Framework Manual

This report is a guide to the Framework methodology, introduced in the Main Report, Chapter 4. It leads the reader from the creation of the geometry, to the material definition, load application, analysis and obtention of the results.

1 Geometry definition

Naturally, the first thing to do when it comes to design is to actually generate the building's volume. In the first phases of the design, the architects' reflections are likely to lead to a general rooms of floor layout, answering the needs of the project. In the modular inspired design, a pattern of similar geometries will generate the overall volume. This allows it to be created from a floor plan delimiting the boundaries of the different spaces.

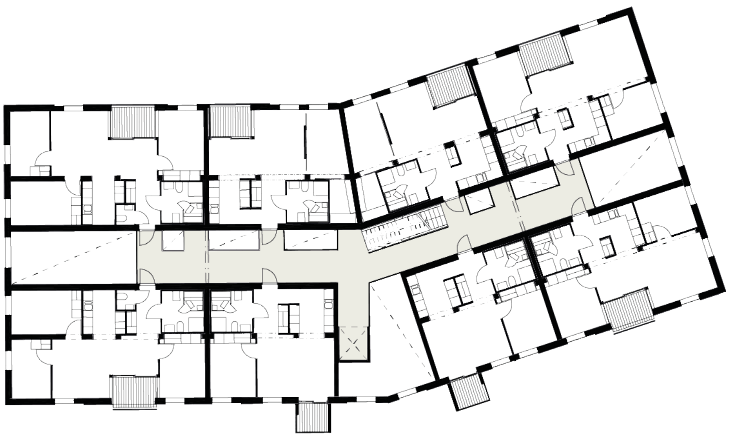


Figure 1.1: Puukuokka building floor plan, example of base geometry

A floor plan like the one presented above can be used as a basis for the framework's development. It should be streamlined to keep only the wall layout, which is enough as an input to a structural model. Along with the number of stories and their height, such data is sufficient to start with the geometry step of the method.

1.1 Building layout

Vertical elements boundaries

The floor plan, either drawn by hand or in a CAD format, is to be reproduced in the modelling software Rhinoceros. For this very first step, the designer is free to choose his/her way to reflect the layout accurately by inputting the physical space boundaries into the model. These boundaries represent vertical 2-dimensional entities, which can host a single structural sub-system. Since this thesis primarily focuses on CLT panels, these are defined by walls. However, in a broader context, this could also mean a more or less complex arrangement of beams and columns. The presence or absence of a structural sub-system on a given boundary relies entirely on the designer's will, as explained later.

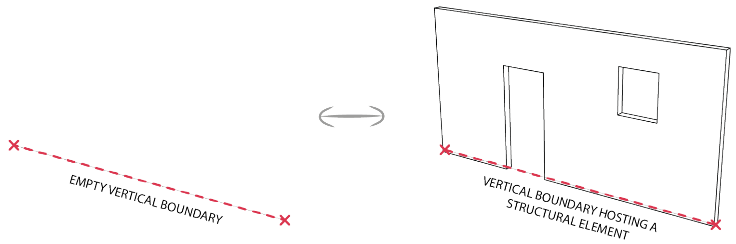


Figure 1.2: Two possible states of a vertical element boundary

Two methods can be thought of when materialising these boundaries: a fully parametric approach, or a simple sketch.

The first approach offers a great flexibility to modify the design at a further stage: it is for example very easy to set a different room length, or change any parameter that is defined. Moreover, parameters can serve as inputs for generative design algorithms that can fulfil different tasks, such as structural optimisation. The detailed parametric design for the case study project can be found in the Digital Appendix.

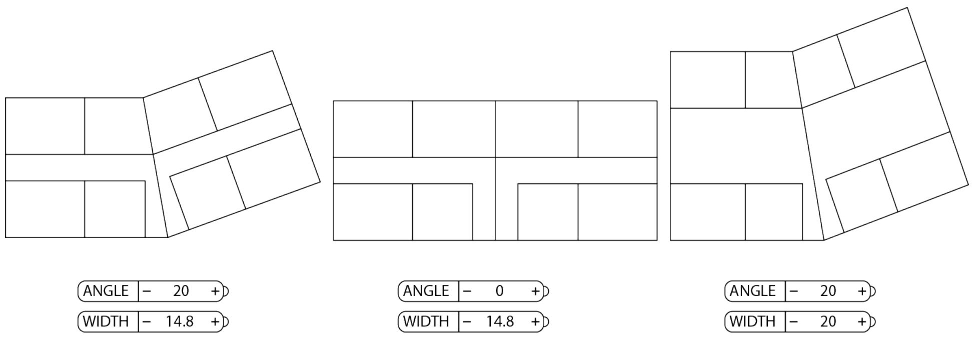


Figure 1.3: Parametric variations for the reference building

However, this method requires a greater mastering of visual programming, taking up more rigour and time. On the other hand, direct sketching of the geometry is very easy and quick, but does not offer as wide as a range of possibilities.

Regardless of the approach chosen by the designer, a simple rule must be remembered: to one given boundary, one and only one structural sub-system can be attributed. This implies, for example, that the building outline (e.g. its envelope) has to be created either as a collection of lines (Figure 1.4, left), or as breakable objects such as individual room boundaries (Figure 1.4, centre). It is not recommended to use macro-boundaries such as e.g. grid lines (see Figure 1.4, right), since more than one structural element might be needed to model this building section.

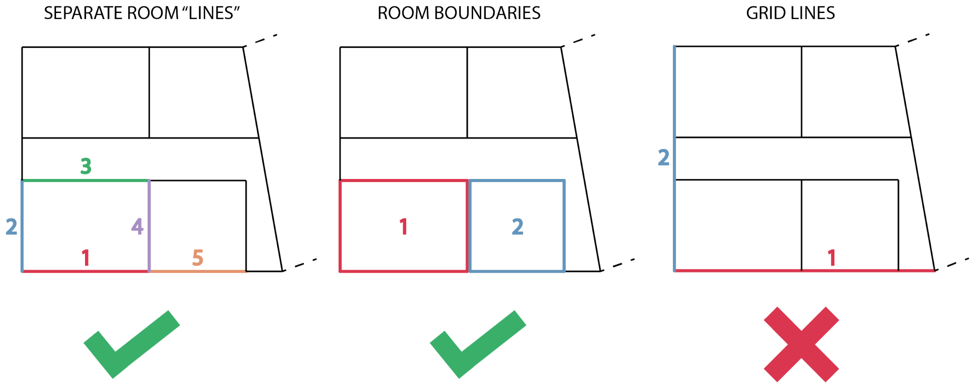


Figure 1.4: Boundary generation

This step results in the first data container, which represents the vertical elements boundaries.

Horizontal elements boundaries

Once each vertical element boundary has been created, it is necessary to assemble the horizontal space boundaries. These spaces can represent, for example, a room, and will be associated with one single slab that spans across them. For a room materialised by 4 edges, the adequate boundaries must be joined to form one space contour, as shown in the Figure 1.5 below.

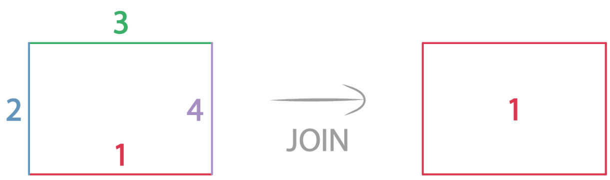


Figure 1.5: Horizontal boundary generation

If the room outline approach was chosen, this step is simplified since the boundaries were already associated from the beginning. The step results in the creation of a second data container, representing the horizontal elements boundaries.

At this stage, the model therefore consists in a collection of boundaries that represent the building layout on the ground floor. They will be later used to generate the horizontal and vertical structural elements, but must first be given an unique identity.

1.2 Object referencing

In order to easily identify the numerous objects that will be created during the process, it is important to identify each of them with a unique ID, allowing the designer to control their characteristics. To this purpose, two data structures are proposed: one for horizontal elements (floors) and one for vertical elements (boundaries, whether hosting a wall or not). The first ones are defined by a 2-level data structure, while the vertical objects show a 3-level data structure, as explained in Figure 1.6 below:

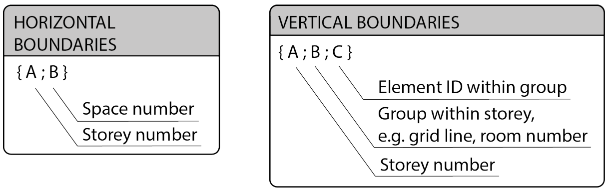


Figure 1.6: Proposed data structure for floor and wall contours

The chosen path relies on the embedded tree structures in Grasshopper, though it is not the only way to realise such data handling.

For an optimum control, it is recommended to create the data structures of the initial level manually; it will then facilitate the identification for further purposes.However, this task might reveal itself to be tedious in larger projects. If the designer wishes it, he/she can make use of a Grasshopper component developed in the frame of this thesis, the "Referencing" component. This component takes in input the geometry (floors and vertical boundaries) and creates the relevant data structure. Moreover, the given identifiers can be displayed on the geometry for further reference.

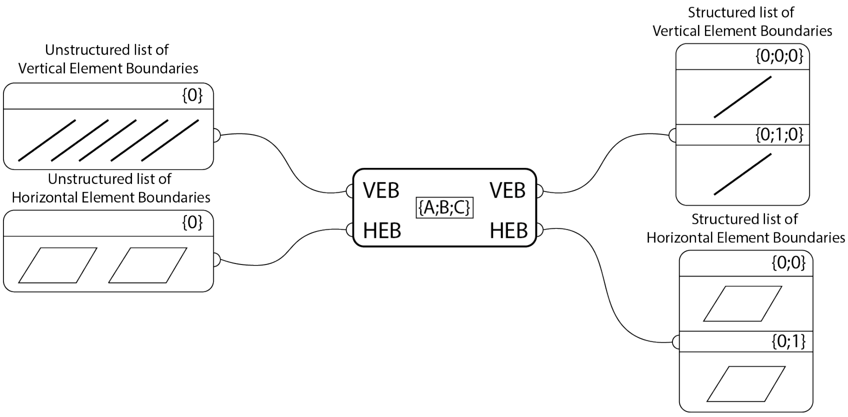


Figure 1.7: Referencing component

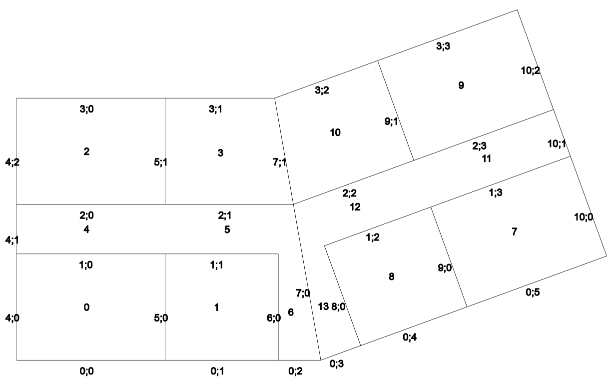


Figure 1.8: Referencing component applied to the case study objects, ground floor

It is unfortunately harder to create a consistent identification, since it depends on the order in which the elements are transferred to the component.

1.3 Storey generation

The two structured data containers created at the end of the previous step can now be used to generate the entire building's volume. To this end, the user must the Grasshopper module "Storey Generator" to specify the number of stories and the storey height, as specified on the Figure 1.9 below:

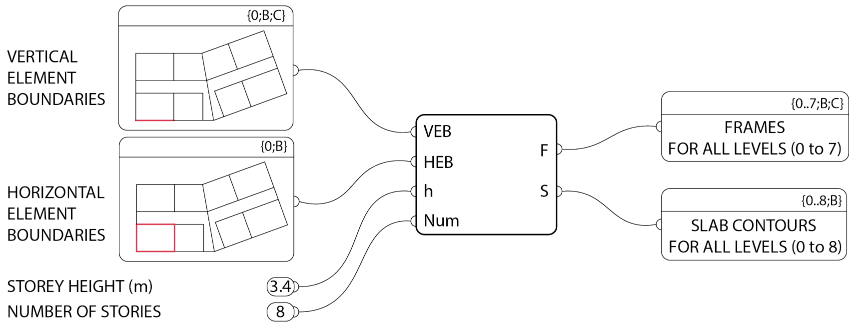


Figure 1.9: Storey Generator component

The module will take the geometry boundaries and duplicate them at the specified interval, while ensuring a consistent report of the data structure. It is also where the *Vertical Element Boundaries* are transformed from unidimensional elements to 2D vertical rectangles, forming a collection of what will be called *Frames*.

1.4 Additional features

At this stage, the building which is created consists in several identical stories. For some projects, this can be sufficient to proceed through the next step, since it features a flat roof. However, in some architectural proposals (cf. the actual case study), more work is needed to reflect features such as a pitched roof. This can be done in many ways, including parametrically (which was chosen in this implementation). The data structure must also be created consistently with the existing one, should such structure be included in the following steps.

2 Populating the model

So far, the geometry that has been created only represents potential locations for structural elements. These elements have yet to be assigned to the designated *Frames* to complete the model. The concept is based on a modular/prefabricated pattern, where a limited amount of designs are repeated to populate the building volume.

2.1 Wall type creation

In this step, the designer starts implementing the architectural proposal in itself. Indeed, the CLT panels design is generated and defines the buildings aesthetics, since they will be an integral part of features such as its envelope or internal flexibility.

To this end, an interface referencing several wall types is created with the *Genotyper* component.

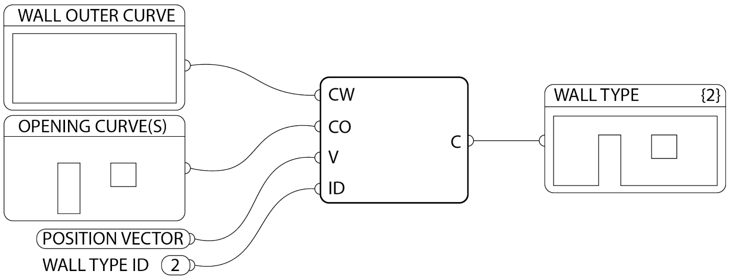


Figure 2.1: Genotyper component

When adding the *Genotyper* component to the Grasshopper definition, a new rectangle will appear. This geometry, with the dimensions 16 x 4m, represents the maximum dimensions for standard CLT panels, as produced by the manufacturer Stora Enso [1]. Within its borders, one can be guaranteed that the product is technically feasible.

These boundaries represent a canvas where the designer is free to sketch a wall type elevation, consisting in one polyline element to describe the outer geometry, and one or several polylines representing openings if necessary.

The curves generated must be fed to the *Genotyper* component, along with an ID number to identify the specific type. The final input (vector type) can be used to move the interface where suitable, for visibility concerns.

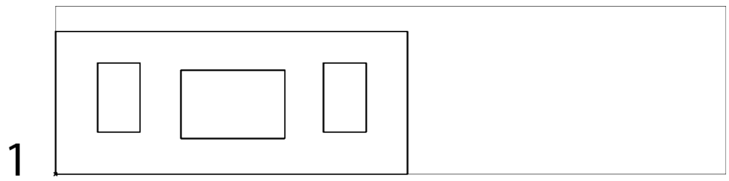


Figure 2.2: Genotyping interface where a wall has been sketched

This example is fairly classic in itself, in that it represents straight elements. In reality, any design is possible, within the specified dimension limits.

2.2 Model population

The wall types that were defined at the previous step constitute the entirety of the vertical structural elements. They can be seen as *Genotypes*, i.e. a set of characteristics (geometry, layer arrangement, etc.) that are adapted to the context of the *Frames*, thus becoming *Phenotypes*. The *phenotypes* are the elements generated when the *genotypes* are placed in the context of the building space; they are individual elements which can vary in shape to adapt to their environment, while inheriting the global identity from their parents. Together, they populate the space to form the structural system, responsible for transferring the loads to the foundations.

The *Phenotyper* component, developed for this framework, ensures the transition from *Genotype* to *Phenotype*. It allows the designer to send a given wall type to a set of *Frames*.

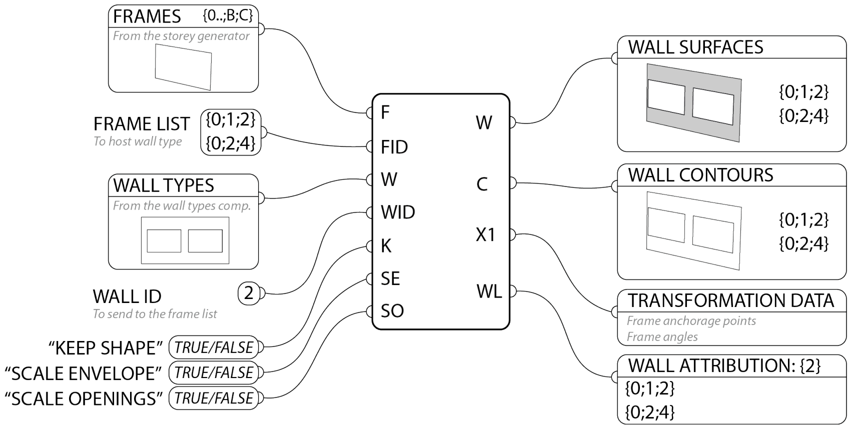


Figure 2.3: Phenotyper component

One *Phenotyper* component is necessary per each wall type. To send a wall type to the desired building location(s), one must plug the referenced vertical element boundaries (from the end of Step 1) and the wall types data, which has been assembled previously. The last mandatory inputs consist in a list of locations where the wall type shall be sent, as well as its identification number. The result can be seen on Figure 2.4.

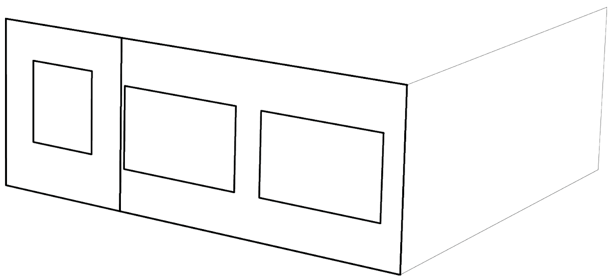


Figure 2.4: Example of frame population or *phenotyping*

This process reflects the *phenotyping* concept: a same wall type can be sent to several *Frames*, which might not accommodate it perfectly. For example, the design could be too large or too high. Therefore, the wall has to adapt to its new host. The adaptation can be adjusted with the remaining three option, which also provide a greater flexibility to the user.

The first option, "Keep shape", can be used for "standard" designs such as the one presented in the case study. When set to *False* (recommended setting), the outer shape of the wall is replaced by the space defined by the *Frame*. For example, when the wall type is too large (as shown in Figure 2.5 or too little, its outer shape is either cropped or extended. Basically, this means that only the openings are sent to the *Frame*.

This option is ideal when dealing with varying spaces, while minimising the number of wall types that need to be designed. For instance, this option has been used to recreate the entirety of the Puukuokka design with only the five wall types introduced previously.

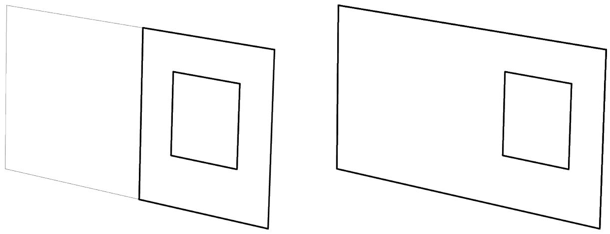


Figure 2.5: Example of use of the "Keep Shape" option - wall keeping its original outer boundary (left) and wall merged with its hosting frame (right)

However, setting the option to *False* is restrictive in terms of design. In cases where the architect wants to introduce complex shapes, it is advisable to craft a specific wall type and ensure that it fits within its given location.

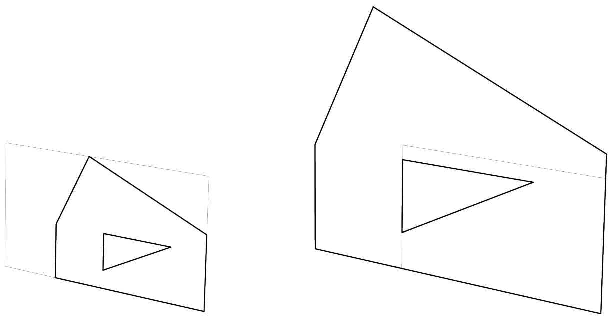


Figure 2.6: Example of use of the scaling options - unscaled wall (left) and scaled wall (right)

As far as the outputs are concerned, the *Phenotyper* component features four of them, crucial for the next stages. The first one (W for Walls), contains the generated wall surfaces. They will be used to generate the Karamba model. The second one, C for Contours, contains the wall contours, which can be used for diverse purposes, such as separating envelope and openings. The third one, WL for Wall List, contains the list of created phenotypes ID, in a data tree referencing their genotype. Its purpose is to determine the heredity of an element; it is therefore possible to assign other characteristics later on, such as the material and CLT panel layup (next step). Finally, the X1 output contains transformation information which will be useful when retrieving the results of the analysis.

For each *Phenotyper* component that has been used (namely, each wall type assignment), these outputs must be assembled in data containers before proceeding to the next step, which will define CLT layups and their respective mechanical properties.

At the end of this preliminary stage, the model should be populated by several wall types. An example of how this should look can be found in the Main Report, Figure 6.2.3.

3 Material Definition

At this stage, the building's geometry has been fully generated. The materials in use must now be defined; this includes the CLT layups, as well as the specification of their mechanical properties and their association with a given wall type. This process is achieved via the Excel file "Layup.xlsx", which is made to communicate with the Grasshopper definition through the *Layup XL* component. One must place the component in the Grasshopper definition and input the file path of the Excel sheet, giving out four outputs: *"CS List"*, *"LAYUP"*, *"STIFFNESS"* and *"STRENGTH"*. The designer must then define the CLT layup as described in section 3.1. All the following sections refer to the Appendix B.1; the reader is invited to consult both documents simultaneously.

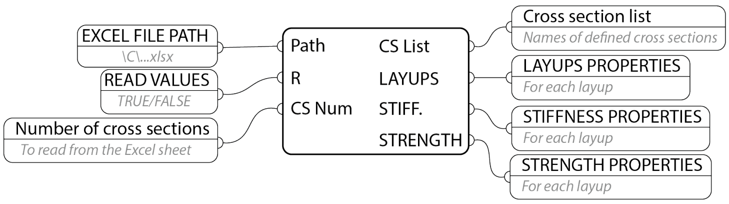


Figure 3.1: LayupXL component

3.1 CLT Layup definition

As explained in the Main Report, Chapters 2 and 3, CLT panels are made of a crosswise arrangement of timber layers. Depending on the thickness of these layers, their orientation, or even the width of the timber boards they are made of, the CLT panels present different strength and stiffness properties. In this step, the designer is left completely free to define the CLT layups to fulfil a given function, whether it will be used as a wall or a slab.

This step is realised with the *"Layup"* sheet of the Excel file, where each column represent a specific layup.

