 2.14, 2018
9. Dhondt, G., CalculiX CrunchiX User's Manual, version 2.14, 2018

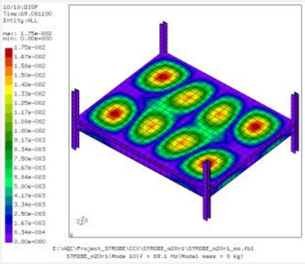
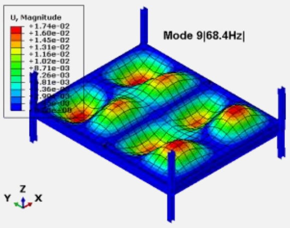
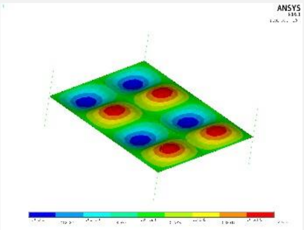
Appendix A Comparison of mode shape and frequency of 1x1 floor model

Mode shape and frequency of the first 10 modes of the 1 x 1 floor system are compared in this appendix.

Table A.1 Comparison of mode shape and frequency of 1x1 floor model

Mode	CCX	ABAQUS	ANSYS
1	 12.5 Hz	 12.4 Hz	 11.7 Hz
2	 22.8 Hz	 23.7 Hz	 21.9 Hz
3	 31.9 Hz	 31.4 Hz	 31.0 Hz
4	 35.2 Hz	 36.3 Hz	 33.2 Hz

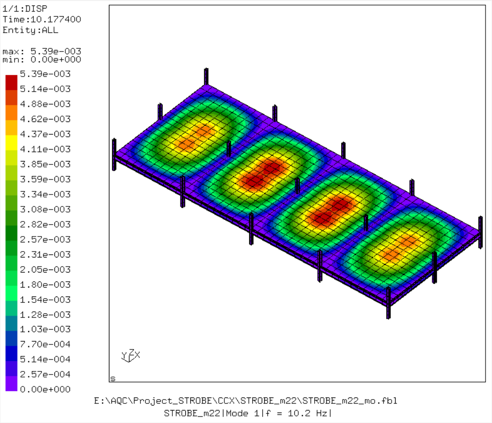
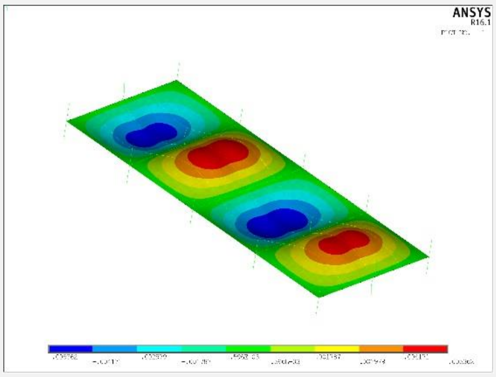
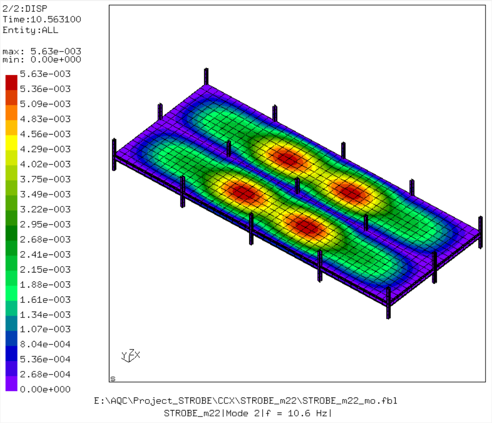
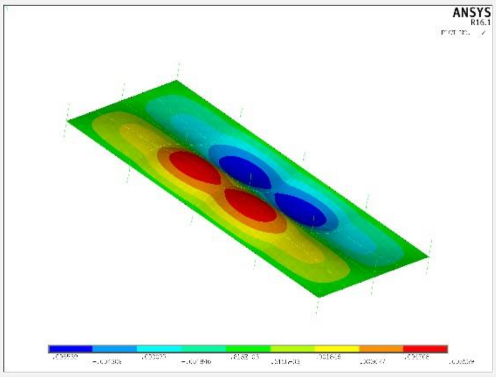
Mode	CCX	ABAQUS	ANSYS
5	 41.0 Hz	 41.2 Hz	 40.3 Hz
6	 49.7 Hz	 49.2 Hz	 47.7 Hz
7	 50.1 Hz	 50.8 Hz	 46.9 Hz
8	 62.0 Hz	 61.1 Hz	 61.1 Hz
9	 68.8 Hz	 69.0 Hz	 64.4 Hz

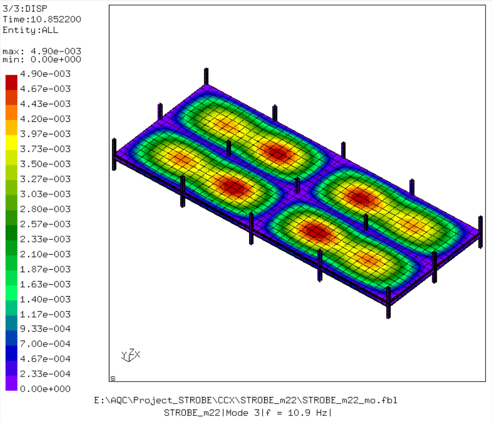
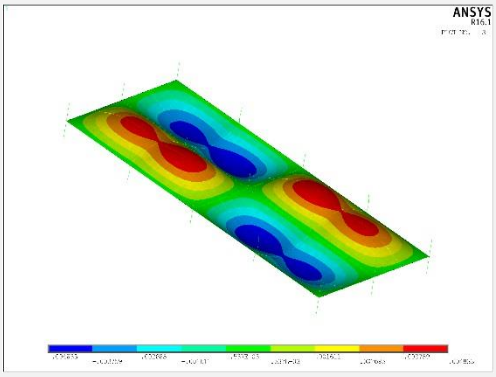
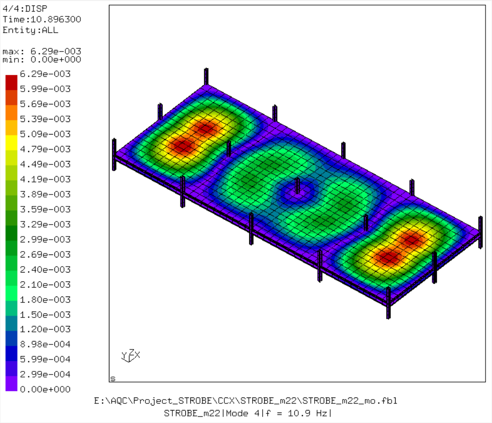
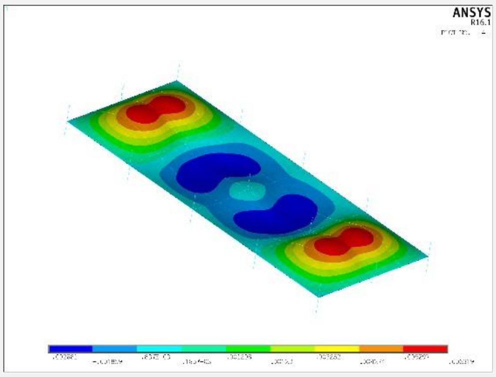
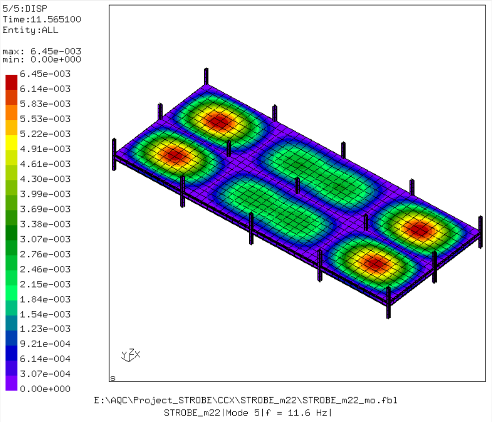
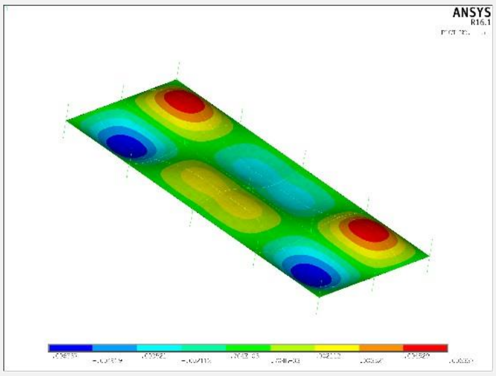
Mode	CCX	ABAQUS	ANSYS
10			
	69.1 Hz	68.4 Hz	65.4 Hz