For each of them, one must give a name which will serve to identify it, and specify the desired grading among *CL24h* and *CL28h*, which were proposed in [2]. The number corresponds to the bending strength of the CLT material, respectively 24 MPa and 28 MPa.

The next entry specifies the width of the timber lamellae forming each layer, between 40 and 300 mm, as well as the gap width between them which must not be over 6 mm [3].

The layup is then defined by specifying the thickness and orientation of each layer, which will in return inform the total thickness of the layup. The latter must not be over 500 mm [3].

When all layups have been defined, one can fill the *Applies to* cells, which will associate a layup with the wall and floor types present in the geometry; to this purpose, the tree selection syntax from Grasshopper must be respected (more information can be read as comments on the Digital Appendix "Layups.xlsx").

Grey cells only serve an informative purpose and shall not be modified by the user.

This process lacks however a visual feedback, and it might be difficult to check if the intended layup has been associated with the right wall. To compensate for this, one can plug the *"LAYUP"* output from the Excel sheet reader to the *Cross Section Visualiser* component. Its effect is to create a visual representation of the cross section, complete with name and thickness information, side by side with the designated wall in the wall types interface.

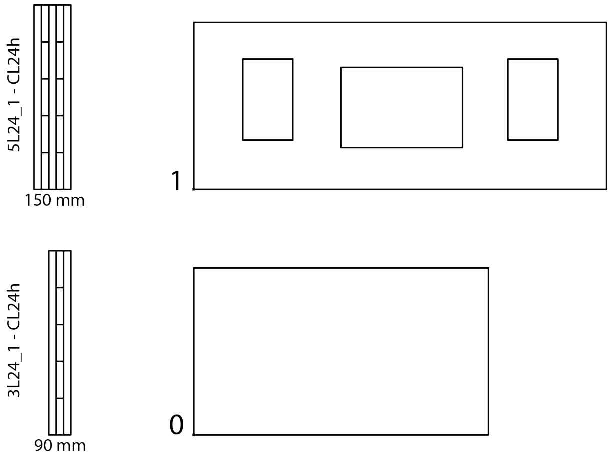


Figure 3.2: Cross section viewer in the wall type interface

3.2 CLT Layup properties

The information set in the previous stage serve as a basis for the calculation of each CLT panel's mechanical properties. The strength and stiffness properties are retrieved for each layup to inform the structural analysis; they are listed in the "STRENGTH" and "STIFFNESS" sheets. The active input of the designer, in this step, ends here; nevertheless, it is recommended to monitor the values to ensure their consistency.

Strength properties

The strength properties of each CLT layup is referenced in the "STRENGTH" tab. All strength properties from the Main Report, Chapter 3 are indexed in two tables, whether they are or not dependent on the CLT layup.

The header of the first table resumes the name, class and elements from the "LAYUP" sheet. Afterwards, the upper end of the table shows the results of the characteristic Bending strength $f_{m,CLT,k}$, Tensile strength $f_{t,0,CLT,net,k}$ and Compression strength $f_{c,0,CLT,net,k}$ in the main panel direction, as well as the Shear strength in-plane $f_{v,net,k}$ and the Rolling shear strength $f_{r,CLT,k}$. They are calculated from the equations 3.1.1 to 3.1.9 presented in the Main Report.

The second table presents the remaining values, including the tensile $f_{t,90,CLT,k}$ and compression strength $f_{c,90,CLT,k}$ perpendicular to grain, the Gross shear $f_{v,gross,k}$ and torsional shear strength $f_{T,k}$, and finally the shear strength out-of-plane $f_{v,CLT,k}$. These values are taken from the table 3.1.1 from the Main Report.

The lower part of each table deals with the design values of these strength properties. Indeed, when checking the resistance of a material, one must apply safety factors on characteristic values. The design values are obtained from the Section 3.1.4, Main Report, and will be later compared to the design stresses in the element.

Stiffness properties

In the same fashion as the "STRENGTH" sheet, the "STIFFNESS" sheet calculates the different stiffness properties of each layup. Once again, the values need not be modified but it is recommended to check their validity.

The two first tables show the characteristic and design stiffness properties, retrieved from Section 3.3 from the Main Report. The latter take into account the nature of the CLT arrangement. However, as explained in the Main Report, Chapter 4, Karamba does not allow the definition of a complex material such as CLT. This is why the first table also presents equivalent Elastic moduli. They are derived from the stiffness properties by division with the gross moment of inertia or area of the CLT panel, for a 1m length:

$$E_{eq,OP} = \frac{K_{CLT}}{I} = \frac{K_{CLT} \cdot 12}{t_{CLT}^3} \tag{3.1}$$

$$E_{eq,IP} = \frac{D_x}{t_{CLT}} \tag{3.2}$$

In this way, properties such as bending and axial stiffness will have identical values when calculated by the software. Some of the design values (as explained later in section 5.3) will serve as inputs for the material definition in Karamba.

Finally, the bottom of the "STIFFNESS" sheet references the mean density of the material. It has been placed here for practical purposes, when reading the values from Excel in Grasshopper.

4 Load definition

Now that the geometry is fully defined along with the material properties, the loads which are to be applied to the model have to be characterised. This is carried through the "Loads" Excel file, which has been conceived in such a way that very little input is necessary in order to perform basic load calculations, while leaving the opportunity to the user to make more advanced considerations if needed. Similarly to the former material definition, the GhExcel component shall be placed in the Grasshopper definition, and the designer must fill in the required fields of the file as described underneath. The loads considered in this framework cover four types of loads, which are dead loads, imposed loads, wind loads and snow loads.

4.1 General information about the project

The "GENERAL" tab contains some crucial information about the project. It includes geometric information as well as incidental information, such as the country where the project is realised and the building's consequence class. An example for the Puukuokka building is found in the Appendix B.2, and each specific field is explained on the Figure 4.1.

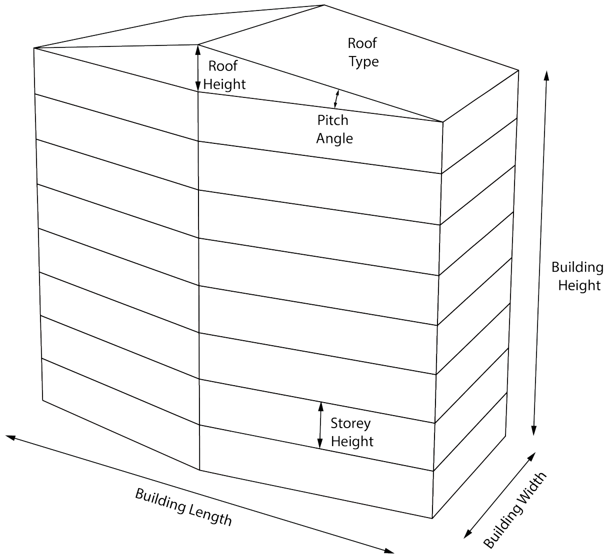


Figure 4.1: Field entry in the General tab

The last two pieces of information relate the location and type of building. The choice of the country is left between Denmark, Finland, Norway and Sweden; the choice impacts values such as wind velocity, snow load, imposed loads, and even load combinations themselves. Up to this point, only values from the Danish National Annexes have been collected, and a warning is displayed if the designer chooses a different country.

The consequence class value relates to potential economic, social, environmental or loss of human lives risks in case of building failure [4]. As a rule, according to the Danish National Annex to the Eurocode 0 [5], a residential building that is more than 12m high falls in the last consequence class, CC3, which is the case in our case study. For more information, the user is kindly asked to refer to the Annex B, section B3 of the Eurocode 0 [4] and the corresponding national annex. The consequence class determines the factor K_{FI} which might be taken into account in the load combinations following a national decision; it ranges from 0.9 for CC1 to 1.1 for CC3.

4.2 Dead loads - Floor and wall assembly definition

The building is subjected to dead loads, due to structure itself, but also to non-structural materials. The gravity applied on the bare structural elements are calculated directly in the Karamba software; nevertheless, it is the responsibility of the designer to inform the "DEAD LOADS" tab to take the remaining part of the dead loads into account. These include the walls and floors finishes, such as insulation, gypsum boards, floor boards, façade elements, etc. The latter are defined as surface loads in the corresponding tables, see Appendix B.2, approximated by the equation 4.1, in kN/m^2:

$$G = \sum_i \rho_i * t_i \tag{4.1}$$

Where ρ_i is the layer density in kN/m^3 and t_i is the layer thickness in mm.

For each assembly layer, one must reference the element to which it applies, the layer name (for reference), its thickness and density. This includes adding (or removing) tables for each new type of assembly that is to be included in the building. Each table is assigned to a *genotype* such as decided when defining the geometry.

For example, during the implementation of the method on the case study project, we defined 5 wall types (see Main Report, Section 6.2) and 2 floor types (for regular storeys and the roof), and these are reflected in the "DEAD LOADS" tab. The loads will therefore be applied to the corresponding elements in the model.

The total load for each *genotype* is reported on the tables situated on the right side of the sheet, which bear the characteristic values used for the load application. As the process is not fully automated, the designer must take great care that all the correct values are reported in the "Total" tables.

4.3 Imposed loads

Imposed loads apply on the floor structure; they arise from the occupancy of a building and differ according to the utilisation of space. They are defined in the Eurocode 1, part 1-1 [6], section 6.3, which attributes a load value and category to each specific use of the storey. For instance, a residential storey falls into the category A, while a roof belongs to the category H. These values might be modified by the corresponding national annex.

To these category loads are added movable partition loads, taking into account the presence of non-structural walls on the said storey. They are classified in three categories, depending on their weight: Light, Medium and Heavy.

The "IMPOSED" tab reflects these definitions. Each row corresponds to a given floor *genotype*; in our case, "Regular storey" and "Roof". The user is free to fill these cells for reference. Then, one must choose in the drop-down lists (red cells) the adequate category for both the occupancy and the partitions. This decision must be motivated by the guidelines given in [6] and the corresponding National Annex, as well as data from the project.

The remaining values are calculated automatically according (as of now) to the Danish National Annex [7]. The table also features the ψ coefficients corresponding to the chosen category, which will be useful for the load combinations.

4.4 Snow loads

The user is guided through the snow load calculations thanks to the Snow load tab. To this end, the required input from the user is wished to be kept at a minimum. However, it is of a primordial importance that the results are checked for consistency, in order to obtain relevant results. The theoretical background is explained in this section.

The snow load is calculated according to the Eurocode 1, part 1-3 [8], and the corresponding Danish national annex [9]. It is included in the load combinations as a variable action, acting on the roof.

The equation giving the snow load value for persistent design situations is the following (Eurocode 1-3 [8], section 5.2(3a)):

$$s = \mu_i \cdot C_e \cdot C_t \cdot s_k \tag{4.2}$$

Where

- μ_i is the roof shape coefficient. It depends on the roof type and is detailed later on

- C_e is the exposure coefficient reflecting the building surroundings. While it might be given directly from the Eurocode 1-1-3, Table 5.1, the danish national annex prescribes a different calculation explained below
- C_t is the thermal coefficient. This factor takes into account eventual heat loss from the roof, leading to a melting of the snow. In this framework, we will use the recommended value of 1, supposing that no heat is emitted from the roof ([8], 5.2(8)). This assumption is on the safe side
- s_k is the characteristic value of snow load on the ground. This value changes according to the project's location. In Denmark, $s_k = 1kN/m^2$ ([9], 4.1(1)).

Exposure coefficient

According to the Danish National Annex ([9], 5.2(7)), the exposure coefficient is expressed by equation 4.3:

$$C_e = C_{top} \cdot C_s \tag{4.3}$$

Where the topography coefficient C_{top}, varying between 0.8 and 1.25, depends on the surroundings ability to shelter the roof from the wind; it is given by Table 5.1a NA (see also Table 4.1) for Sheltered, Normal and Windswept topographies.

Topography	C_{top}
Windswept	0.8
Normal	1
Sheltered	1.25

Table 4.1: Recommended C_{top} values for different topographies

The size coefficient C_s takes its values between 1 and 1.25. For a *Sheltered* topography, it is always equal to 1; it is defined by the following equations in the other cases, where h is the building's height, l_1 its larger and l_2 its shorter dimensions.

- For $2h > l_1$: $C_s = 1$
- For $2h < l_1$:
  - $C_s = 1$ for $l_2 \leq 10h$
  - $C_s = 1.25$ for $l_2 \geq 20h$
  - $C_s = 1 + 0.025 \cdot (l_2 - 10h)/h$ otherwise

Shape coefficient

In this framework, calculations have been developed for two types of roof: monopitch (including flat roofs) and duopitch. For both of them, the roof shape coefficient are calculated similarly depending on the pitch angle(s) α (see [8], Table 5.2):

- $\mu_i = 0.8$ for $0° \leq \alpha \leq 30°$
- $\mu_i = 0.8 \cdot (60 - \alpha)/30$ for $30° < \alpha < 60$

- $\mu_i = 0$ for $\alpha \geq 60°$

For the monopitch roof, a single case ensues, where the shape coefficient $\mu_1(\alpha)$ is applied on the entire roof surface ([8], 5.3.2). However, the duopitch roof considers three arrangements reflecting the eventual drifting of the snow, due to the wind. These arrangements are obtained by weighting each shape coefficients $\mu_1(\alpha_1)$ and $\mu_1(\alpha_2)$, as shown on Figure 4.2, left, cases (ii) and (iii) ([8], 5.3.3).

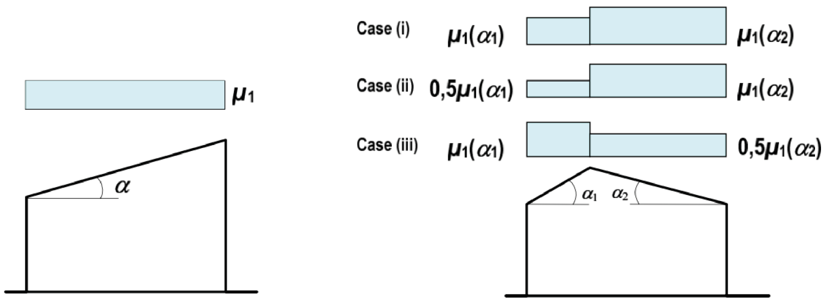


Figure 4.2: Load arrangements for monopitch (left) and duopitch (right) roofs ([8], 5.3.2-3)

Application to the Snow Load tab

The snow loads detail is available in the "SNOW" tab, which follows the considerations presented in the previous section. References to the relevant code are indicated in blue.

The calculation takes into account four parameters, among which three should have already been defined on the "GENERAL" tab: the building location (only DENMARK fully available at the moment), the roof type (monopitch/pitched) and the pitch angle. They are reported in the yellow cells. The last parameter is connected to the location's topography, namely if the building surroundings are able to provide shelter from the wind which might cause snow drift. The designer must choose between "Windswept", "Normal" or "Sheltered" in the red cell. This is the only input required from the designer in the "SNOW" tab; he/she is invited to motivate the decision by referring to the corresponding code section referenced near this cell.

The second table considers the parameters prescribed earlier on to evaluate the snow load for each roof type, for drifted and undrifted cases. To limit the amount of load cases, only the maximum of these values is reported to the green cells, which will be used in the load application.

4.5 Wind Loads

The "WIND" tab generates the values for wind loads as applied on the façade and the roof. It is therefore separated in these two major parts. The basic principles of wind load calculation is a must to ensure sensible results.

In this framework, the wind calculation is performed according to the Eurocode 1, part 4 [10] and the danish national annex [11] in the simplified case of a rectangular building. This approach is conservative and should yield safe results in our case; it is however recommended to perform advanced tests using wind tunnels for complex shapes and high rise structures.

The pressure applied by the wind on the façade is linked to what the building code names the peak velocity pressure $q_p(z)$, whose profile varies with the height. Its value depends on the mean wind speed $v_m(z)$ and the turbulence intensity $I_v(z)$.

Mean wind calculation

The Danish National Annex [11] gives a basic wind velocity of $v_{b,0} = 24m/s$. This value, which can vary between countries, is coupled with a directional factor c_{dir} and a seasonal factor c_{season} which allows it to fluctuate. However, the structural design considers the worst case by using a recommended value of 1. The basic wind velocity v_b can therefore be obtained with the following equation:

$$v_b = c_{dir} \cdot c_{season} \cdot v_{b_0} \tag{4.4}$$

This basic velocity can be altered depending on the structure surroundings. Indeed, the building can be more or less sheltered from the wind whether it is located in a dense area, for instance in a city, or in a very opened coastal area. The terrain category, selected from I to IV as per the Annex A from the Eurocode 1, part 1-4 [10], sets the basic terrain parameters z_{min}, z_{max}, z_0 and $z_{0,II}$ which will serve as boundary values for the wind profile acting on the façade (see EC1-1-4, Table 4.1).

Indeed, it will influence the calculation of the terrain factor k_r and the roughness factor $c_r(z)$ as is shown below:

$$k_r = 0.19 \cdot \left(\frac{z_0}{z_{0,II}}\right)^{0.07} \tag{4.5}$$

$$c_r(z) = \begin{cases} k_r \cdot ln\left(\frac{z}{z_0}\right) & \text{for } z_{min} \leq z \leq z_{max} \\ c_r(z_{min}) & \text{for } z \leq z_{min} \end{cases} \tag{4.6}$$

The mean wind velocity $v_m(z)$ for a given height combines all these influencing factors:

$$v_m(z) = c_r(z) \cdot c_o(z) \cdot v_b \tag{4.7}$$

In the implementation, we will use the recommended value of the orography factor $c_o(z) = 1$.

Wind turbulence

The second influencing factor when calculating the peak velocity pressure $q_p(z)$ is associated with the dynamic nature of the wind: it is the turbulence intensity $I_v(z)$, expressed as the ratio between the standard deviation of the turbulence σ_v and the mean wind velocity that has just been calculated (see [10] section 4.4):

$$I_v(z) = \frac{\sigma_v}{v_m(z)} \tag{4.8}$$

With

$$\sigma_v = k_r \cdot v_b \cdot k_I \tag{4.9}$$

The turbulence factor k_I is set on the recommended value of 1.

Peak velocity pressure

These intermediate values allow to characterise the wind profile to apply on the building, by mean of the peak velocity pressure:

$$q_p(z) = (1 + 7 \cdot I_v(z)) \cdot \frac{1}{2} \cdot \rho \cdot v_m^2(z) \tag{4.10}$$

Where $\rho = 1.25kg/m_3$ is the air density.

The peak velocity pressure profile can be applied on a façade in three different cases, that depends heavily on the height to width ratio for a given wind direction, as reflected in EN 1991-1-4, Figure 7.4 [10]. For instance, a wide single storey building requires that a uniform pressure is applied on its façade. On the other extreme, the velocity pressure profile varies with the height on a more slender building, to reflect the wind profile more accurately and allow a less conservative design. In this third case, the velocity profile in simplified, conservatively, in comparison with the code. The framework, however, provides guidance (Section 4.5) in case a finer adjustment is desired.

Wind pressure on the façade

The wind pressure that is applied on the surface takes into account the external pressure coefficients c_{pe}. They vary according to several zones. To evaluate the overall action on the structural shear walls, the wind pressure that is to be applied on the façade is expressed as follow:

$$w(z) = q_p(z) \cdot (c_{pe,10,E} - c_{pe,10,D}) \tag{4.11}$$

Once again, the external pressure coefficients depend on the height to width ratio h/d, as one can see in the Table 7.1 from the Eurocode 1991-1-4 [10].

Wind pressure on the roof

In a similar fashion, the Eurocode separates the roof in several zones, depending on its type (mono-pitch, pitched, vaulted, etc.) and its geometry. To each zone correspond a given wind pressure, following the same formula:

$$w(z_{roof}) = q_p(z_{roof}) \cdot c_{pe} \tag{4.12}$$

In some cases, several c_{pe} coefficients can be applied, meaning that more than one case needs to be considered. For simplicity, the calculations realised in the frame of this thesis only takes into account the highest suction or pressure. The coefficients that are to be applied for each type of roof are described in the Eurocode 1-1-4, Sections 7.2.3 to 7.2.8 [10].

Application to the Wind Load tab

- Wind load on the façade

The first part concerns the wind load that is to be applied on the façade, for both long and short sides of the edifice. To undertake simple load calculations (sections A to C.1 in the "WIND" tab), the input from the designer is kept to a minimum. The only parameters needed are the building's general geometry (height, storey height, width, length) reported from the "GENERAL" tab in the yellow cells. Moreover, the designer must specify the terrain category from the drop-down list (red cell) by following the recommendation of the Annex A from the Eurocode 1, part 1-4 [10]. In a similar fashion to the snow load 'topography', this category specifies whether the construction can get shelter from the surrounding environment. The categories go from a very exposed area (e.g. open sea) gradually to a dense area (e.g. a city). Note that the category might vary upon national decision, as it is the case in Denmark [11].

Once the input filled and/or checked, the spreadsheet calculates all the coefficients necessary to determine the peak velocity pressure $q_p(z)$ (section A), as detailed in equations 4.4 to 4.10. The section B then establishes the representative values needed to characterise the wind load profiles, according to the Eurocode 1, part 1-4 ([10], 7.7.2). The "Long side" values correspond to a cross wind acting on the largest side (Length) of the building, and the "Short side" on the smallest side (Width) (see Figure 4.1).

Finally, the section C.1. calculates the wind profiles themselves, for each direction previously defined. Building on the boundary values obtained earlier on, the tables determine the associated storey numbers and computes the peak velocity pressures $q_p(z)$ and wind pressures w_x and w_y at the said height, displayed in the first set of green cells. The second information that can be retrieved is the storey range on which each wind pressure applies, to build a more precise picture of the wind profile, as well as its translation in Grasshopper's tree selection syntax (second green cells set). As before, the green cells represent outputs that must be transferred to the "OUTPUT" tab.

- Advanced wind load profile for the façade

The results presented in section C.1. are in good concordance with the code prescriptions, when the building falls in the case 1 or 2 (see [10], 7.7.2). When the third case is considered, the profile that is generated gives out conservative values: only two steps of uniform pressure are retained, while the code allows a progressive transition from the lower to the upper profiles.

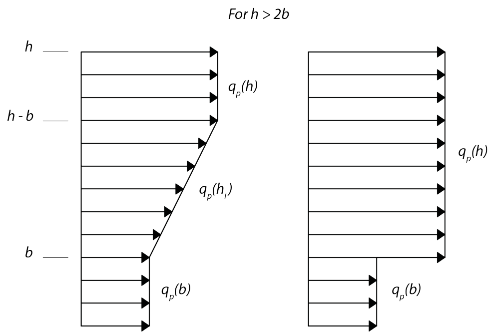


Figure 4.3: Advanced (left) and simplified (right) velocity pressure profiles

It might be in the designer's interest to fine tune the wind profile in order to optimise the structure. In this case, the section C.2. provides a template generating the wind pressure for each individual storey. Since the table varies for each specific project, it is necessary to adapt it in an adequate manner, while ensuring that the values obtained are sensible.

- Wind load on the roof

The lower section of the "WIND" tab deals with the wind loading on the roof. Supposedly, no input is needed from the designer, since the two parameters having an influence on the result, namely the roof type and the pitch angle, should have already been defined in the "GENERAL" tab. Moreover, the peak velocity pressure at the roof height has just been calculated in the previous section.

The external pressure values are referred in the corresponding tables for each zone for duopitch roofs, and for cross and along wind. Unfortunately, the zone values are not fully exploited. This decision has been made to limit the amount of load cases to be evaluated by Karamba. Therefore, only the maximum suction (negative load) and pressure (positive load) are reported in the green output cells. In this study, only the suction case is evaluated in the combinations. This is usually the most unfavourable load for the design of the roof itself.

4.6 Load combinations

The verification of the structure resistance is made using the partial factor method as prescribed by the Eurocode 0 [4]. The method considers two distinct state, namely the Ultimate Limit State (ULS) which deals with material resistance, and the Serviceability Limit State (SLS) considering the deformations and vibrations. Each of these states include a set of combination of actions that must be checked against the relevant criteria. The impact of each action is pondered by various coefficients: the consequence class factor K_{FI}, varying between 0.9 and 1.1 [4], section B3), and the ψ factors, which depend on the building occupancy and the nature of the load; they are given in the Danish National Annex, Table A1.1 DK NA [5].

Ultimate Limit States

The proposed load combinations will be checked for the "STR" Ultimate Limit State, using the factors from the Table A1.2(B+C) DK NA [5]. They combine the four actions considered in the method: dead loads, imposed loads, snow loads and wind loads. The criteria that must be fulfilled is the following:

$$E_d \leq R_d \tag{4.13}$$

Where E_d is the load combination value and R_d is the relevant design resistance.

In order to reduce the number of load combinations and the calculation time, four combinations have been chosen for the ULS:

$$E_d = K_{FI} \cdot G + 1.5 \cdot K_{FI} \cdot I + 1.5 \cdot \psi_{0,s} \cdot S + 1.5 \cdot \psi_{0,w} \cdot W \tag{4.14}$$

$$E_d = K_{FI} \cdot G + 1.5 \cdot K_{FI} \cdot S + 1.5 \cdot \psi_{0,I} \cdot I + 1.5 \cdot \psi_{0,w} \cdot W \tag{4.15}$$

$$E_d = K_{FI} \cdot G + 1.5 \cdot K_{FI} \cdot W + 1.5 \cdot \psi_{0,I} \cdot I + 1.5 \cdot \psi_{0,s} \cdot S \tag{4.16}$$

$$E_d = 0.9 \cdot G + 1.5 \cdot K_{FI} \cdot W + 1.5 \cdot \psi_{0,I} \cdot I + 1.5 \cdot \psi_{0,s} \cdot S \quad (4.17)$$

The first three equations evaluate the effect of each variable action among the imposed loads I, the snow S and the wind W as leading variable actions. The last one considers a leading wind load actions, with a favourable gravity load G: its goal is provide the maximum uplift force (tension) at the base of the CLT structural elements, due to an increased overturning moment (due to the wind) and a reduced stabilising force from the vertical loads.

Serviceability Limit States

The SLS are checked, in our case, for two main purposes: to ensure a suitable vertical deflection of the CLT slabs, and a controlled horizontal sway of the entire building.

We will use the characteristic and quasi-permanent load combinations from the Eurocode 0; Section 6.5.3. [4]. The characteristic combinations are expressed by the following equations:

$$E_d = G + I + \cdot\psi_{0,s} \cdot S + \cdot\psi_{0,w} \cdot W \quad (4.18)$$

$$E_d = G + W + \cdot\psi_{0,I} \cdot I \cdot \psi_{0,s} \cdot S \quad (4.19)$$

They are used to determine the instantaneous deflection w_{inst} for the roof and floors, as well as the maximum horizontal sway (2nd equation).

The quasi-permanent combination is used to calculate the final deflection w_{fin} for the roof and the floors, and is expressed as:

$$E_d = G + \cdot\psi_{2,I} I + \cdot\psi_{2,s} \cdot S + \cdot\psi_{2,w} \cdot W \quad (4.20)$$

Load outputs in Excel

The "OUTPUT" tab gives out the final output for the structural analysis in Karamba. The sheet is split in 4 main corpus, corresponding to each load type considered in the frame of this thesis. As explained in the main report, Section 5.5, partial factors and load values are separated to provide a better control over the results. Only the characteristic values of the loads are applied in the first time, and the combinations are created as a post-processing input. Examples for each of the following sections are illustrated in the "Appendices" book, Appendix B.2; the reader is invited to consult both documents in parallel.

- Dead Loads

The first section (A) lists all the partial factors which will be used for the load combinations for Dead Loads. These include the additional dead load induced by the floor and roof assemblies defined on the "DEAD LOADS" tab. The partial factors are prefilled for the load combinations as described in Section 4.6.

The characteristic values of these loads are listed in the table on the right. For each type of load, a column must be created which specifies the function of the element it is applied to (Floors or Walls), their *Genotype* number in the model, and the value of the load which must be applied. The value must be reported from the corresponding value in the "DEAD LOAD" tab. The green cells (output) must be then retrieved in the parametric environment with the GhExcel plug-in.

- Imposed loads on floors

The section B concerns the imposed loads that are to be applied on floors. Depending on the occupancy, the partial factors for these loads vary (cf. Section 4.3) due to different ψ factors. The left table lists the product of the partial factor $\gamma \cdot K_{FI}$, which is constant, as well as each adequate ψ factor for each load combination. The floor type to which they apply must be reported in the red cell. Once again, one new column must be created for every different imposed load type.

The table on the right keeps this philosophy: to each columns corresponds one load. These columns lists the floor type which is concerned, as defined in the Grasshopper model, the characteristic value of the load and the final partial factor which will be applied for the load combinations, i.e. the product $\gamma \cdot K_{FI} \cdot \psi$. The entire table (red and green cells) must be retrieved in Grasshopper.

- Snow Loads

The section (C) deals with the snow loading, typically applied on the roof. Similarly, detailed partial factors are listed on the left table, and the floor type to which the loads are applied, along with their characteristic values and final factors for each combination are compiled in the right table, which constitutes the output to Grasshopper. Snow loads coefficients are independent from the project parameters; there is therefore no need to modify the default values. The designer must however specify the type attributed to the roof, and retrieve the snow load value from the "SNOW" tab.

- Wind Loads

Finally, the last section (D) reports the wind load factors, in the same fashion as previously, as well as its different components, i.e. the wind load on the roof, on the "Long side" of the façade and on the "Short side" of the façade, on the right table. The elements to which each of these are applied must be transferred from the "Wind Loads on Roofs" section and the table C.1. of the "WIND" sheet respectively. The green cells (output) are the ones supposed to be transferred to the model, as well as the final factors from the left table.

5 Preparation for the structural analysis in Karamba

At this stage, all the information which is necessary for the structural calculations is available: Geometry, Materials, and Loads. It must be conditioned in the right way to allow Karamba to assemble the model, and perform the analysis.

5.1 Geometry input to Karamba

Geometry meshing

In order to perform finite element calculations with Karamba, the current geometry, slabs and walls, must be meshed. For this step, the *Mesher* component can be used.

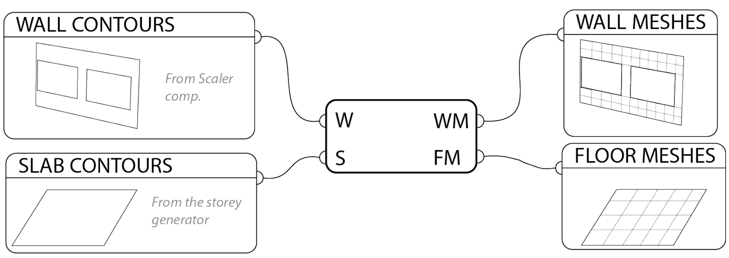


Figure 5.1: The Mesher component

The component must be fed with the walls and slab surfaces, as obtained from the end of Section 1. These surfaces will help selecting the connecting points between walls and slabs.

Indeed, each element must be rigidly connected to each other; Karamba makes this association when two different meshes share identical points. The *Mesher* component therefore ensures that walls and shells are meshed with common points, connecting each element with its neighbours.

Mesh conversion

The meshes that result from the *Mesher* component will serve as an input to create the shell elements, representing the structural walls and floors in the model. They need to be declared as such for Karamba. The framework features the *Karamba Elements* component to realise this action. One just needs to plug the respective meshes directly from the *Mesher* component to obtain, as outputs, the converted Karamba elements. The component also gives out the model points, which will be used to obtain the horizontal sway (cf. Section 6.2), and the number of walls and floors. As Karamba input must be flattened, this last data can be used to retrieve each separate list and reconstruct the initial data structure for further manipulation.

Figure 5.2: Karamba Elements component

5.2 Support input to Karamba

As for most FEM software, Karamba needs to know where the building's supports are located. The *Karamba Supports* component let the designer select two types of foundation: either a line foundation, or a mat foundation. Simply toggle the boolean input to choose between them. The supports, as needed for the Karamba model, are given as an output.

Figure 5.3: Karamba Supports component

5.3 Material input to Karamba

In a similar fashion, the cross section defined in the "Layups" Excel sheet must be converted to Karamba materials and cross sections. For this purpose, one shall take the "Layups", "Stiff." and "Strength" outputs from the "LayupXL" component, and plug them in the respective *CLT Materials* component inputs. The user then has to define the desired cross section index, which must match the index in the "CS List" output from the "LayupXL" component. Technically, one *CLT Material* component is needed for each layup that has been defined in the excel sheet.

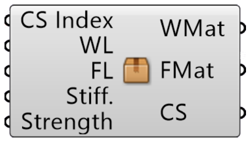


Figure 5.4: CLT Material component

The component creates the corresponding cross section (via the Karamba Cross Section component) at the right thickness, as well as the material for both floors and walls.

The material for walls is defined by the design Equivalent In-Plane Elastic Modulus $E_{eq,IP}$, which serves as the equivalent isotropic Elastic Modulus E of the Karamba material component. As Karamba does not handle orthotropic materials, the equivalent shear modulus G^*_{IP} can however not be used for this purpose, as the Shear Modulus G must be comprised between E/3 and E/2. The CLT shear modulus being more of a E/10 order, $E_{eq,IP}/2$ is used instead to approach a Poisson ratio $\nu = 0$, as explained in the Main Report, Section 5.2. The design tensile strength $f_{t,0,CLT,net,d}$ is used for the material yield strength, as well as the characteristic mean density $\rho_{CLT,mean}$.

The same principle is implemented for the floor material, using the design Equivalent Out-of-Plane Elastic Modulus $E_{eq,OP,d}$ (divided by 2 for the shear modulus) and the design bending strength $f_{m,CLT,d}$ as yield strength.

The outputs, *WMat* (for wall materials) and *FMat* (for floor materials), must be assembled in a Karamba material container, and will serve as an input to the model. Similarly, the output *CS* (for Cross Section) aliments the Cross Section input for Karamba.

5.4 Load input to Karamba

The loads that are to be applied on the model have been defined in the "Loads" sheet. The data contained in the "OUTPUT" tab will serve, once transformed, as input for the different load cases to be analysed. To retrieve these values, the 8 outputs from the said tab, respectively the Dead Loads, Imposed Loads, Snow Loads and Wind Loads characteristic values and factors must be separately read by the GhExcel component, creating 8 outputs in total. As explained in the Main Report, Section 5.5, the load combinations are performed after the analysis. Therefore, only the characteristic values are given to Karamba, and will define at least 4 load cases. It is suggested to use 0 for the dead loads, 1 for the snow, 2 for the wind, and 3+ for the imposed loads. The latest type of loads indeed possess different factors for each of the occupancy types. For example, in the Appendix B.2, they are defined for regular floors and the roof, which will occupy the load cases 3 and 4 respectively.

Gravity loads

The gravity load is calculated by Karamba on the basis of the model materials. It must be added to the Load input to Karamba. To this end, the user must place the Karamba *Load* component and input a vector value of $-z$. A load case must then be attributed; it is recommended to set it to 0.

Load components

Five load components have been created to prepare the load input for Karamba.

Figure 5.5: Load components - From left to right: Dead Load, Snow Load, Wind Load on Roof, Wind Load and Live Load

The *Dead Load* component is meant for Dead Loads which will be applied to walls and floors. For each type of dead load (i.e. each column of the Dead Load table from the "OUTPUT" tab), one *Dead Load* component must be used. The following inputs are needed:

- DL (for Dead Load) represents the Dead Load table from the "Output" tab
- DL i (Dead Load index) allows the user to select which specific load to apply
- WM and FM are the Wall and Floor Meshes respectively, from the *Mesher* component
- WL and FL are the Wall and Floor Lists specifying the attributed types

The Dead Load retrieves the right elements from the Load input to create the load, which must be connected to Karamba. By default, the load case is set on 0.

The *Snow Load* component applies snow load on the roof. Similarly, it requires the Snow Load (SL) to be inputted, as well as the Floor Mesh (FM) and Floor List (FL) since loads are applied on this type of elements. The Snow Load index can be used in the situation where more that one snow load needs to be applied. By default, snow loads are defined by the Load Case 1 in the Karamba model.

The *Wind Load Roof* must be used to define the wind action on the roof. Since the roof is defined in the Floor Meshes (FM), this input must be given as well as te Floor List. Moreover, the Wind Load (WL) from the "OUTPUT" tab must be provided. Normally, the field "Applies to" from this table should correspond to the floor type assigned to the roof, which ensures the application of the load on the right elements. Once again, the load index is implemented in case where more than one load should be defined, for instance in a future implementation of the Framework which could make use of the zone pressures (cf. Section 4.5). Wind Loads are defined by the Load Case 2.

The *Wind Load* component is, on the other hand, dedicated to the wind load acting on the façade. The Wind Load (WL) must be informed to specify the load that must be applied, and the number of the first column (WL i) that is associated in the "OUTPUT" tab as well; the component considers this column and the two next to it which corresponds to the three phases of the wind profile. The specificity of this load component is that it does not act on the Wall geometries. Indeed, the walls defined for the building have a reduced area due to openings, which is not the case on a real façade. Moreover, the load that is generated is based on the area of the meshes. Therefore, the *Frames* surfaces (FR.) are used instead. The ones corresponding to the desired façade must be identified in the "Fac. ID" input, using the data structure as presented in Section 1.2. Finally, the direction of the wind, in degrees, must be specified in the "Dir" input, with 0 corresponding to a y direction. Once again, this load is placed under the Load Case 2.

The last load component is the *Live Load* component corresponding to the Imposed Load definition. As explained previously, imposed loads are combined with different factors depending on their nature. Therefore, for each load category, one *Live Load* component must be used. It must be fed with the Live Load (LL) from the "OUTPUT" sheet, whose corresponding column is selected by means of the Live Load index (LL i). The Floor Meshes (FM) and Floor List (FL) will be used to retrieve the surfaces subjected to the loads, as specified in the "type" cell from the load table ("OUTPUT" tab). These loads correspond to the Load Case "3+" in the Karamba model.

All the load outputs from these components must be joined and given as input to the Karamba *Assemble Model* component.

5.5 Model generation and analysis

Assembling of the Karamba model

When each input has been defined, the user should be able to gather the "Element", "Supports", "Loads", "Material" and "Cross Section" data that need to be plugged in the "Assemble Model" component from Karamba. One shall place each of them in their respective Karamba containers, and most importantly Flatten all the data. When this is done, simply plug each container to the respective input. The model is generated and ready for analysis.

First order analysis

If everything has been done successfully, the model should be ready for analysis. Simply input the model to the "AnalyzeTHI" component from Karamba; this performs a first order analysis of the structure. All the data is now ready to pursue the design of CLT elements.

6 Results

6.1 Model validation

The first action that designers are advised to perform is to check the consistency of their model. To this end, one can use the "Model View" as well as the "Shell View" component, both from the Karamba plug-in. These components provide a visualisation of deformations, loads, supports and utilisation for the model elements. Although this information is not accurate, due to the fact that load combinations are not implemented yet, and that the calculation of the utilisation of CLT elements is more complex, this allows to ensure that the loads have been correctly applied and that the model is behaving accordingly. This includes the possibility to check if every element is connected as intended.

After this first check, the user can proceed to more detailed results, through the components that were developed for the framework.

6.2 Horizontal sway

The *Horizontal Sway* component allows the user to check the horizontal stability of the building. As inputs, it needs the analysed model from Karamba as well as the model points, from the *Karamba Elements* component. The aim for these is to find the highest point of the building and to retrieve its horizontal displacement. The second set of inputs consist in the load factors for each load: dead loads (DL), live load (IL) (or imposed load), snow load (SL) and wind load (WL). These factors will combine the displacements from each Karamba load case according to the load combinations defined in Section 4.6.

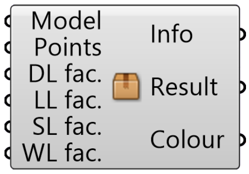


Figure 6.1: Horizontal Sway component

The criteria of the analysis are available by means of the "Info" output. It informs the user of the maximum displacement that is allowed; this is bound by the ratio $H/500$ where H is the building height. Advice is given on the process to follow if the design does not comply with it.

The "Result" output contains the results from the calculation, i.e. the horizontal displacement for each load combination with the numbering set by the "OUTPUT" tab from the load excel sheet. The results must be compared to the maximum displacement, for each SLS combination; the ULS combination are not concerned.

Finally, the "Colour" output provides a visual feedback (red or green) that can be displayed on the geometry.

Example on the visual and textual feedback provided by this component are detailed in the Main Report, Section 6.3.1.

6.3 Wall Design

The *Wall Design* component ensures the function to verify the design of a chosen wall. It requires a considerable amount of inputs which are detailed below:

- Model - The model from the analysis
- W ID (Wall ID) - The wall that needs to be designed. For this purpose, the identification path of the wall must be used, which corresponds to the structure as presented in Section 1.2; For example, "A;B;C"
- WL (Wall List) - The list of wall types which specifies which Genotype is associated to each wall
- Lay., Stiff. and Str., which are the layup, stiffness and strength outputs of the *LayupXL* component
- DL, SL, WL, LL fac. which are the partial factors for dead loads, snow loads, wind loads and live loads respectively
- L Case - Load case - selects one desired load combination
- Scale - this input, Boolean, allows the user to choose between two scales for the visual feedback. Set to "True", the results are displayed on a scale from 0 to 100 %. If set to "False", the range between the minimum and maximum value is displayed
- Load bear. for "Load bearing"; setting this factor to "True" will multiply the loads by 1.5, as explained in the Main Report, Section 5.5.
- Wind safe. for "Wind safety factor" allows the user to choose whether to apply the wind load safety factor or not, c.f. Main Report, Section 5.4.3.
- Tension Safety consists in a boolean input which complements the Wind safety factor. Set to "True", it cancels the Imposed Loads to help the verification of walls in tension, due to the wind (cf. Main Report, Section 5.5).

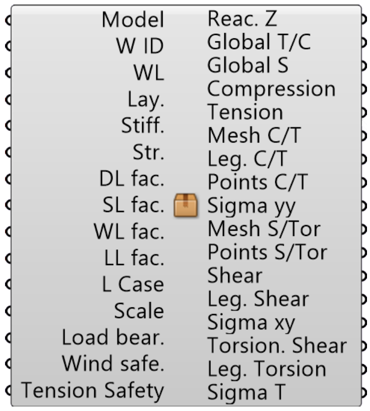


Figure 6.2: Wall design component

All this data is necessary to perform the calculations as explained in the main report, Section 4.4. The component provides the verification of the wall element for tension, compression, shear in plane and torsion at the interface. The outputs bearing the same names inform the user on the maximum stresses and compares it to the maximum strength, providing the user with advice; more details in the Main Report, Section 6.3.2. The other outputs must be used to display the results; they account for three types:

- Mesh and Leg. (Legend) - can be coupled to a geometry preview to obtain stress maps
- Points and Sigma - can be coupled to a Tag component to display stress values at the finite element centres

Sigma yy represent the tension/compression stresses (tension > 0, compression < 0), Sigma xy the in-plane shear stresses and Sigma T the torsional stresses.

Moreover, some additional information are provided with the three first outputs. "Reac. Z" retrieves the vertical reactions at the supports, "Global T/C" calculates the global tension or compression force in kN/m acting on the wall, and "Global S" the global shear force in kN/m. An example of these complementary outputs can be seen in the Main Report, Figure 6.3.5.

6.4 Slab design

The *Slab design* component is used for the design of slabs. In terms of outputs, it is fairly similar to the *Wall design* component, with a few exceptions:

- FS - Floor Surfaces - consists in the floor surfaces from the geometry
- F ID - Floor ID - allows the selection of a given floor to design; it must respect the floor structure as explained section 1.2, for example "A;B"
- FL - Floor List: the list of floors sorted by types, similar to the wall list
- DL, LL, SL, WL are the Dead, Live (Imposed), Snow and Wind loads from the *Layup XL* component. They need to be fed to the component since the slab is evaluated in a sub-model, as presented in the Main Report, Section 5.5.

All the remaining inputs, i.e. DL, LL, SL and WL factors are identical to the *Wall Design* component, as well as the material properties input (Lay., Stiff., Str.), the scale and the chosen load combination (LCase).

Figure 6.3: Slab design component

Calculations are performed according to the Main Report, Section 4.4, and the available range of outputs is detailed in the Main Report, Section 6.3.3. As for the *Wall Design* component, the *Slab Design* components provides the user with "Mesh" and "Leg." that can be coupled with a geometry preview, as well as "Points" and stress values that can be displayed at the finite element centres. The latter are Sigma M (bending stresses) and Tau V/R which are either transverse or rolling shear (cf. Main Report, Section 4.4).

The textual outputs, stating whether the criteria is fulfilled or not, can be obtained for displacements, bending stresses, transverse and rolling shear via the outputs bearing the same name.

7 Concluding remarks

The methodology and components that are developed in this manual organise a parametric framework, with the goal to make CLT structural design as accessible as possible in the early design stages. The multitude of components describe a workflow starting with the definition of the geometry, and ending with the verification of horizontal stability, walls and slabs, with a special care to provide a seamless experience.

Bibliography

[1] Stora Enso. *Stora Enso CLT - Wood: the world's oldest and yet most modern building material*. 2016.

[2] R Brandner et al. "Cross laminated timber (CLT): overview and development". In: *European Journal of Wood and Wood Products* 74.3 (2016), pp. 331–351.

[3] *Timber structures – Cross laminated timber – Requirements*. Standard. European Commitee for Standardization, 2015.

[4] *Eurocode - Basis of Structural Design*. Standard. European Commitee for Standardization, 2010.

[5] *National Annex to Eurocode: Basis of structural design*. Standard. Danish Energy Agency, 2013.

[6] *Eurocode 1: Actions on structures — Part 1-1: General actions — Densities, self-weight, imposed loads for buildings*. Standard. European Commitee for Standardization, 2009.