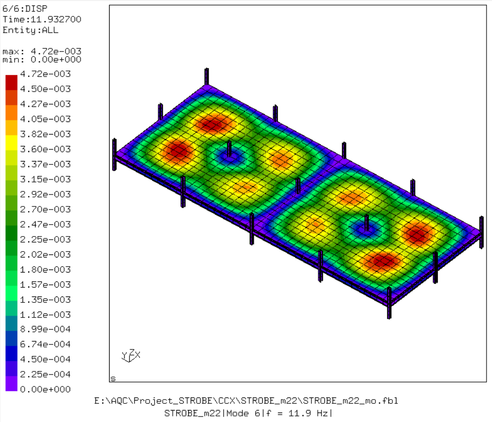
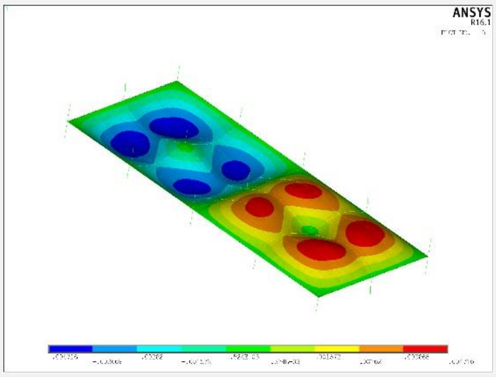
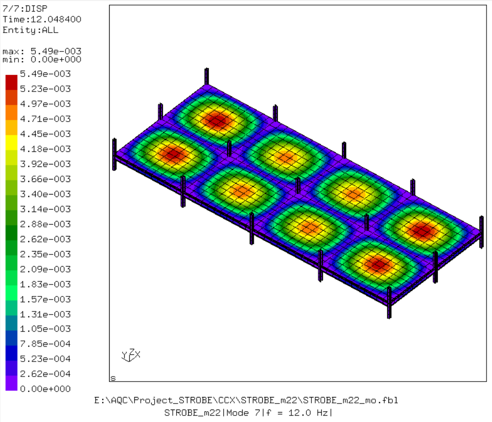
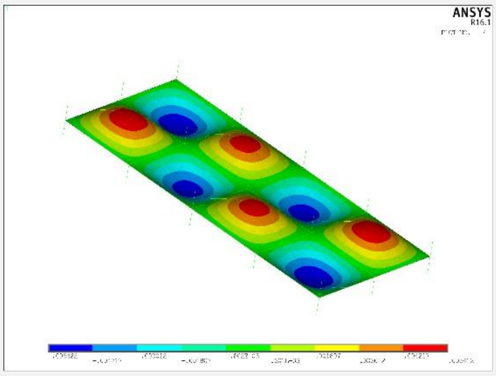
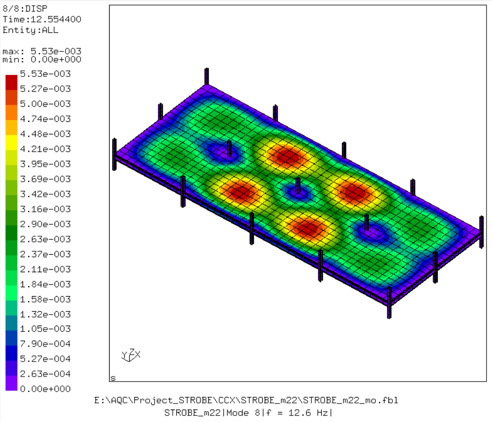
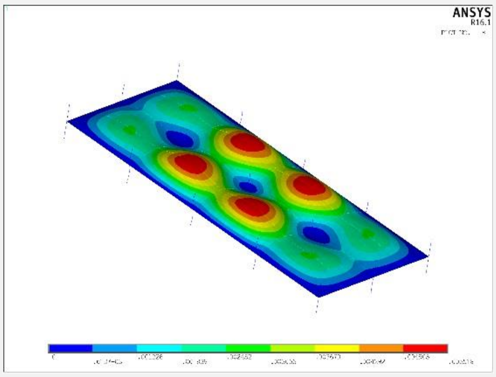
Appendix B Comparison of mode shape and frequencies of 2x4 floor model