[7] *National Annex to Eurocode 1: Actions on structures - Part 1-1: General actions - Densities, self-weight, imposed loads for buildings*. Standard. Danish Energy Agency, 2013.

[8] *Eurocode 1 — Actions on structures — Part 1-3: General actions — Snow loads*. Standard. European Commitee for Standardization, 2009.

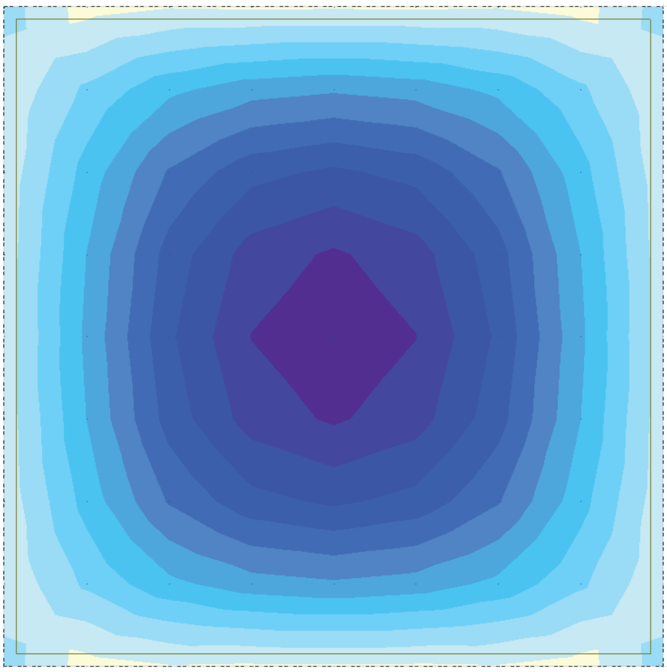
[9] *National Annex to Eurocode 1: Actions on structures - Part 1-3: General actions - Snow loads*. Standard. Danish Energy Agency, 2015.

[10] *Eurocode 1: Actions on structures — Part 1-4: General actions — Wind actions*. Standard. European Commitee for Standardization, 2009.

[11] *National Annex to Eurocode 1: Actions on structures - Part 1-4: General actions - Wind actions*. Standard. Danish Energy Agency, 2015.

Modular Timber Structures with Parametric Design Frameworks

BOOK 3 - Appendices



Master Thesis Project

Simon Clavelin

June 2017

Technical University of Denmark
Department of Civil Engineering
Brovej, Building 118
2800 Kgs. Lyngby
Denmark

Author:

Simon Clavelin
s151369

Supervision:

Kristoffer Negendahl
Department of Civil Engineering

Preface

This book forms the third part of the Master Thesis entitled "Modular Timber Structures with Parametric Design Frameworks", submitted as partial fulfilment of the requirements to the obtention of the degree of Master of Science in Architectural Engineering, from the Technical University of Denmark.

It contains additional information which complement the Main Report.

Kongens Lyngby, June 23rd, 2017

Simon Clavelin

Contents

Appendix A.1 - Floor results

This appendix presents additional results from the simulation introduced in the Main Report, Section 5.3.1. It aims at characterising the discrepancies between an orthotropic and isotropic modelling of CLT slabs, in terms of displacement, moments and shear, depending on the length-to-width ratio of the slab.

Displacements - Detailed results

l/w	δ_{iso} (mm)	$\delta_{iso,tot}$ (mm)	δ_{ortho} (mm)	Δ
1	2.03	2.59	6.06	57 %
1.5	3.79	4.35	7.30	40 %
2	4.92	5.48	7.21	24 %
3	5.35	5.91	6.75	12 %
4	6.10	6.66	6.73	1 %
8	6.17	6.73	6.71	0.3 %

Horizontal deflection and error percentage for the orthotropic and isotropic plates

Bending moments - Detailed results

l/w	M_{iso} (kNm/m)	M_{ortho} (kNm/m)	Δ
1	3.0	8.6	65 %
1.5	6.1	10.9	44 %
2	7.9	10.8	26 %
3	9.5	10.4	9 %
4	9.9	10.4	5.5 %
8	10.0	10.4	4.2 %

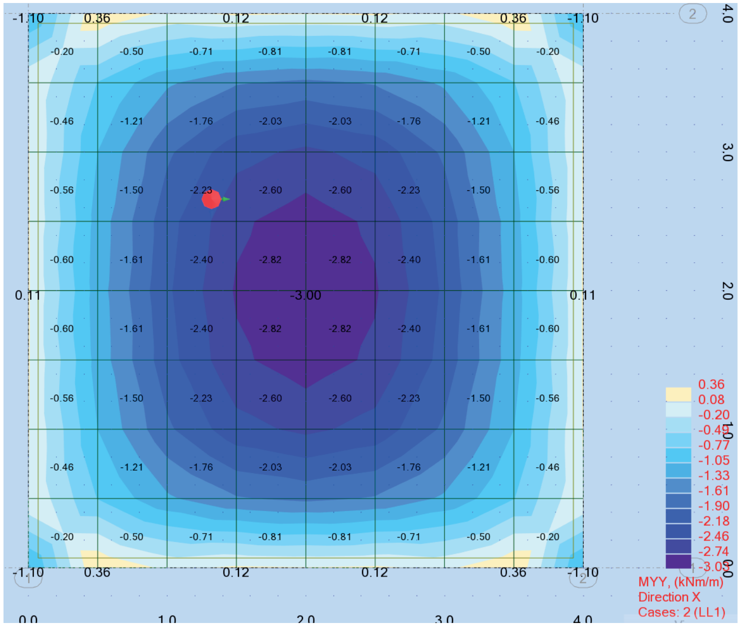
Bending moments along the width and error percentage for ortho and isotropic plates

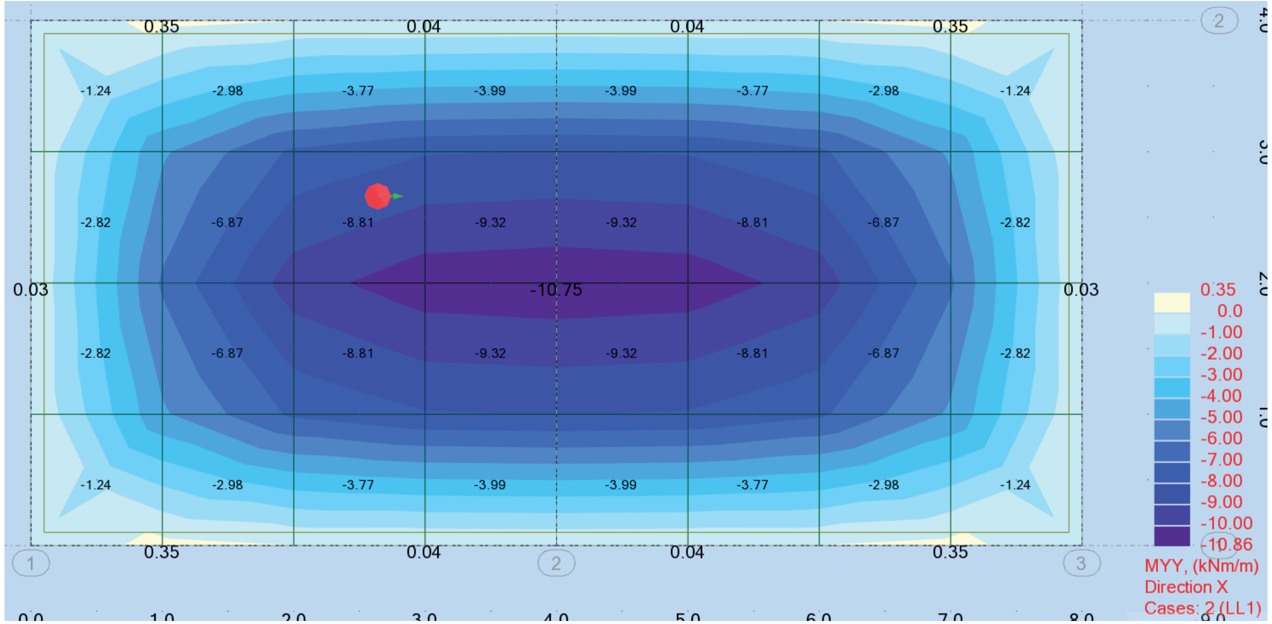
MOMENT MAPS

l/w = 1

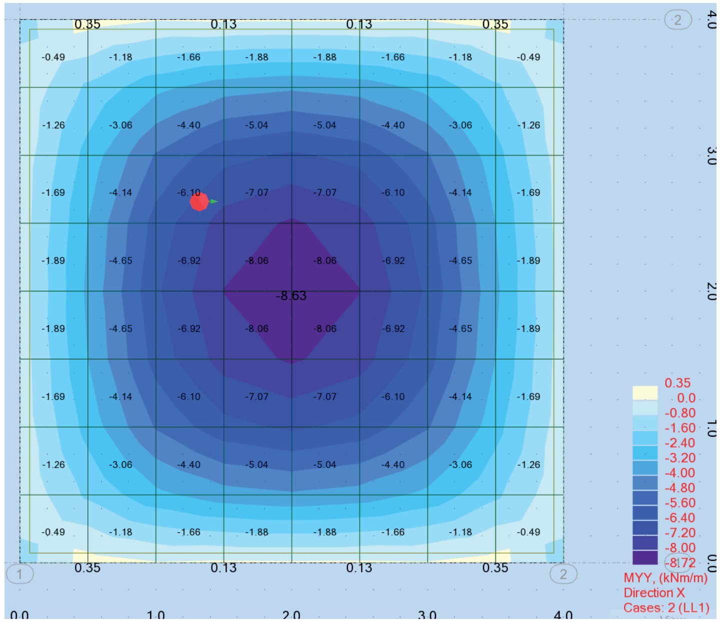
l/w = 2

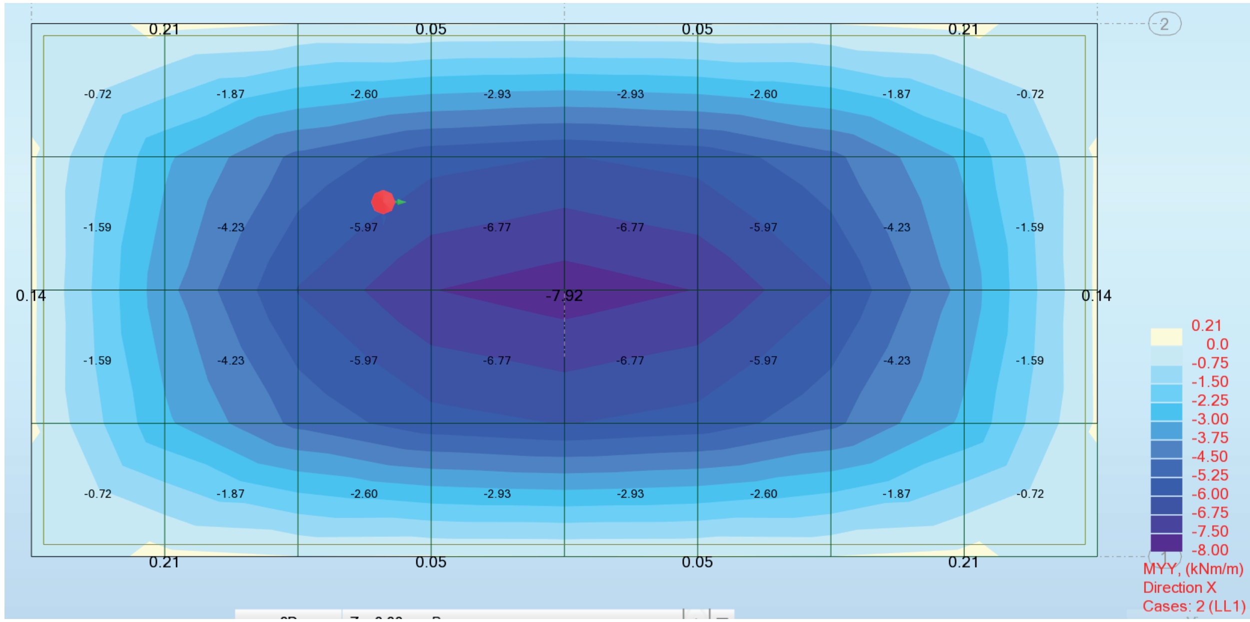
ORTHOTROPIC





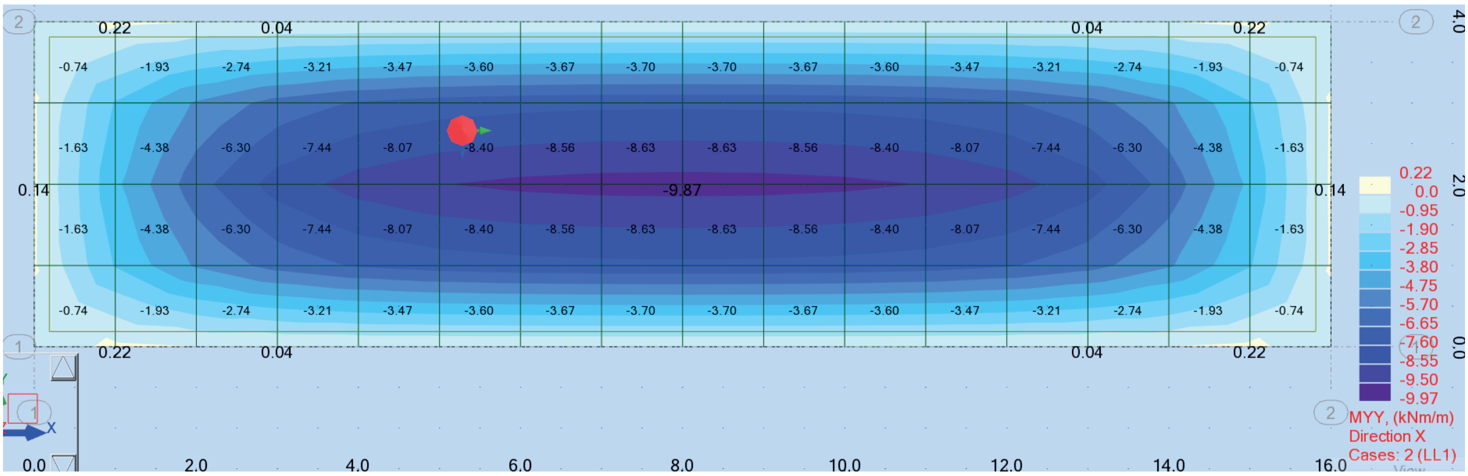
ISOTROPIC



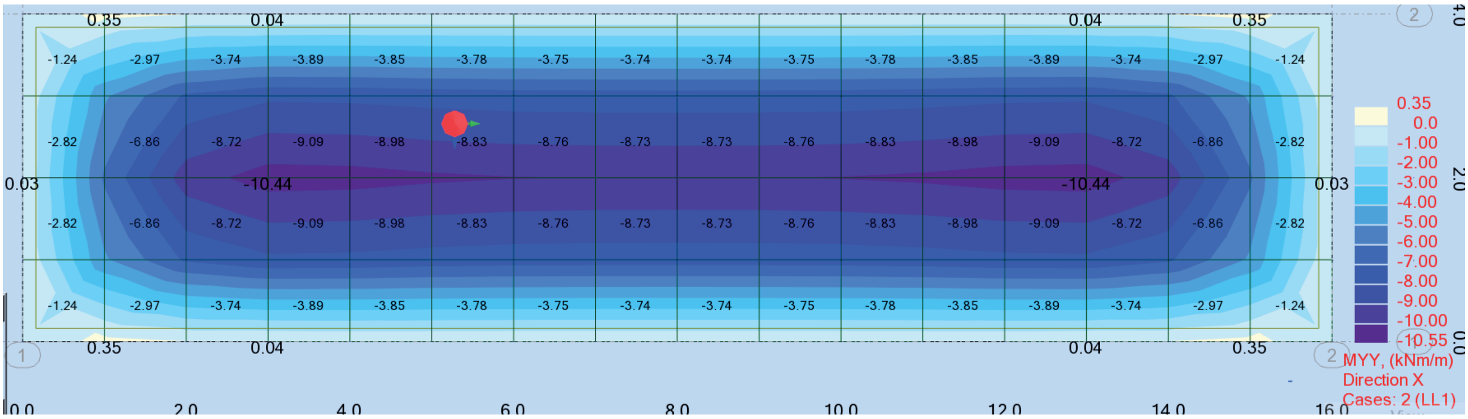


l/w = 4

Isotropic slab



Orthotropic slab



Shear forces - Detailed Results

l/w	Q_{iso} (kNm/m)	Q_{ortho} (kNm/m)	Δ
1	3.7	8.2	54 %
1.5	4.6	8.3	41 %
2	6.1	8.0	24 %
3	7.1	7.9	9 %
4	7.4	7.9	5.8 %
8	7.5	7.9	4.8 %

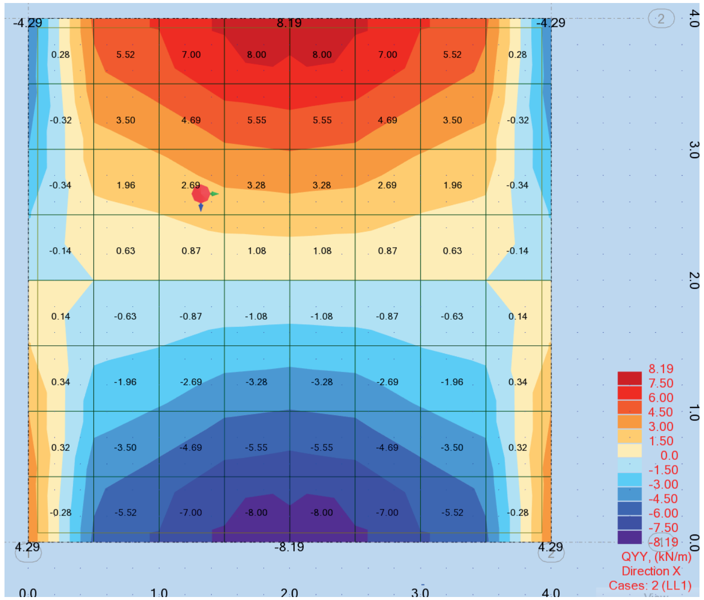
Transverse shear force at supports (along length) and error percentage for ortho and isotropic plates

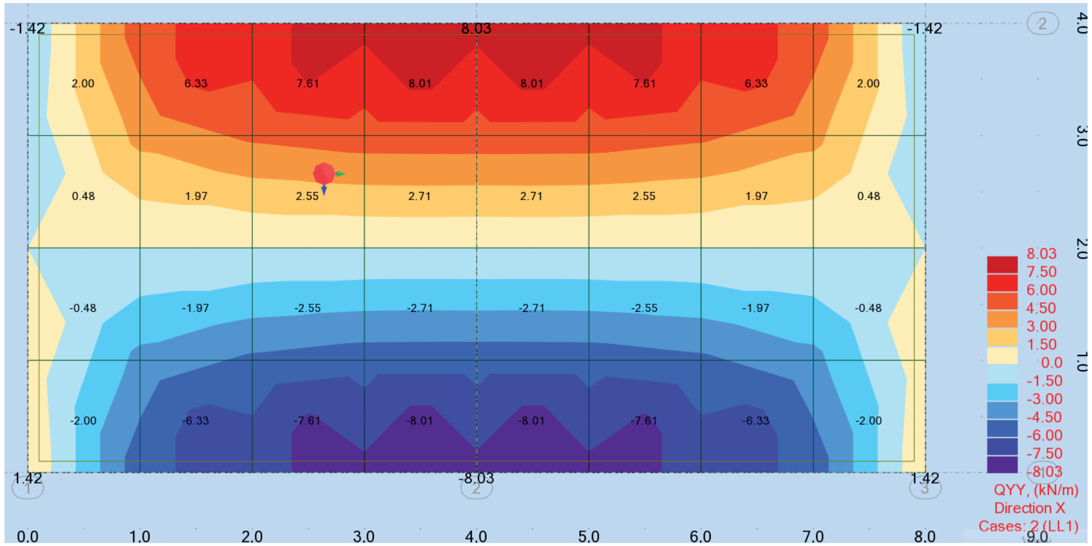
TRANSVERSE SHEAR MAPS

l/w = 1

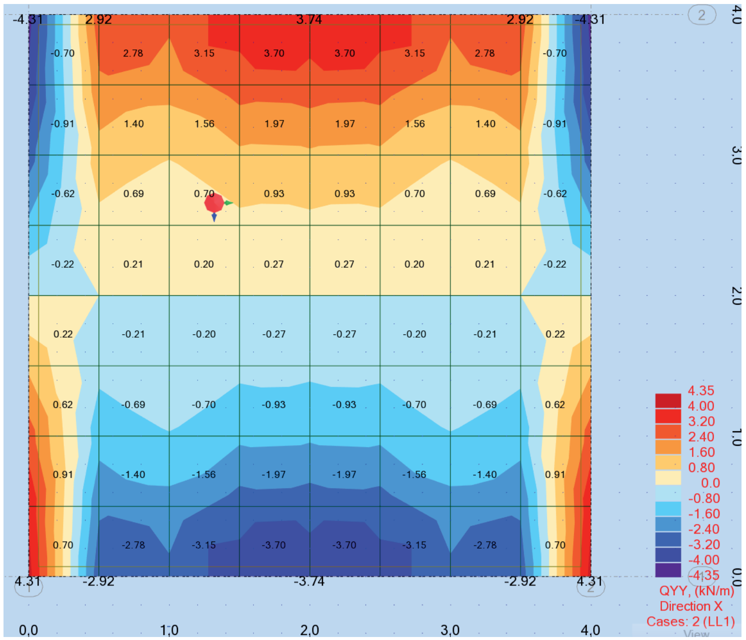
l/w = 2

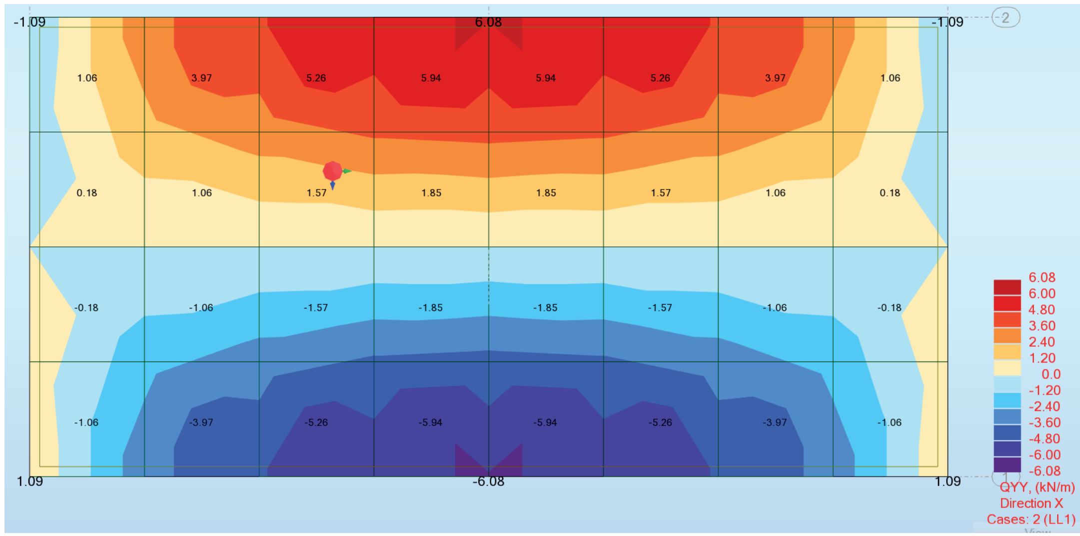
ORTHOTROPIC





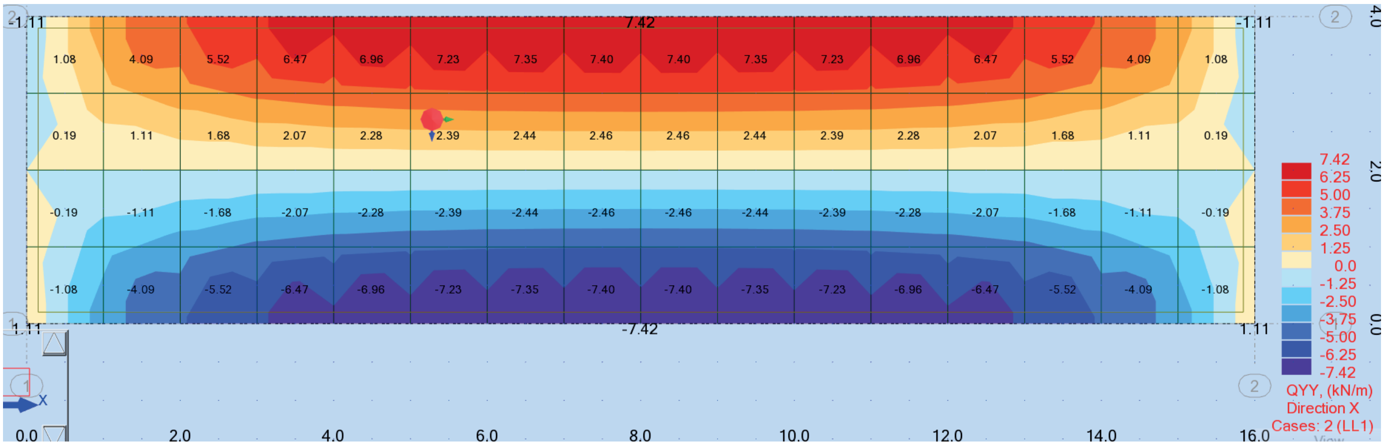
ISOTROPIC



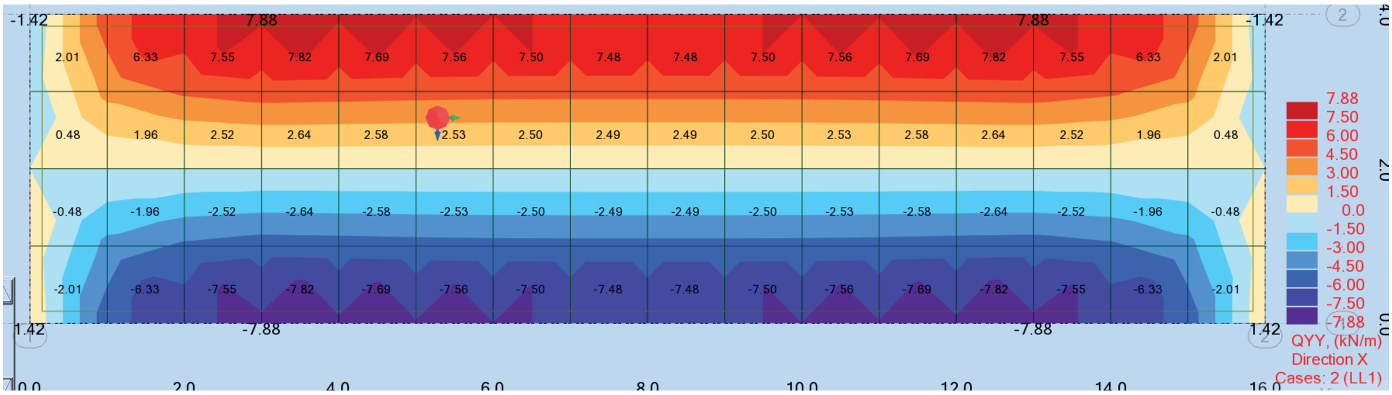


l/w = 4

Isotropic slab



Orthotropic slab



Appendix A.2 - Load repartition results

This appendix refers to the simulations carried out and detailed in the Main Report, Section 5.3.2. It evaluated the fluctuations between the load distributed by an orthotropic slab and an isotropic slab, against the varying cross bending stiffness ratio K_y/K_x. Detailed results are available in the table below.

Cross Section	$\boldsymbol{K_x}$ kNm^2/m	$\boldsymbol{K_y}$ kNm^2/m	$\frac{\boldsymbol{K_y}}{\boldsymbol{K_x}}$	$\%_{\boldsymbol{Ortho}}$	γ
160 L5s-2	4200	66.7	0.02	36%	1.44
120 L3s	1733	66.7	0.04	35%	1.41
90 3x30	675.7	26	0.04	36%	1.44
120 L5s	1583	216.7	0.14	32%	1.30
150 5x30	2572.8	675.7	0.26	30%	1.18
210 7x30	6341	2572.8	0.41	28%	1.11
200 L7s	4533	3800	0.84	25%	1.00

Load distribution on the most critical wall for the orthotropic model

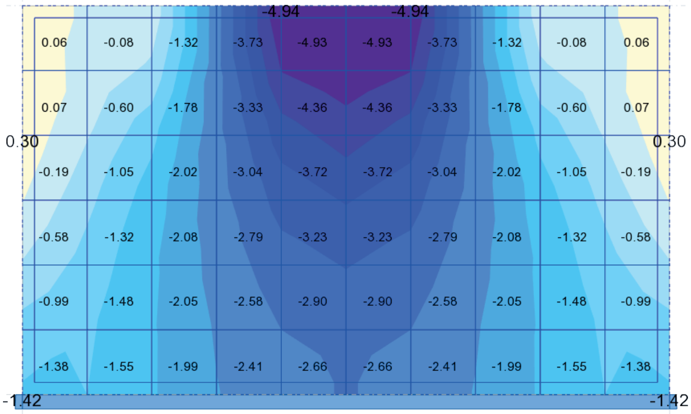
Appendix A.3 - Force distribution in walls

The results relate to the simulations explained in the Main Report, section 6.4. The first set of maps corresponds to the evolution of vertical and shear forces in a panel, with a varying ratio D_y/D_x. The second one varies the ratio $D_{xy,ortho}/D_{xy,iso}$. Finally, a last set of results corresponds to the additional simulations carried out by evaluating the combined stiffness ratio of the wall D_{ortho}/D_{iso}. Tabulated results are given to the evolution of the vertical force in the panels.

Tabulated results are given for the evolution of the vertical force in the panels.

VERTICAL MEMBRANE FORCES - FROM THE VERTICAL LOAD

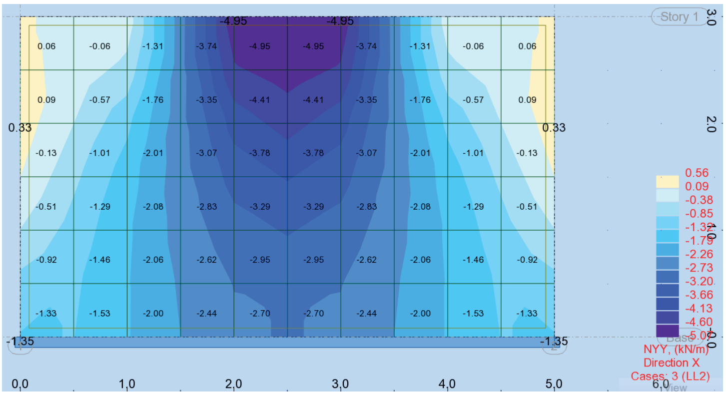
Isotropic Wall



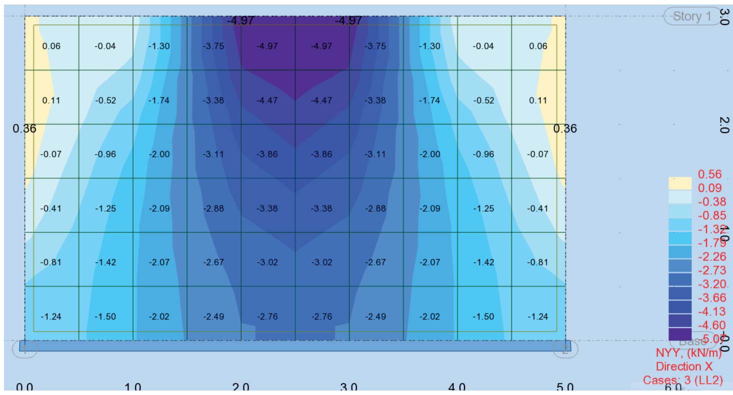
Model	D_x	D_y	D_y/D_x	Max. vertical force	$N_{max,o}$ / $N_{max,i}$
Isotropic	1039500	1039500	1	4.94	
Orthotropic	1039500	1039500	1	4.94	1.000
	1039500	831600	0.8	4.95	1.002
	1039500	623700	0.6	4.97	1.006
	1039500	415800	0.4	5	1.012
	1039500	311850	0.3	5.01	1.014
	1039500	207900	0.2	5.03	1.018
	1039500	155925	0.15	5.04	1.020
	1039500	103950	0.1	5.07	1.026

Orthotropic results for varying Dy/Dx

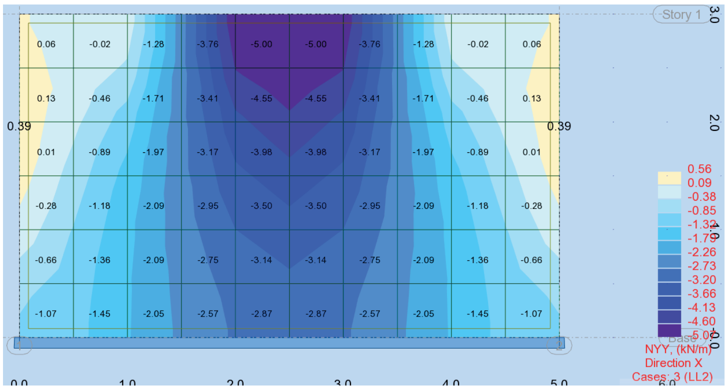
Dy/Dx = 0.8



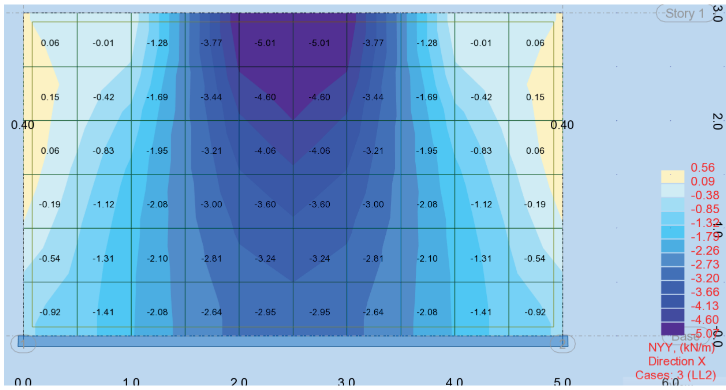
Dy/Dx = 0.6



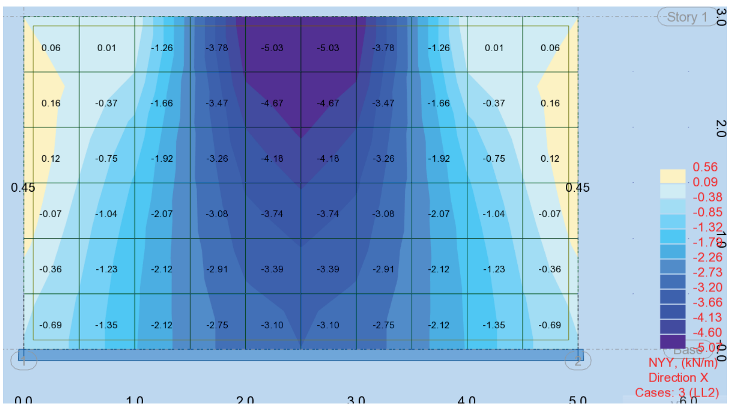
Dy/Dx = 0.4



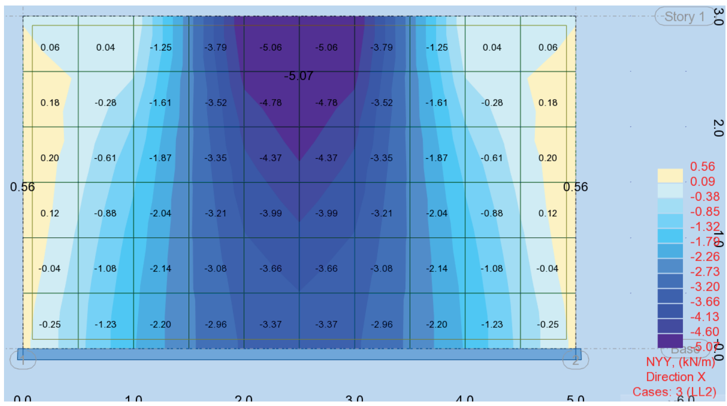
Dy/Dx = 0.3



Dy/Dx = 0.2

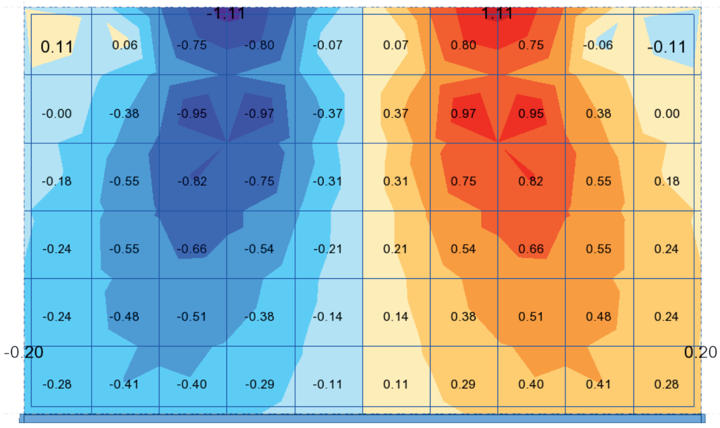


Dy/Dx = 0.1



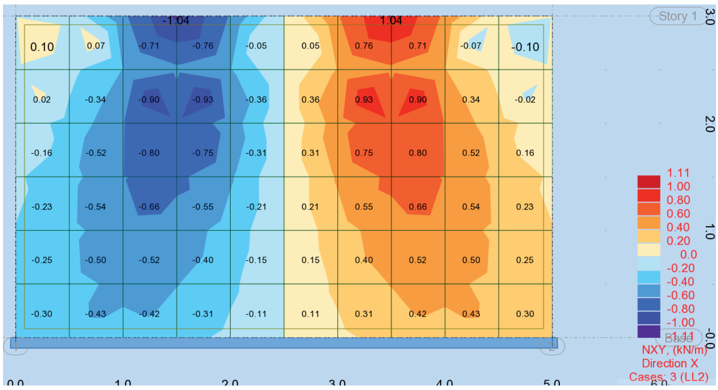
SHEAR MEMBRANE FORCES - FROM THE VERTICAL LOAD

Isotropic Wall

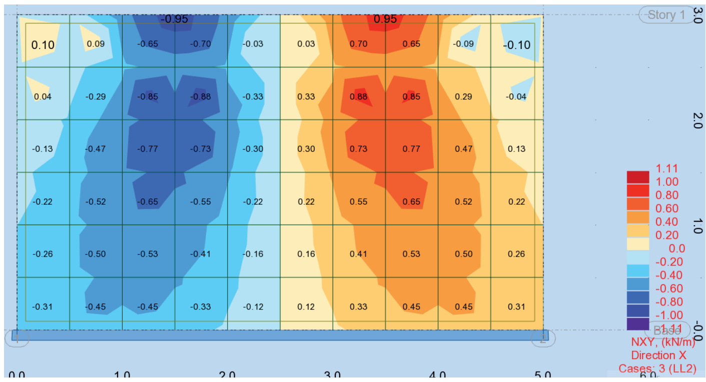


Orthotropic results for varying Dy/Dx

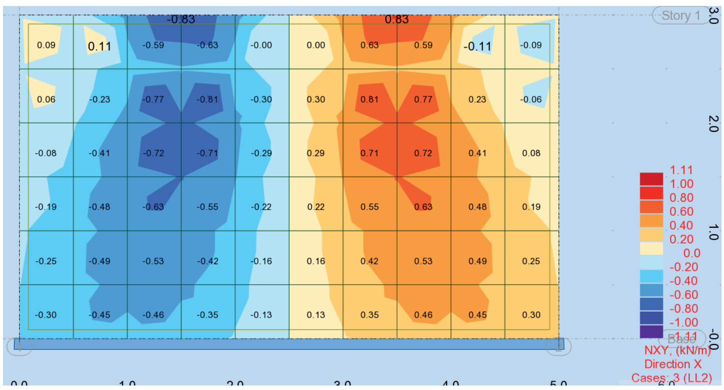
Dy/Dx = 0.8



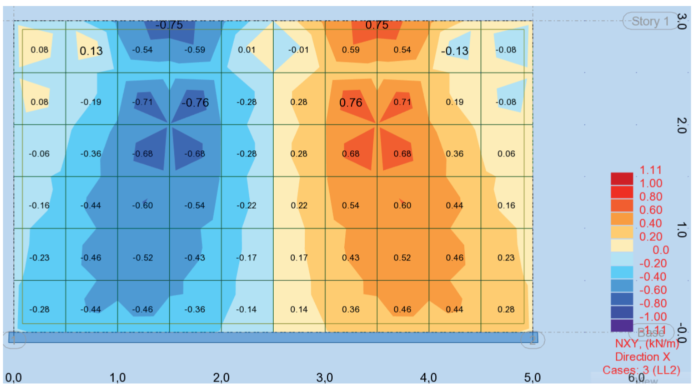
Dy/Dx = 0.6



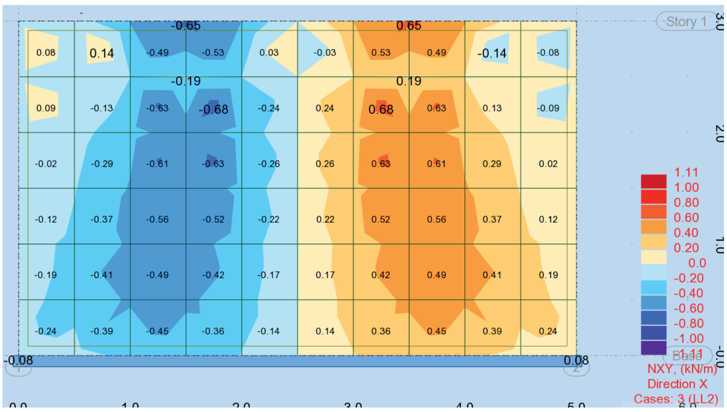
Dy/Dx = 0.4



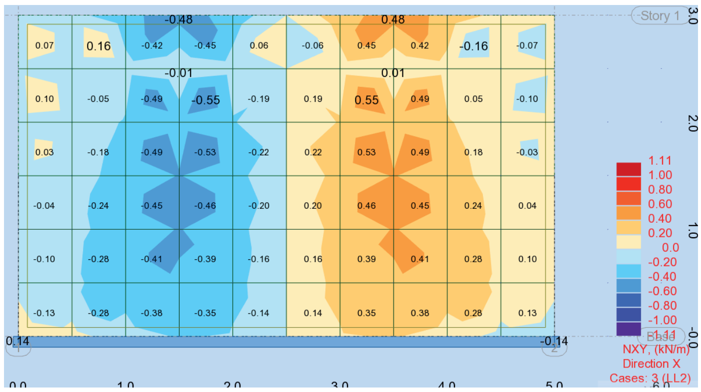
Dy/Dx = 0.3



Dy/Dx = 0.2

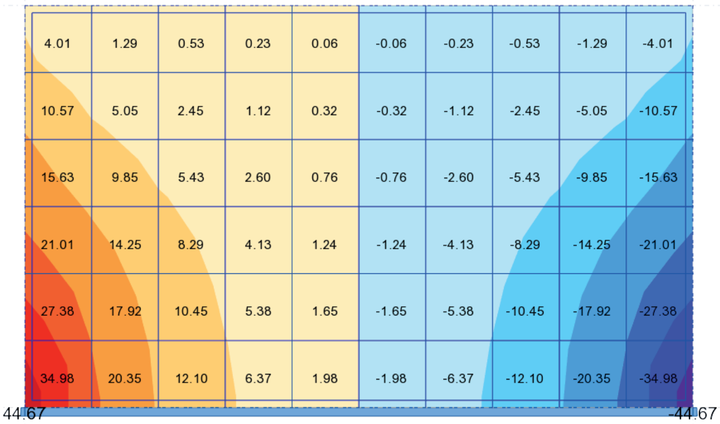


Dy/Dx = 0.1



VERTICAL MEMBRANE FORCES - FROM THE HORIZONTAL LOAD

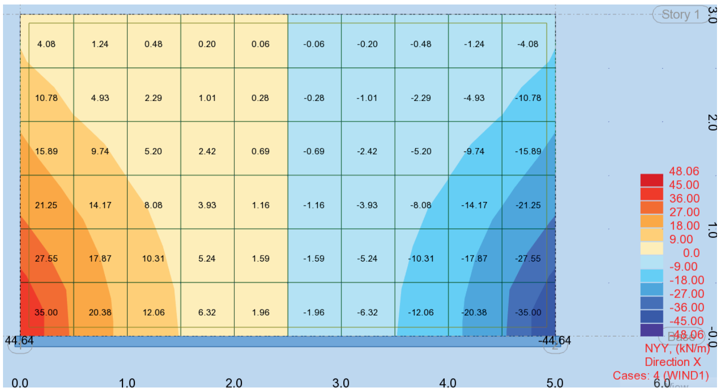
Isotropic Wall



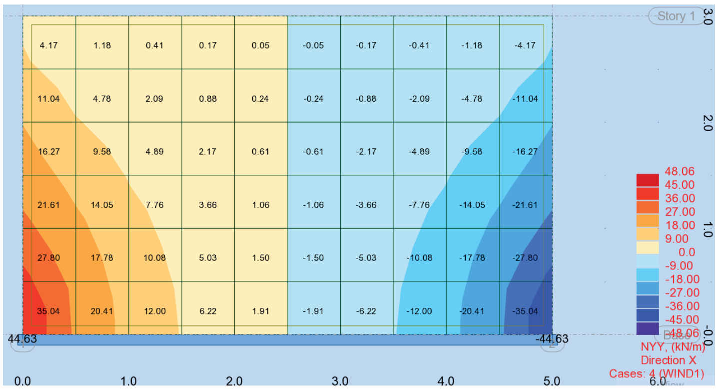
Model	D_x	D_y	D_y/D_x	Max. vertical force	$N_{max,o}$ / $N_{max,i}$
Isotropic	1039500	1039500	1	44.67	1.00
Orthotropic	1039500	1039500	1	44.67	1.00
	1039500	831600	0.8	44.64	1.00
	1039500	623700	0.6	44.63	1.00
	1039500	415800	0.4	44.73	1.00
	1039500	311850	0.3	44.92	1.01
	1039500	207900	0.2	45.48	1.02
	1039500	155925	0.15	46.15	1.03
	1039500	103950	0.1	47.58	1.07