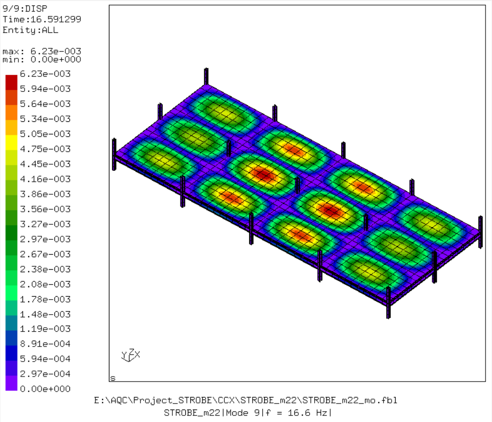
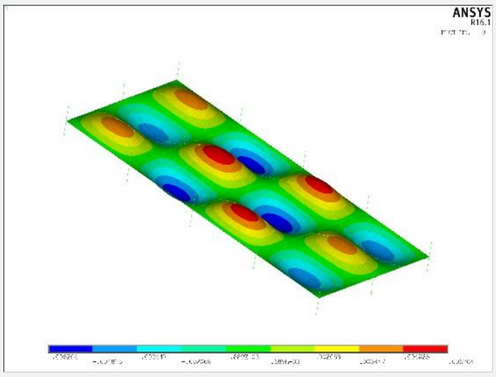
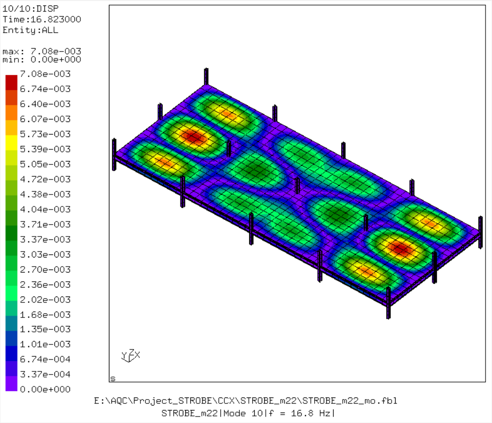
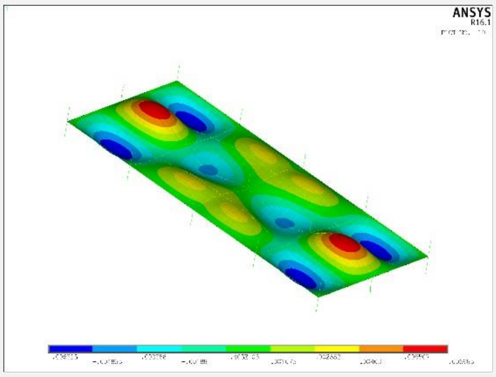
Mode shape, frequency and modal mass of the first 10 modes of the 2 x 4 floor system are compared in this appendix. CalculiX FE model is in good agreement with ANSYS.

Table B.1 Comparison of mode shape and frequency of 2x4 floor model

Mode	CCX	ANSYS
1	 10.2 Hz	 9.7 Hz
2	 10.6 Hz	 10.1 Hz

Mode	CCX	ANSYS
3		
	10.9 Hz	10.5 Hz
4		
	10.9 Hz	10.5 Hz
5		
	11.6 Hz	11.2 Hz

Mode	CCX	ANSYS
6	 11.9 Hz	 11.6 Hz
7	 12.0 Hz	 11.7 Hz
8	 12.6 Hz	 12.3 Hz

Mode	CCX	ANSYS
9		
	16.6 Hz	16.2 Hz
10		
	16.8 Hz	16.4 Hz

The frequencies and modal mass are presented and compared in Figure B.1 and Figure B.2 respectively.