Orthotropic results for varying Dy/Dx

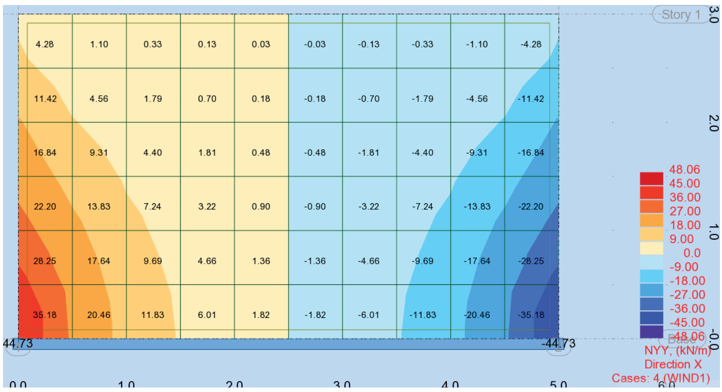
Dy/Dx = 0.8



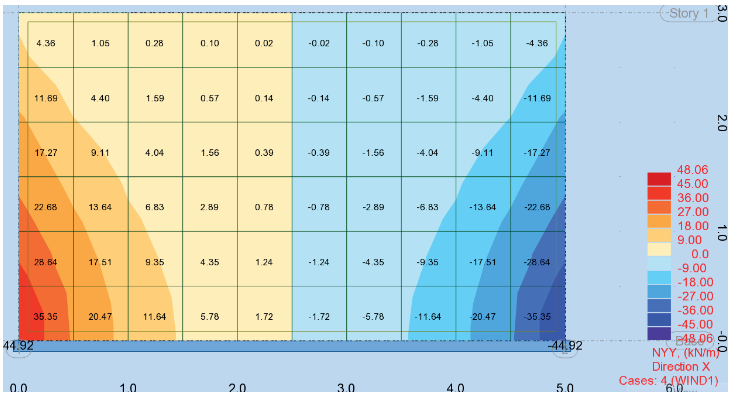
Dy/Dx = 0.6



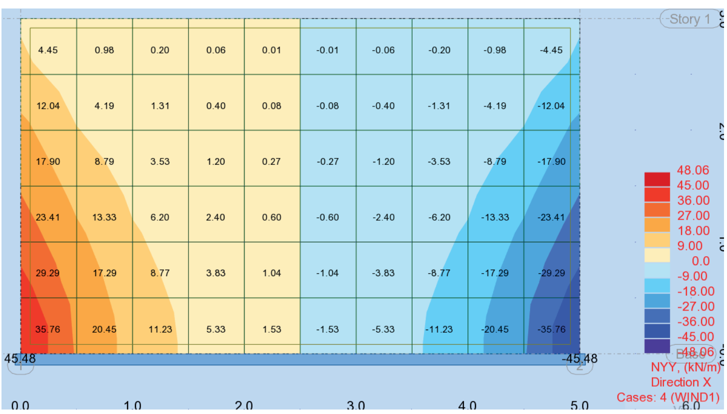
Dy/Dx = 0.4



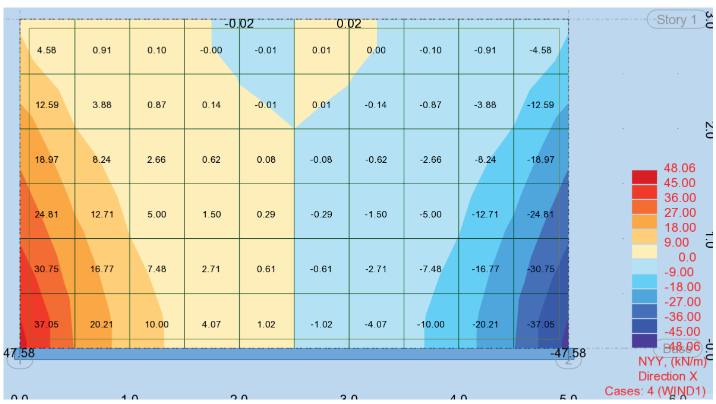
Dy/Dx = 0.3



Dy/Dx = 0.2

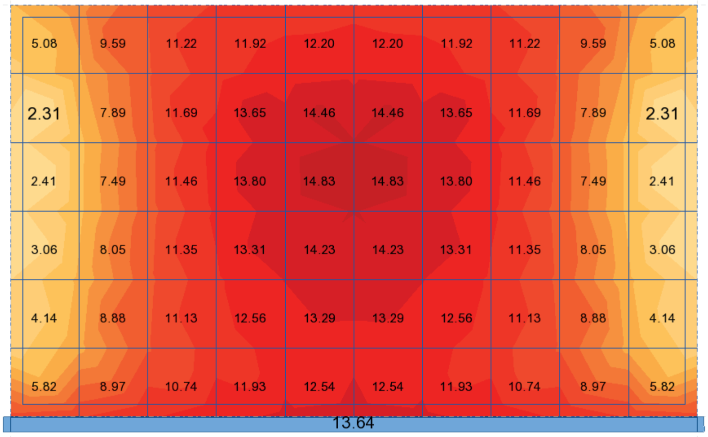


Dy/Dx = 0.1



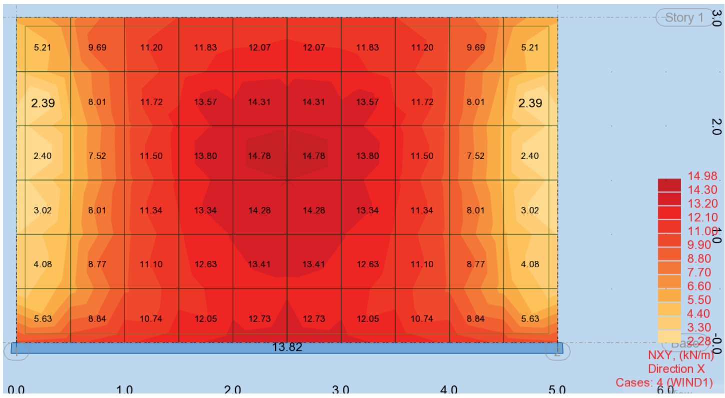
SHEAR MEMBRANE FORCES - FROM THE HORIZONTAL LOAD

Isotropic Wall

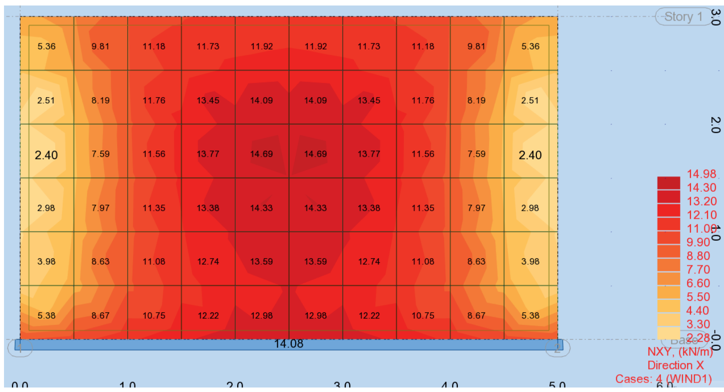


Orthotropic results for varying Dy/Dx

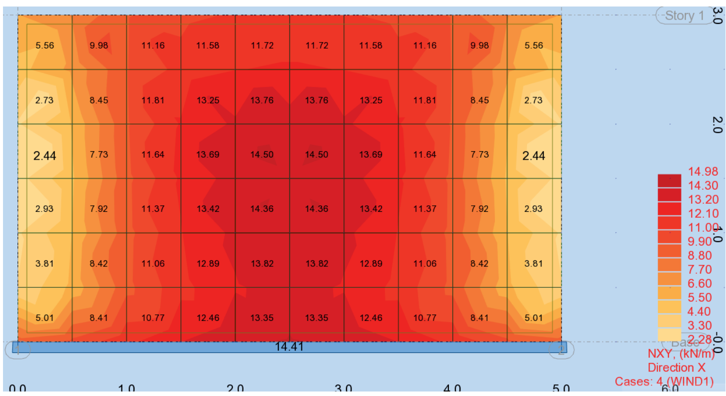
Dy/Dx = 0.8



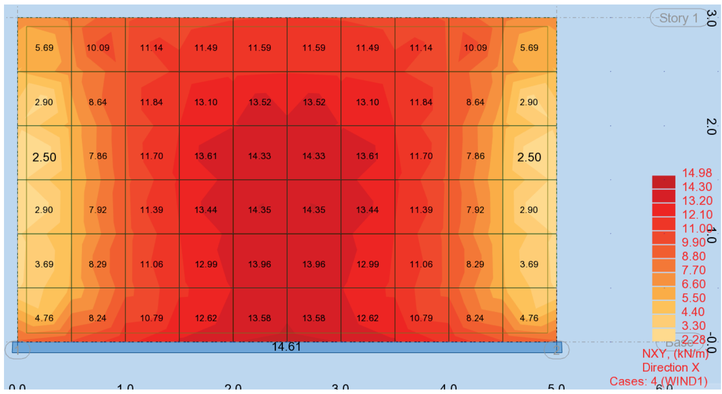
Dy/Dx = 0.6



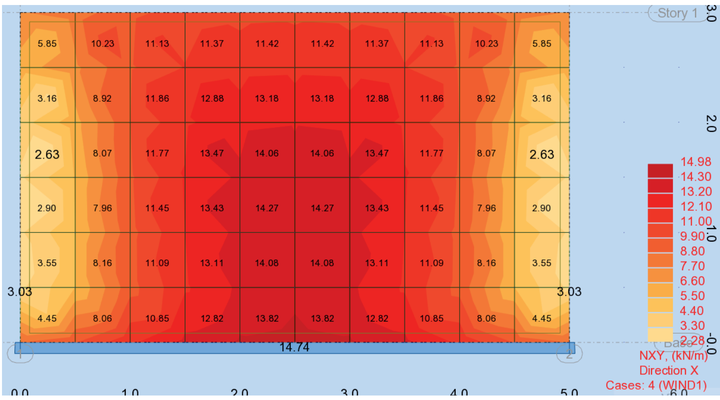
Dy/Dx = 0.4



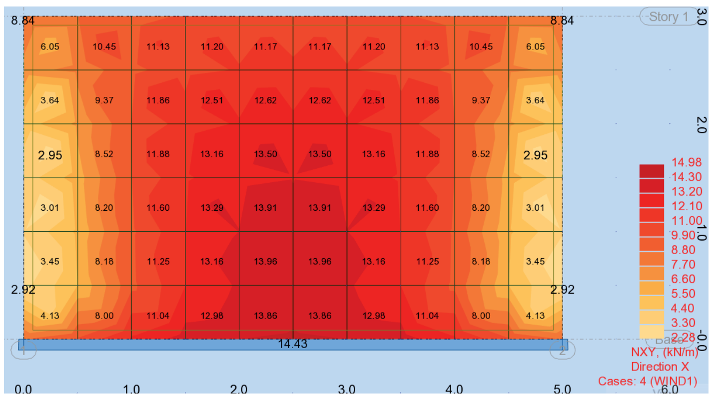
Dy/Dx = 0.3



Dy/Dx = 0.2

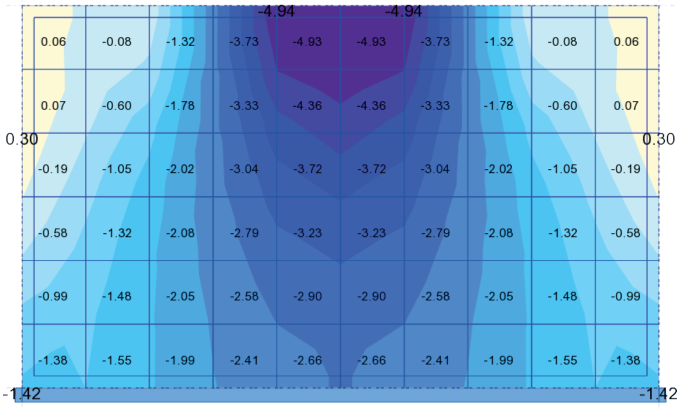


Dy/Dx = 0.1



VERTICAL MEMBRANE FORCES - FROM THE VERTICAL LOAD

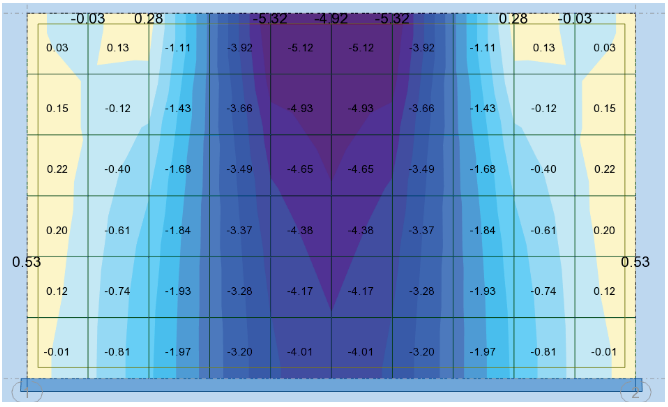
Isotropic Wall



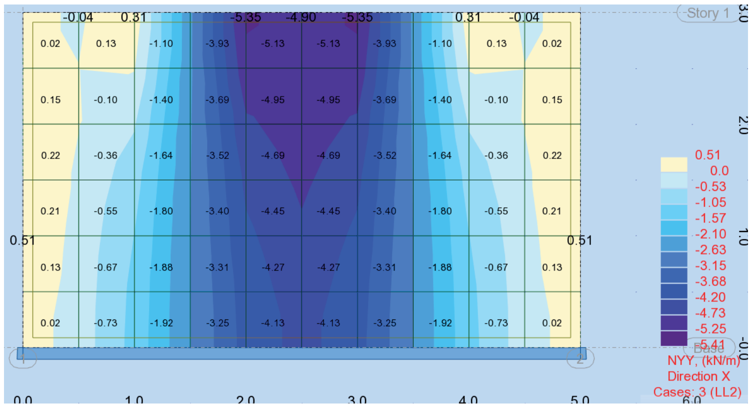
$D_{xy,iso}$ (G_{iso})	$D_{xy,ortho}$	$D_{xy,o}/D_{xy,i}$	Max. vertical force	$N_{max,o}$ / $N_{max,i}$
0	0	1	4.94	1.00
519750	129937.5	0.25	5.32	1.08
519750	103950	0.2	5.35	1.08
519750	77962.5	0.15	5.39	1.09
519750	64968.75	0.125	5.41	1.10
519750	51975	0.1	5.44	1.10
519750	46777.5	0.09	5.45	1.10
519750	44178.75	0.085	5.46	1.11
519750	41580	0.08	5.46	1.11
519750	25987.5	0.05	5.51	1.12
519750	10395	0.02	5.59	1.13

Orthotropic results for varying Dxy_ortho/Dxy_iso

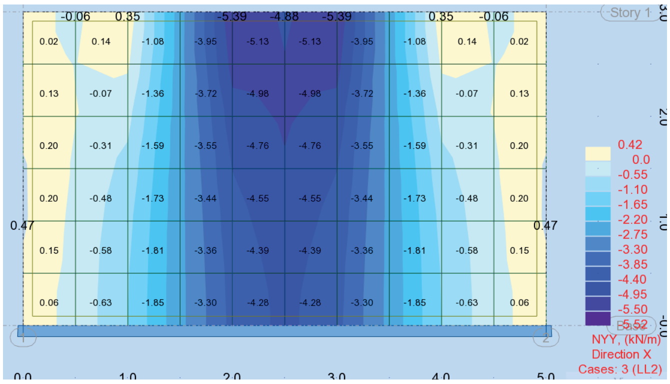
Dxy_o/Dxy_i = 0.25



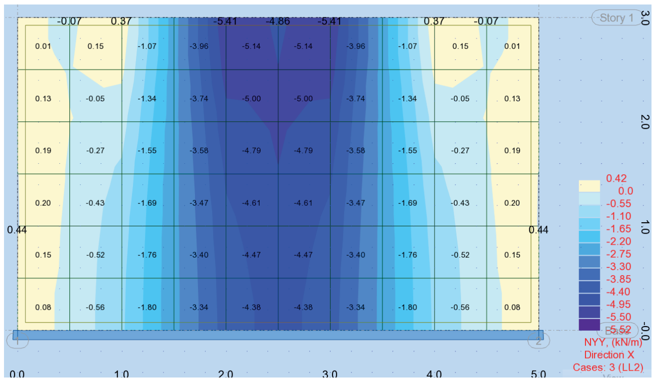
Dxy_o/Dxy_i = 0.22



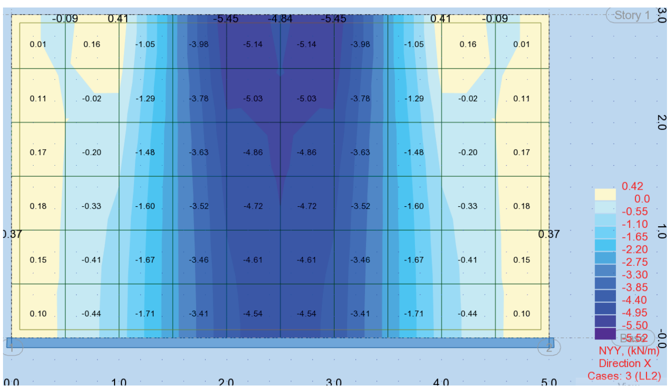
Dxy_o/Dxy_i = 0.15



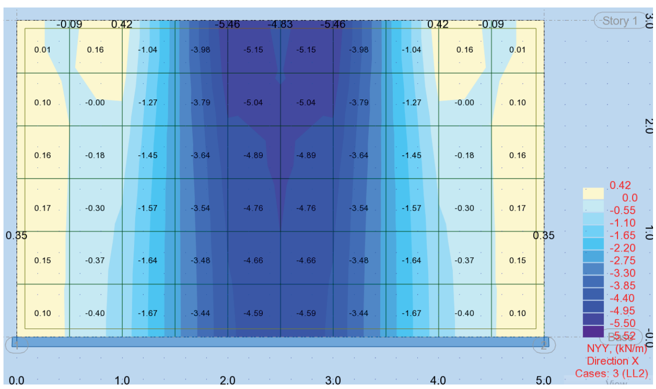
Dxy_o/Dxy_i = 0.125



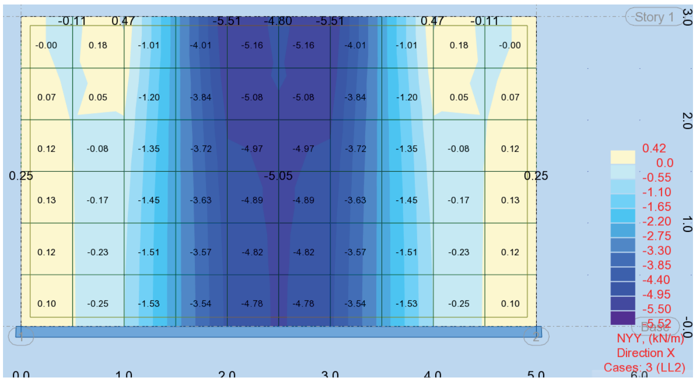
Dxy_o/Dxy_i = 0.09



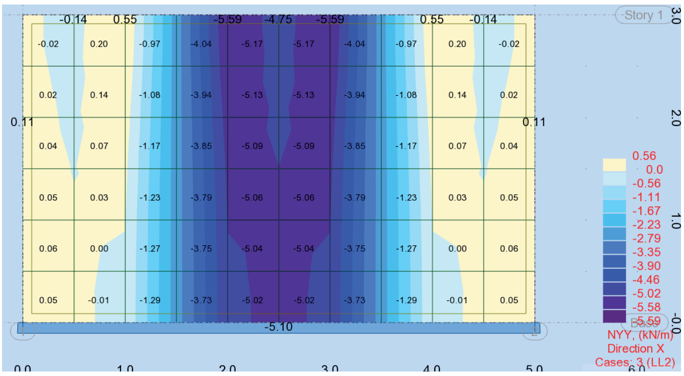
Dxy_o/Dxy_i = 0.08



Dxy_o/Dxy_i = 0.05

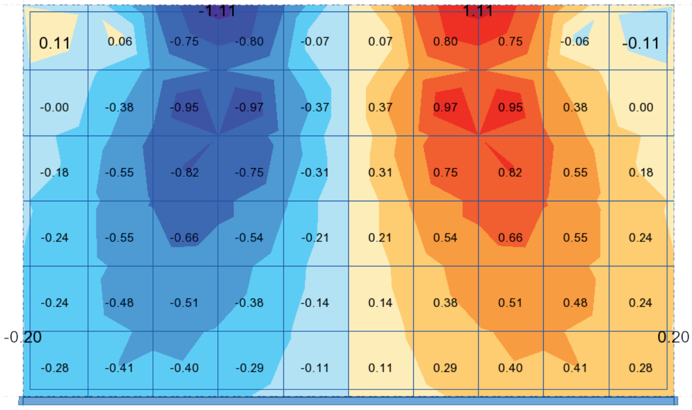


Dxy_o/Dxy_i = 0.02



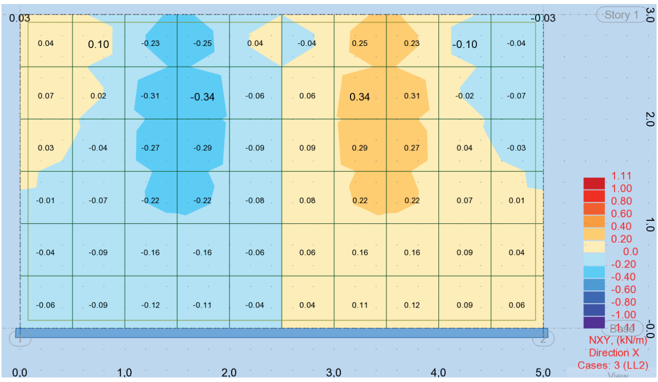
SHEAR MEMBRANE FORCES - FROM THE VERTICAL LOAD

Isotropic Wall

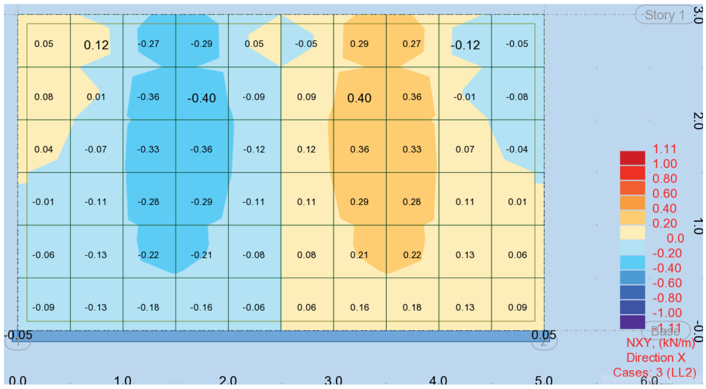


Orthotropic results for varying Dxy_ortho/Dxy_iso

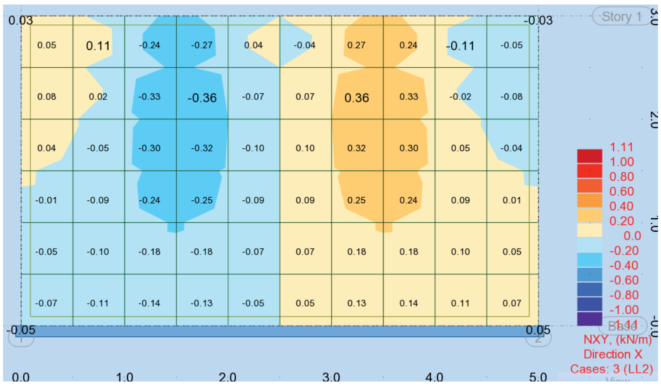
Dxy_o/Dxy_i = 0.25



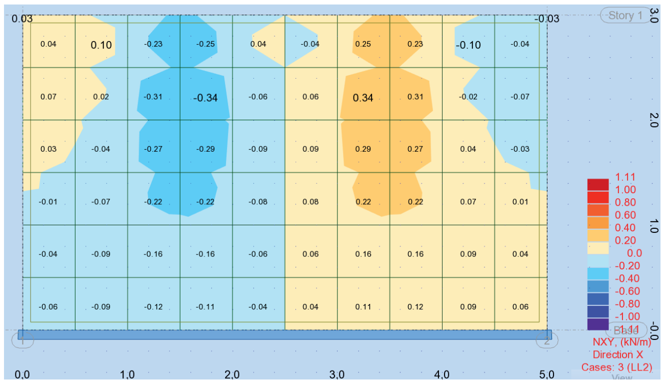
Dxy_o/Dxy_i = 0.22



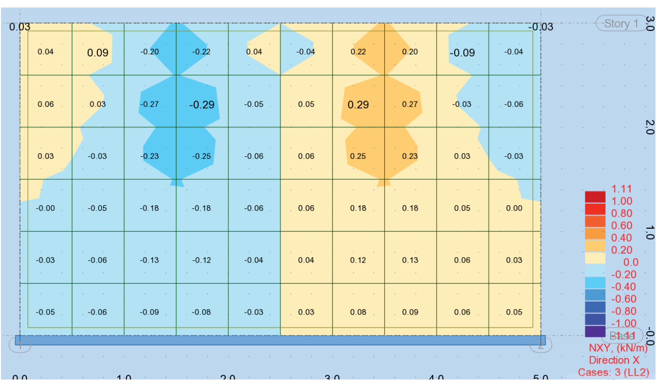
Dxy_o/Dxy_i = 0.15



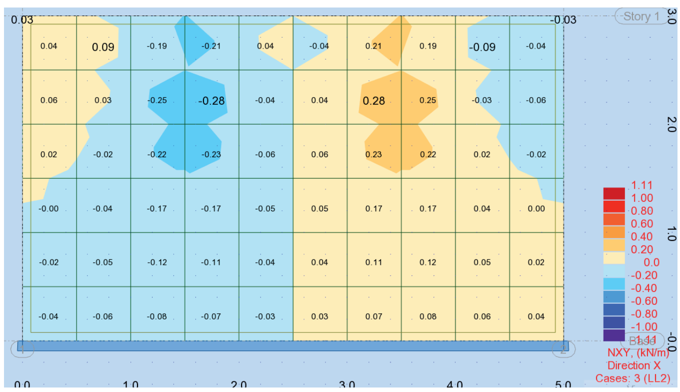
Dxy_o/Dxy_i = 0.125



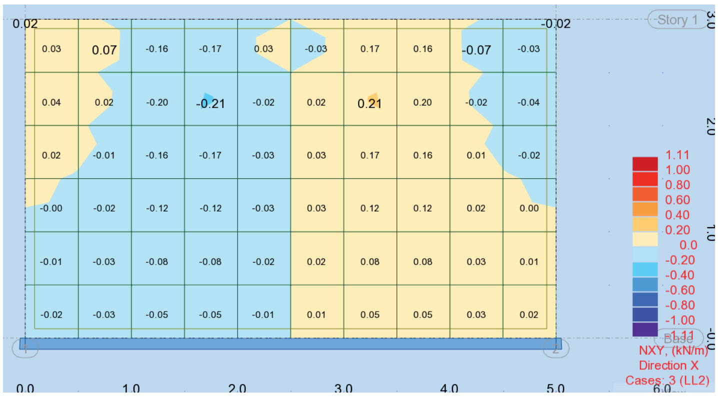
Dxy_o/Dxy_i = 0.09



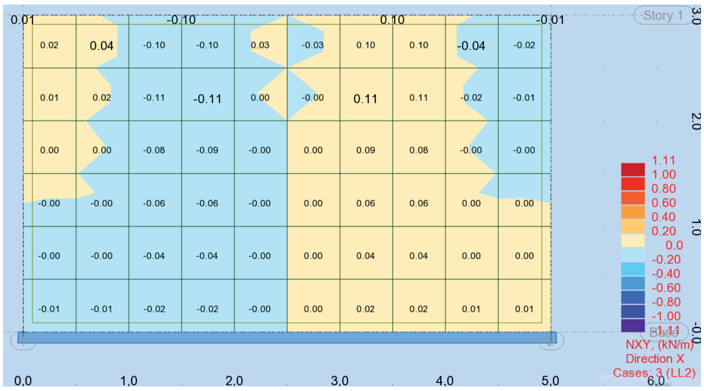
Dxy_o/Dxy_i = 0.08



Dxy_o/Dxy_i = 0.05

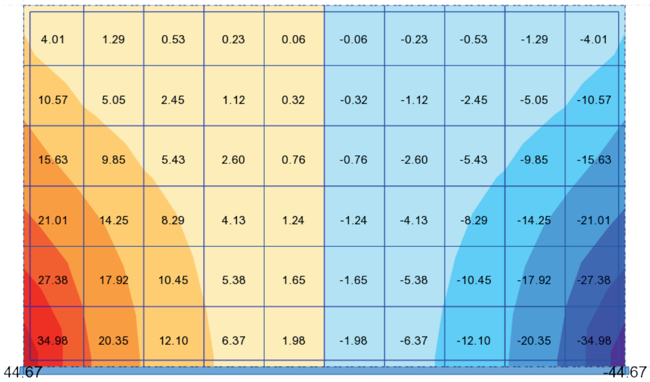


Dxy_o/Dxy_i = 0.02



VERTICAL MEMBRANE FORCES - FROM THE HORIZONTAL LOAD

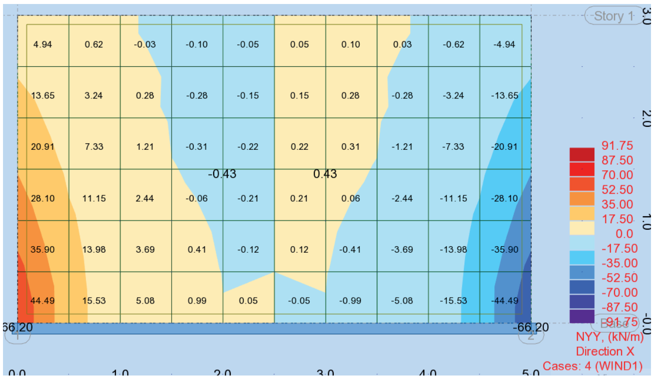
Isotropic Wall



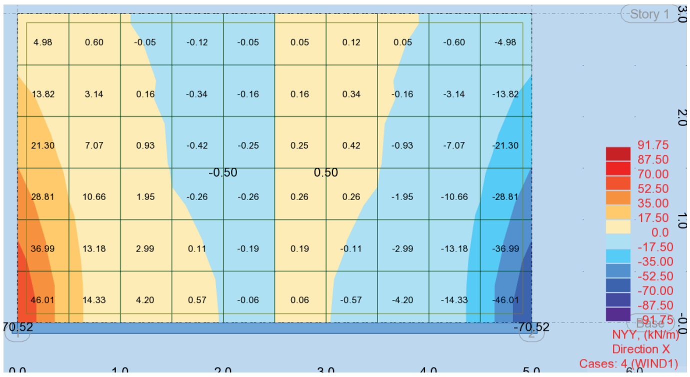
$D_{xy,iso}$ (G_{iso})	$D_{xy,ortho}$	$D_{xy,o}/D_{xy,i}$	Max. vertical force	$N_{max,o}$ / $N_{max,i}$
519750	519750	1	44.67	1.00
519750	129937.5	0.25	66.2	1.48
519750	103950	0.2	70.52	1.58
519750	77962.5	0.15	76.6	1.71
519750	64968.75	0.125	80.74	1.81
519750	51975	0.1	86.1	1.93
519750	46777.5	0.09	88.73	1.99
519750	44178.75	0.085	90.18	2.02
519750	41580	0.08	91.75	2.05
519750	25987.5	0.05	104.5	2.34
519750	10395	0.02	131.25	2.94

Orthotropic results for varying Dxy_ortho/Dxy_iso

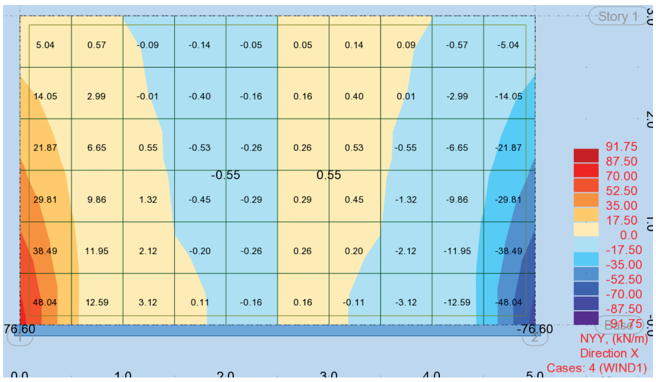
Dxy_o/Dxy_i = 0.25



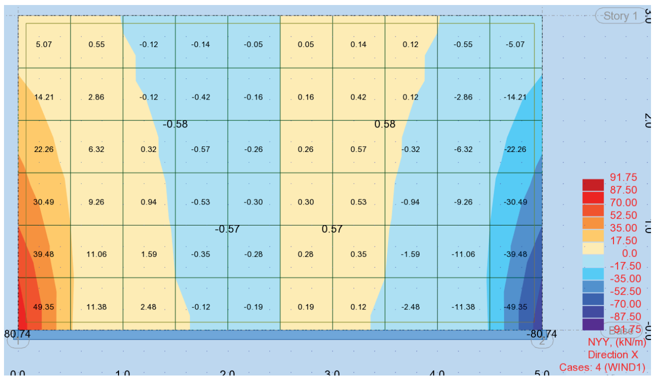
Dxy_o/Dxy_i = 0.22



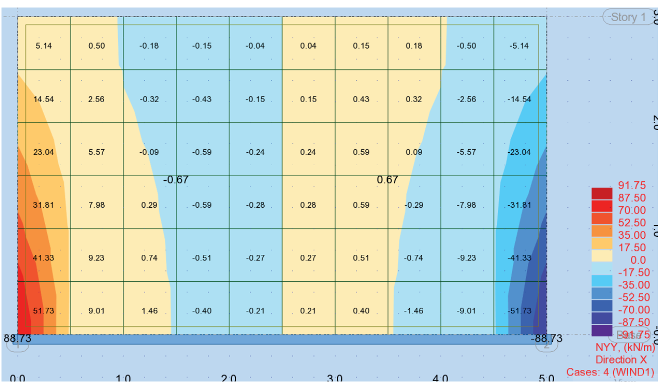
Dxy_o/Dxy_i = 0.15



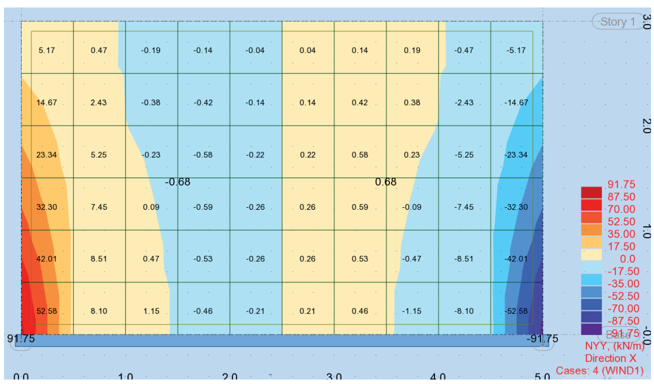
Dxy_o/Dxy_i = 0.125



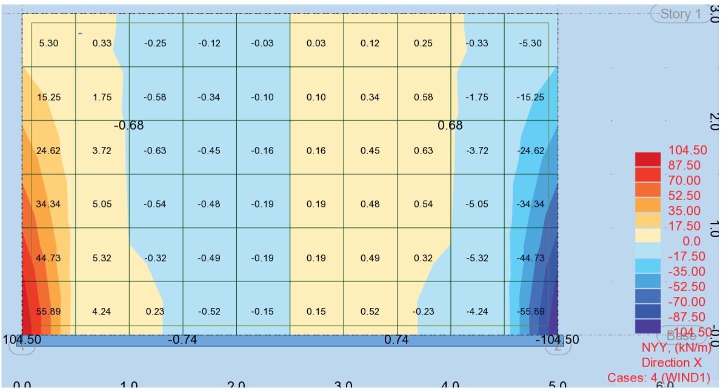
Dxy_o/Dxy_i = 0.09



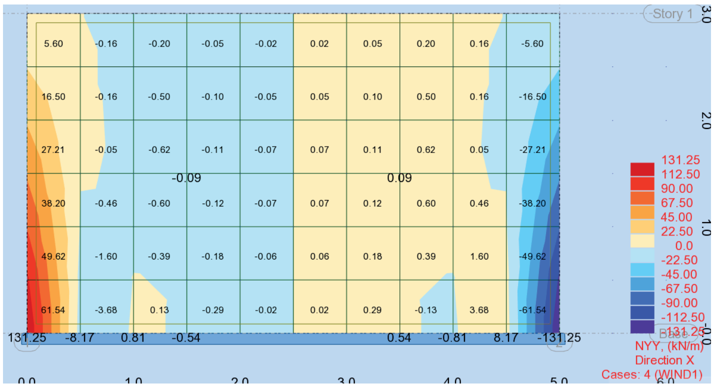
Dxy_o/Dxy_i = 0.08



Dxy_o/Dxy_i = 0.05

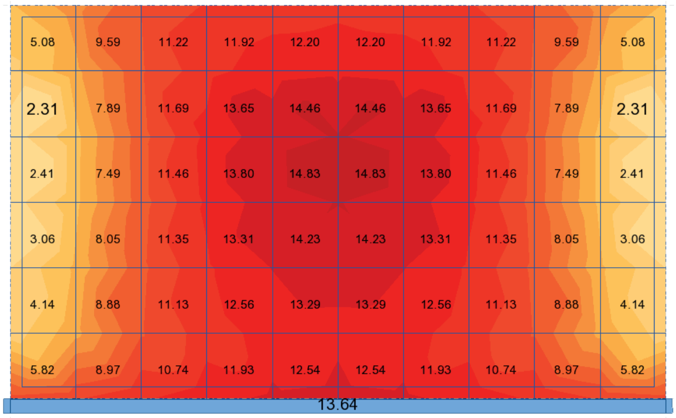


Dxy_o/Dxy_i = 0.02



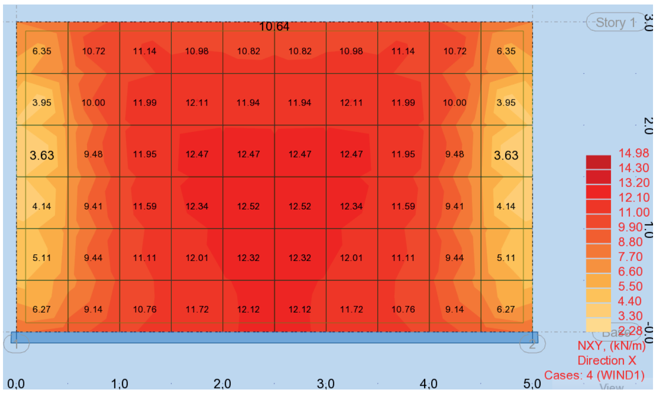
SHEAR MEMBRANE FORCES - FROM THE HORIZONTAL LOAD

Isotropic Wall

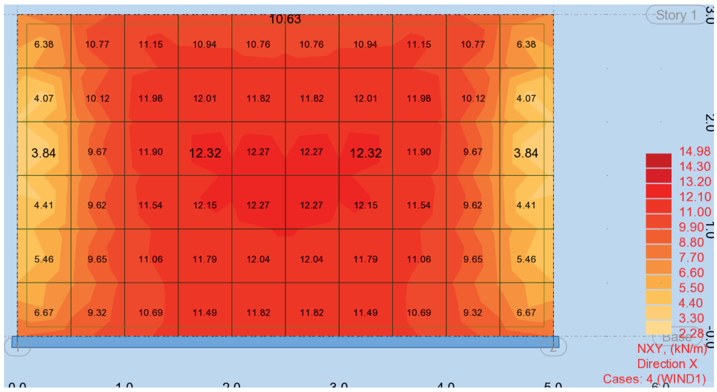


Orthotropic results for varying Dxy_ortho/Dxy_iso

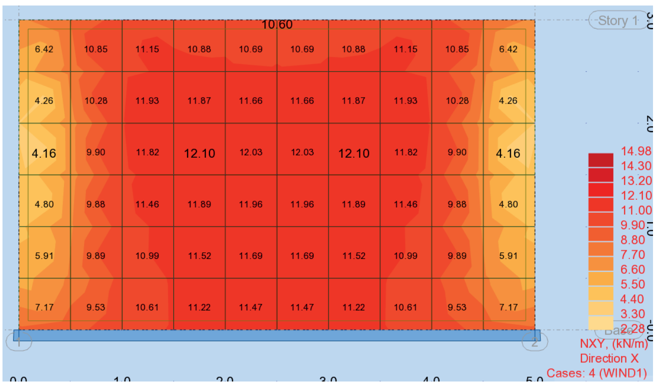
Dxy_o/Dxy_i = 0.25



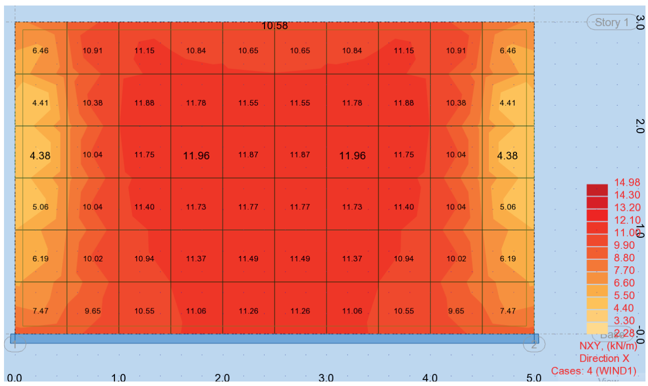
Dxy_o/Dxy_i = 0.22



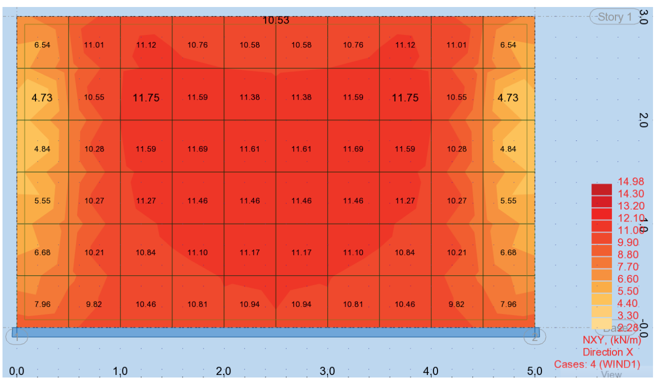
Dxy_o/Dxy_i = 0.15



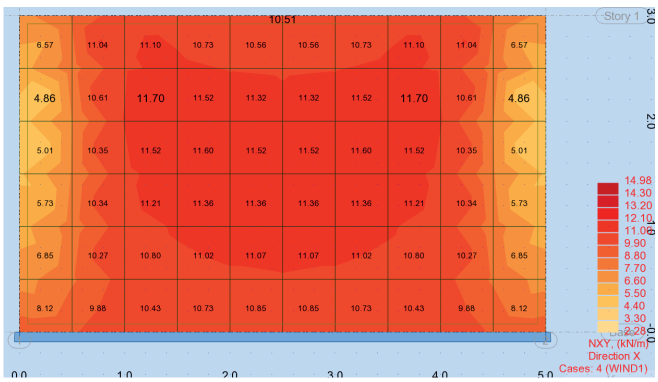
Dxy_o/Dxy_i = 0.125



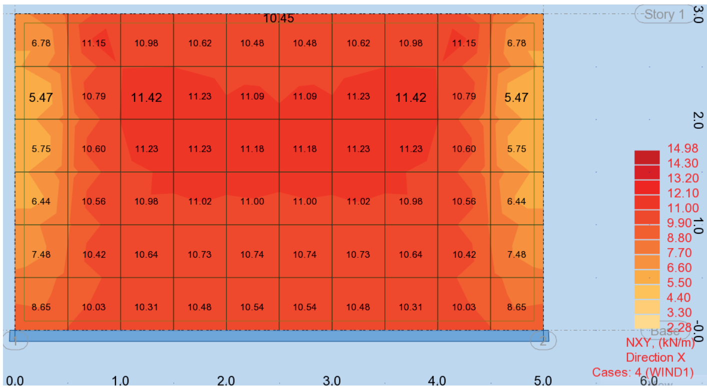
Dxy_o/Dxy_i = 0.09



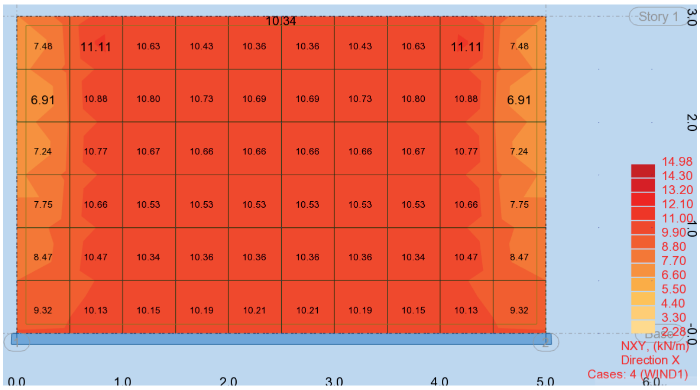
Dxy_o/Dxy_i = 0.08



Dxy_o/Dxy_i = 0.05



Dxy_o/Dxy_i = 0.02



Combined stiffness results

D_{iso}	D_{ortho}	D_{ortho} / D_{iso}	$N_{max,o}$ / $N_{max,i}$
69300	69300	1	1.00
69300	31500	0.45455	1.48
69300	26653.85	0.38462	1.58
69300	21214.29	0.30612	1.71
69300	18236.84	0.26316	1.81
69300	15065.22	0.21739	1.93
69300	13737.89	0.19824	1.99
69300	13060.98	0.18847	2.02
69300	12375	0.17857	2.05
69300	8058.14	0.11628	2.34
69300	3364.078	0.04854	2.94

Vertical force due to a horizontal load - results based on the combined stiffness ratio

Height	Combined stiffness			Simulated values			Predicted Values	
H (m)	D_{iso}	D_{ortho}	D_{ortho} / D_{iso}	$N_{max,iso}$	$N_{max,ortho}$	$N_{max,o}$ / $N_{max,i}$	$N_{max,o}$ / $N_{max,i\ predicted}$	Δ
6.00	12375.00	1546.88	0.13	80.21	192.38	2.40	2.29	4.5%
10.00	2941.98	779.63	0.27	127.97	248.22	1.94	1.81	6.9%
25.00	197.69	134.85	0.68	294.64	376.63	1.28	1.23	3.6%

Vertical force due to a horizontal load - simulation with varying wall height

Appendix B.1 - Layup file

This appendix presents the different tabs from the Layups Excel sheet. They are used to define the CLT cross sections and calculate their properties, as explained in the Framework Manual, Section 3.