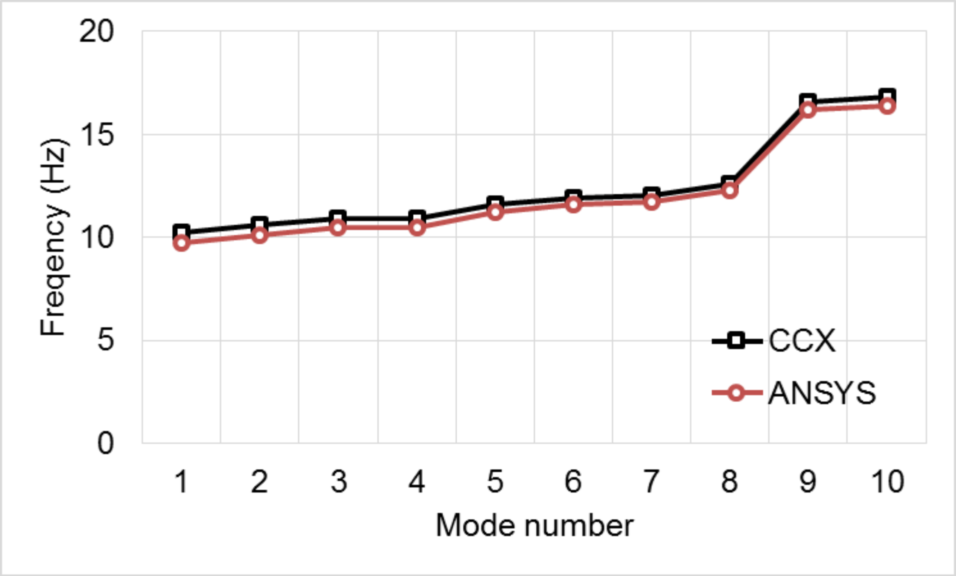


Figure B.1 Comparison of modal frequencies for the first 10 modes

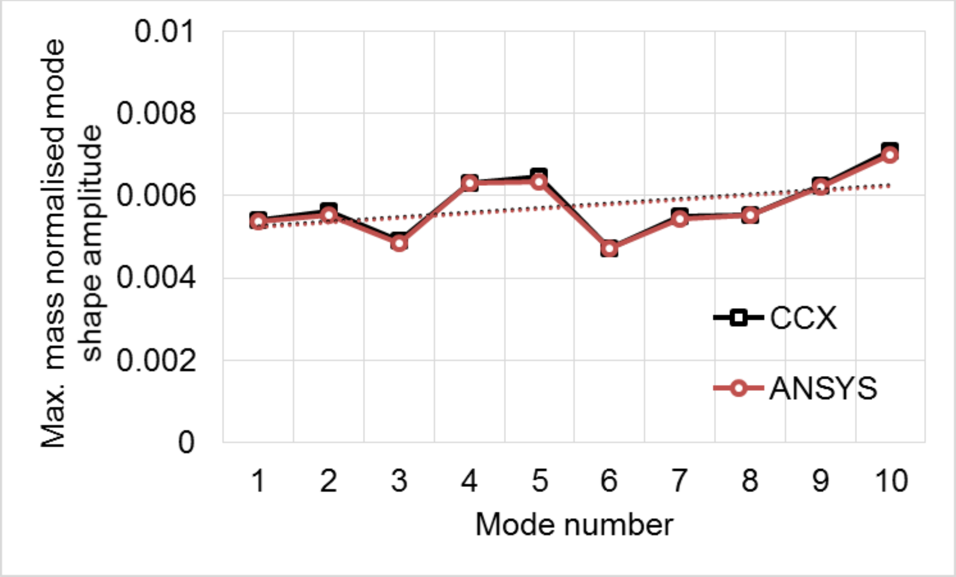


Figure B.2 Comparison of mass normalised mode shape amplitude for the first 10 modes