- Layup tab
- Strength tab
- Stiffness tab

Layup tab

CLT PANELS LAYUPS

	Name	3L24_1	5L24_1	7L24_1	9L24_1
	Class	CL24h	CL24h	CL24h	CL24h
Applies to	Walls	0,1,2,3,4			
	Floors	0,1			
Board Width	(mm)	150	150	150	150
Gap	(mm)	0	0	0	0
Total thickness	(in mm, formula)	90	180	210	270
Warnings					
1	t (mm)	30	40	30	30
	O (°)	0	0	0	0
	e1 (mm)	30	70	90	120
2	t (mm)	30	30	30	30
	O (°)	90	90	90	0
	e2 (mm)	0	35	60	90
3	t (mm)	30	40	30	30
	O (°)	0	0	0	90
	e3 (mm)	-30	0	30	60
4	t (mm)		30	30	30
	O (°)		90	90	0
	e4 (mm)	0	-35	0	30
5	t (mm)		40	30	30
	O (°)		0	0	90
	e5 (mm)	0	-70	-30	0
6	t (mm)			30	30
	O (°)			90	0
	e6 (mm)	0	0	-60	-30
7	t (mm)			30	30
	O (°)			0	90
	e7 (mm)	0	0	-90	-60
8	t (mm)				30
	O (°)				0
	e8 (mm)	0	0	0	-90
9	t (mm)				30
	O (°)				0
	e9 (mm)	0	0	0	-120

Strength tab

STRENGTH VALUES OF LAYUPS

LAYUP-DEPENDENT STRENGTH VALUES

	Layup	Name	3L24_1	5L24_1	7L24_1	9L24_1
		Class	CL24h	CL24h	CL24h	CL24h
	Applies to	Walls	0,1,2,3,4			
		Floors	0,1			
Characteristic Values	Bending strength	$f_{m,CLT,k}$	25.26	23.57	23.21	22.63
	Tensile strength //	$f_{t,0,CLT,net,k}$	16.00	16.00	16.00	16.00
	Compression strength //	$f_{c,0,CLT,net,k}$	25.26	23.57	23.21	22.63
	Shear strength in-plane	$f_{v,net,k}$	6.00	6.00	6.00	6.00
	Rolling shear strength	$f_{r,CLT,k}$	1.10	1.10	1.10	1.10
Design Values	Bending strength	$f_{m,CLT,d}$	18.19	16.97	16.71	16.29
	Tensile strength //	$f_{t,0,CLT,net,d}$	11.52	11.52	11.52	11.52
	Compression strength //	$f_{c,0,CLT,net,d}$	18.19	16.97	16.71	16.29
	Shear strength in-plane	$f_{v,net,d}$	4.32	4.32	4.32	4.32
	Rolling shear strength	$f_{r,CLT,d}$	0.79	0.79	0.79	0.79

NON LAYUP-DEPENDENT STRENGTH VALUES

Characteristic Values	Tensile strength □	$f_{t,90,CLT,k}$	0.5	0.5	0.5	0.5
	Compression strength □	$f_{c,90,CLT,k}$	3	3	3	3
	Gross shear strength	$f_{v,gross,k}$	3.5	3.5	3.5	3.5
	Torsional shear strength	$f_{T,node,k}$	2.5	2.5	2.5	2.5
	Shear strength out-of-plane	$f_{v,CLT,k}$	3.5	3.5	3.5	3.5
Design Values	Tensile strength □	$f_{t,90,CLT,D}$	0.36	0.36	0.36	0.36
	Compression strength □	$f_{c,90,CLT,d}$	2.16	2.16	2.16	2.16
	Gross shear strength	$f_{v,gross,d}$	2.52	2.52	2.52	2.52
	Torsional shear strength	$f_{T,node,d}$	1.8	1.8	1.8	1.8
	Shear strength out-of-plane	$f_{v,CLT,d}$	2.52	2.52	2.52	2.52

Stiffness tab

STIFFNESS VALUES OF LAYUPS

LAYUP-DEPENDENT STIFFNESS VALUES

	Layup	Name Class	3L24_1 CL24h	5L24_1 CL24h	7L24_1 CL24h	9L24_1 CL24h
	Applies to	Walls Floors	0,1,2,3,4 0,1			
	Thickness	mm	90	180	210	270
Characteristic Values	Bending Stiffness Out-of-Plane [kN.m2]	K_{CLT}	676.35	4735.80	6407.78	16438.95
	Equivalent Elastic Modulus [MPa]	$E_{eq,OP}$	11133.33	9744.44	8302.92	10022.22
	Shear Stiffness Out of Plane [kN]	S_{CLT}	11620.00	23240.00	24070.00	34860.00
	Equivalent Shear Modulus [MPa]	$G_{eq,OP}$	129.11	129.11	114.62	129.11
	Axial Stiffness x [kN/m]	$EA_{eff,x}$	6.93E+05	1.39E+06	1.39E+06	2.08E+06
	Equivalent Elastic Modulus [MPa]	$E_{eq,IP}$	7700.00	7700.00	6600.00	7700.00
	Shear stiffness In Plane [kN/m]	S_{xy}	46237.87	87846.76	107888.36	138713.61
	Equivalent Shear Modulus [MPa]	G^*_{IP}	513.75	488.04	513.75	513.75
Design Values	Bending Stiffness Out-of-Plane [kN.m2]	K_{CLT}	541.08	3788.64	5126.22	13151.16
	Equivalent Elastic Modulus [MPa]	$E_{eq,OP}$	8906.67	7795.56	6642.33	8017.78
	Shear Stiffness Out of Plane [kN]	S_{CLT}	9296.00	18592.00	19256.00	27888.00
	Equivalent Shear Modulus [MPa]	$G_{eq,OP}$	103.29	103.29	91.70	103.29
	Axial Stiffness x [kN/m]	$EA_{eff,x}$	5.54E+05	1.11E+06	1.11E+06	1.66E+06
	Equivalent Elastic Modulus [MPa]	$E_{eq,IP}$	6160.00	6160.00	5280.00	6160.00
	Shear stiffness In Plane [kN/m]	S_{xy}	36990.30	70277.41	86310.69	110970.89
	Equivalent Shear Modulus [MPa]	G^*_{IP}	411.00	390.43	411.00	411.00

NON LAYUP-DEPENDENT STIFFNESS VALUES

Elastic modulus //	$E_{0,CLT,mean}$	11550	11550	11550	11550
Elastic modulus T	$E_{90,CLT,mean}$	300	300	300	300
Shear modulus	$G_{CLT,mean}$	650	650	650	650

OTHER MATERIAL VALUES

Mean density [kN/m3]	$\rho_{CLT,mean}$	4.2	4.2	4.2	4.2

Appendix B.2 - Load Calculations file

This appendix presents the different tabs from the Loads Excel sheet. They are used to calculate the different loads that are to be applied to the building, as explained in the Framework Manual, Section 4.

By order of apparition:

- General tab
- Dead Load tab
- Imposed Load tab
- Snow Load tab
- Wind Load tab
- Output tab

General Tab

I. GENERAL INFORMATION

A. BUILDING GEOMETRY

Building		
Height	30.40	m
Length	34.40	m
Width	14.80	m

Levels		
Level height	3.40	m
Roof height	3.20	m
Roof Type	Pitched Roof	
Pitch Angle	10.54	°

Consequence class	CC3

Country Location	DENMARK

Dead Load Tab

DEAD LOADS

Please include only the finishes and not the structural CLT itself; the structure's self-weight is integrated directly by the Karmaba software

FLOOR STRUCTURE

Regular storey 0	Material	Thickness [mm]	Density [kN/m3]	Load [kN/m2]
	Laminated wood finish	5	7	0.04
	Wood chip board	19	8	0.15
	OSB	18	7	0.13
	Impact sound insulation	30	1.4	0.04
	Ceiling installations			0.50
	Gypsum boards	30.8	8	0.25

Totals Floor type	Load [kN/m2]
0	1.10

WALL STRUCTURE

Wall Type 0	Material	Thickness [mm]	Density [kN/m3]	Load [kN/m2]
	Gypsum board	13	8	0.10
	Insulation	50	1.4	0.07
	Insulation	50	1.4	0.07
	Gypsum board	13	8	0.10

Totals Wall Type	Load [kN/m2]
0	0.35
1	0.41
2	0.35
3	0.41
4	0.41

Wall Type 1	Material	Thickness [mm]	Density [kN/m3]	Load [kN/m2]
	Wooden cladding	30	5	0.15
	Sheathing board	9	5	0.05
	Insulation	150	1.4	0.21

Wall Type 2	Material	Thickness [mm]	Density [kN/m3]	Load [kN/m2]
	Gypsum board	13	8	0.10
	Insulation	50	1.4	0.07
	Insulation	50	1.4	0.07
	Gypsum board	13	8	0.10

Wall Type 3	Material	Thickness [mm]	Density [kN/m3]	Load [kN/m2]
	Wooden cladding	30	5	0.15
	Sheathing board	9	5	0.05
	Insulation	150	1.4	0.21

Wall Type 4	Material	Thickness [mm]	Density [kN/m3]	Load [kN/m2]
	Wooden cladding	30	5	0.15
	Sheathing board	9	5	0.05
	Insulation	150	1.4	0.21

Imposed Loads Tab

IMPOSED LOADS

DENMARK NA

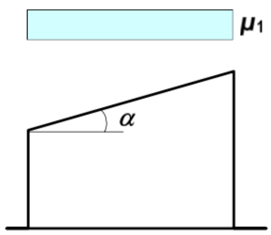
When adding a new load category, please pull down the formulae from the previous row

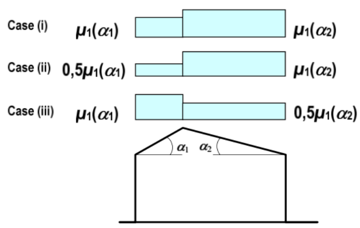
		EC1-1 DK NA Table 6.1		*EC 1-1 DK NA Table 6.2*	*EC 1-1 6.3.1.2(8)*			*EC 0 DK NA Table A1.1*		
Storey	Type	Applies to (Floor type)	Category	Load [kN/m2]	Partitions	Load [kN/m2]	Total [kN/m2]	Ψ_0	Ψ_1	Ψ_2
Regular storey	Residential	0	A	1.5	Medium	0.8	2.3	0.5	0.3	0.2
Roof	Roof	1	H	0	None	0	0	0	0	0

SNOW LOAD

Snow Load Calculation				DENMARK NA
sk		1.00	kN/m2		DS/EN 1991-1-3 DK NA, 4.1(1)
Topography	Normal				EN 1991-1-3, Table 5.1
					DS/EN 1991-1-3 DK NA, Table 5.1.a NA
ctop		1.00			DS/EN 1991-1-3 DK NA, 5.2(7)
Cs		1.00			DS/EN 1991-1-3 DK NA, 5.2(7)
ce		1.00			DS/EN 1991-1-3 DK NA, 5.2(7)
ct		1.00		Recommended value: 1	EN 1991-1-3, 5.2(8)

			Monopitch Roof EN 1991-1-3, 5.3.2	Pitched Roof EN 1991-1-3, 5.3.3
Max. Load			0.80	0.80
Pitch Angle		α1 [°]	10.54	10.54
		α2 [°]	-	10.54
Shape coefficients		μ1 (α1)	0.80	0.80
		μ1 (α2)	-	0.80
Snow Load EN 1991-1-3, 5.2(3a)	Undrifted (i)	s1 [kN/m2]	0.80	0.80
		s2 [kN/m2]	-	0.80
	Drifted (ii)	s1 [kN/m2]	0.80	0.40
		s2 [kN/m2]	-	0.80
	Drifted (iii)	s1 [kN/m2]	-	0.80
		s2 [kN/m2]	-	0.40





Wind load tab

WIND LOADS CALCULATION

A. GENERAL

Terrain category		III		(DK NA includes costline in I)
Fundamental value of basic wind	$v_{b,0}$	24.00	m/s	DENMARK
Directional factor	c_{dir}	1.00		(Recommended value)
Seasonal factor	c_{season}	1.00		(Recommended value)
Basic Wind Velocity	v_b	24.00	m/s	
Terrain parameters	z_0	0.300	m	
	z_{min}	5.00	m	
	z_{max}	200.00	m	(Defined by EC1-4)
	$z_{0,II}$	0.05	m	
Orography factor	c_0	1.00		(Recommended)
	k_r	0.22		
Turbulence factor	k_l	1.00		
Standard deviation of turbulence	σ_v	5.17	m/s	
Air density	ρ	1.25	kg/m^3	(Defined by EC1-4)
External Pressure coefficients	$c_{pe,10}$ (D)	0.80		
	$c_{pe,10}$ (E)	-0.70		(Recommended)
Reduction factor for lack of correlation	Long side	0.89		
	Short side	0.85		

B. BOUNDARY VALUES FOR WIND PROFILE

Height	30.40	m
Storey height	3.40	m
Length	34.40	m
Width	14.80	m

Boundary values for wind profile		Long side	Short side	
	b	14.80	34.40	m
	h	30.40	30.40	m
	h-b	15.60	-4.00	m
	q_p (b)	0.69	0.00	kN/m^2
	q_p (h)	0.90	0.90	kN/m^2

C.1. SIMPLIFIED WIND PROFILE ON BUILDING

This section can be used accurately only for the first two cases described in Figure 7.4, EC 1-1-4. For the last case (CASE 3), a (safe) simplification is made.
For fine tuning of the wind load in the last case, it is recommended to adapt the tables from section E

Long side Profile	CASE 3 Corresponding storey n°	Height	Roughness factor cr(z)	Mean wind velocity (m/s), vm(z)	Turbulence intensity Iv(z)	Peak velocity pressure (kN/m2)	Wind pressure long side (kN/m2)	Applies on storey	GH syntax for storey selection
h	Roof	30.40	0.99	23.87	0.22	0.90	1.20	4 to Roof	!(0to3)
h-b	3	15.60	0.85	20.43	0.25	0.72	0.96	None	
b	3	14.80	0.84	20.15	0.26	0.71	0.95	0 to 3	0to3

Long side Profile	CASE 1 Corresponding storey n°	Height	ess factor cr(z)	Mean wind velocity (m/s), vm(z)	Turbulence intensity Iv(z)	Peak velocity pressure (kN/m2)	pressure long side (kN/m2)	Applies on storey	syntax for storey selection
h	Roof	30.40	0.99	23.87	0.22	0.90	1.14	ALL	?
h-b	-1	-4.00	0.61	14.54	0.36	0.46	0.59	None	
b	-1	34.40	1.02	24.51	0.21	0.93	1.19	None	

C.2. ADVANCED WIND PROFILE ON BUILDING

If needed (e.g. in CASE 3), the tables in this section must be adapted to obtain a more accurate wind profile. Note that results from section D are on the safe side.

Storey	Top Height (m)	Roughness factor	Mean wind velocity (m/s)	Turbulence intensity	Peak velocity pressure (kN/m2)	Wind pressure long side (kN/m2)	Wind pressure short side
n°	z	cr(z)	vm(z)	Iv (z)	qp(z)	wy (z)	wx (z)
Roof	30.40	0.99	23.87	0.22	0.90	1.20	1.14
7	27.20	0.97	23.30	0.22	0.87	1.20	1.14
6	23.80	0.94	22.61	0.23	0.83	1.20	1.14
5	20.40	0.91	21.81	0.24	0.79	1.20	1.14
4	17.00	0.87	20.87	0.25	0.74	1.20	1.14
3	13.60	0.82	19.72	0.26	0.69	0.95	1.14
2	10.20	0.76	18.23	0.28	0.62	0.95	1.14
1	6.80	0.67	16.13	0.32	0.53	0.95	1.14
0	3.40	0.61	14.54	0.36	0.46	0.95	1.14
-1	0.00						

Wind Loads on Roofs

		F	G	H	I	J
Duopitch Roof	0	0.10	0.10	0.10	-0.24	-0.24
		-1.13	-0.88	-0.39	-0.44	-0.42
		0.10	0.10	0.10	-0.44	-0.42
		-1.13	-0.88	-0.39	-0.24	-0.24
	180	0.10	0.10	0.10	-0.24	-0.24
		-1.13	-0.88	-0.39	-0.44	-0.42
		0.10	0.10	0.10	-0.44	-0.42
		-1.13	-0.88	-0.39	-0.24	-0.24
	90	-1.28	-1.17	-0.58	-0.49	0.00
		Min. load	Max. load			
Load for combinations		-1.28	0.10			

Output tab

LOAD VALUES

Consequence	CC3
K_{FI}	1.1

NOTE: Load combinations are taken from the Danish National Annex to Eurocode 0.

A. DEAD LOADS - LOAD CASE 0

1	COMBINATIONS	Comb ID	Factors γ * KFI
ULS	- STR 6.10b Lead IL	0	1.1
	- STR 6.10b Lead W	1	1.1
	- STR 6.10b Lead W + fav. DL	2	0.9
	- STR 6.10b Lead S	3	1.1
SLS	- Characteristic Value (6.14b) Lead IL	4	1
	- Characteristic Value (6.14b) Lead W	5	1
	- Quasi-Permanent Value (6.16b)	6	1

Load type		DL Floors	DL Roof	DL Walls	DL Walls
Applied to	Floor/Walls	Floors	Floors	Walls	Walls
Typeeristic		0	1	0,2	1,3,4
Value	[kN/m^2]	1.10	1.10	0.35	0.41

B. IMPOSED LOADS - LOAD CASE 1

1	COMBINATIONS	Comb. ID	Factors	Floor type	Floor type
				0	1
			γ * KFI	Ψ	Ψ
ULS	- STR 6.10b Lead IL	0	1.65	1	1
	- STR 6.10b Lead W	1	1.65	0.5	0
	- STR 6.10b Lead W + fav. DL	2	1.65	0.5	0
	- STR 6.10b Lead S	3	1.65	0.5	0
SLS	- Characteristic Value (6.14b) Lead IL	4	1	1	1
	- Characteristic Value (6.14b) Lead W	5	1	0.3	0
	- Quasi-Permanent Value (6.16b)	6	1	0.2	0

	Total	Total
Floor type	0	1
Load	2.3	0
0	1.65	1.65
1	0.825	0
2	0.825	0
3	0.825	0
4	1	1
5	0.3	0
6	0.2	0

C. SNOW LOADS - LOAD CASE 2

1	COMBINATIONS	Comb. ID	Factors γ * KFI	Floor type 1 Ψ
ULS	- STR 6.10b Lead IL	0	1.65	0.3
	- STR 6.10b Lead W	1	1.65	0
	- STR 6.10b Lead W + fav. DL	2	1.65	0
	- STR 6.10b Lead S	3	1.65	1
SLS	- Characteristic Value (6.14b) Lead IL	4	1	0.3
	- Characteristic Value (6.14b) Lead W	5	1	0
	- Quasi-Permanent Value (6.16b)	6	1	0

	Total
Floor type	1
Load	0.80
0	0.50
1	0.00
2	0.00
3	1.65
4	0.30
5	0.00
6	0.00

D. WIND LOADS - LOAD CASE 3

1	COMBINATIONS	Comb. ID	Factors γ * KFI	Ψ	Total
ULS	- STR 6.10b Lead IL	0	1.65	0.3	0.50
	- STR 6.10b Lead W	1	1.65	1	1.65
	- STR 6.10b Lead W + fav. DL	2	1.65	1	1.65
	- STR 6.10b Lead S	3	1.65	0.3	0.50
SLS	- Characteristic Value (6.14b) Lead IL	4	1	0.3	0.30
	- Characteristic Value (6.14b) Lead W	5	1	1	1.00
	- Quasi-Permanent Value (6.16b)	6	1	0	0.00

Load type	Roof	Long side	Long side	Long side	Short side	Short side	Short side
Floors/ Stories	Floors	Stories	Stories	Stories	Stories	Stories	Stories
Applied to	1	!(0to3)		0to3	?		
Value	-1.28	1.20	0.96	0.95	1.14	0.59	1.19

Appendix C - List of Digital Annexes

Excel files

- Layups.xlsx
- PuukuokkaLoads.xlsx

Grasshopper scripts

- Puukuokka.gh

Grasshopper clusters

- Referencing.ghcluster
- Storeygenerator.ghcluster
- Genotyper.ghcluster
- Phenotyper.ghcluster
- LayupXL.ghcluster
- Crosssectionviewer.ghcluster
- Mesher.ghcluster
- Karambaelements.ghcluster
- Karambasupports.ghcluster
- CLTMaterial.ghcluster
- DeadLoad.ghcluster
- ImposedLoad.ghcluster
- WindLoad.ghcluster
- WindLoadRoof.ghcluster
- SnowLoad.ghcluster
- HorizontalSway.ghcluster
- WallDesign.ghcluster
- SlabDesign.ghcluster