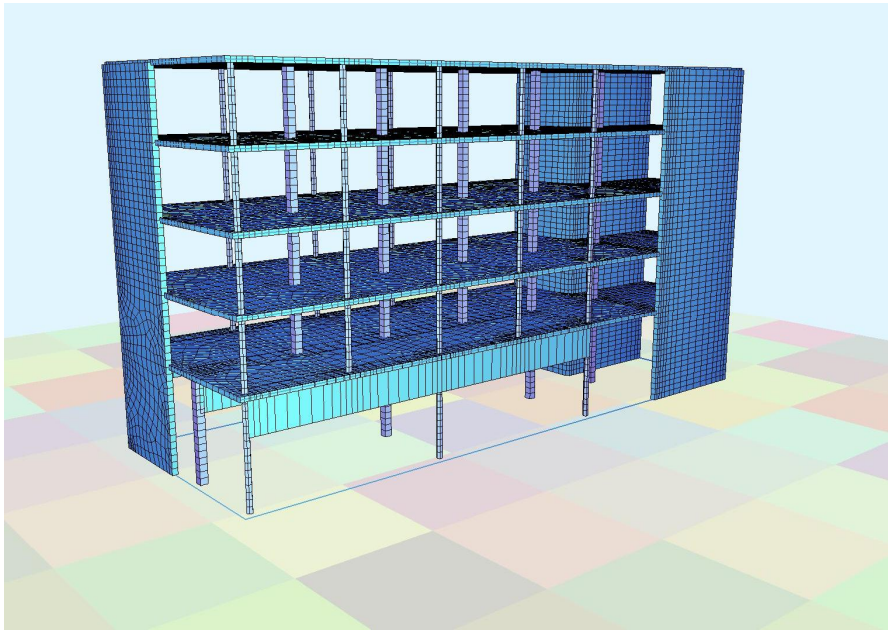
Tutorial

3D multi-storey office building

SSD

SOFiPLUS(-X)



Contents

1 Preface

1.1 What is the intention of this tutorial?

This tutorial is an introduction into 3-d modelling of a multi-storey building. It will guide you through the whole process of modelling. Having a focus on the general approach of handling a 3-d model with our software, this example shows you the analysis according to EC 1 and 2.

Our graphical user interface, the SOFiSTiK Structural Desktop (SSD) will be used as a command center. It allows you to control pre-processing, processing and post-processing for the entire SOFiSTiK Software suite. For the system- and load generation we will use SOFiPLUS(-X).
The chosen example of a multistorey office biulding deals only with the upgoing construction. The modelling of basement and foundation will not be discussed here. Please aware that we use rigid support conditions to simplify the model. This has to be modified for every project.

1.2 What can the tutorial not provide?

The tutorial can neither discuss all program parameters nor substitute the program manuals. We assume a general knowledge for all basic program features. For more information about SSD we refer to the basic SOFiSTiK Structural Desktop Tutorial, you will find in the SSD Menu Help > Quick Reference… .

1.3 Program versions

- SOFiSTiK 23
- SOFiPLUS-X 16.4 build 22 or SOFiPLUS 17.1 build 22 with AutoCAD 2007 or higher.

1.4 Legend for this tutorial

SOFiPLUS(-X):

- Commands that can/ should be written in the command line begin with an underline (i.e. _audit)
- All other commands are marked with bold letters and command (i.e. **command structural line**); The commands are availabel via Icon in toolbox or via menu (command line is also possible, but then you have to know the right syntax)
- If you have to use the menu, the menu path is signed by a > (i.e. file>save)

2 Description of the project

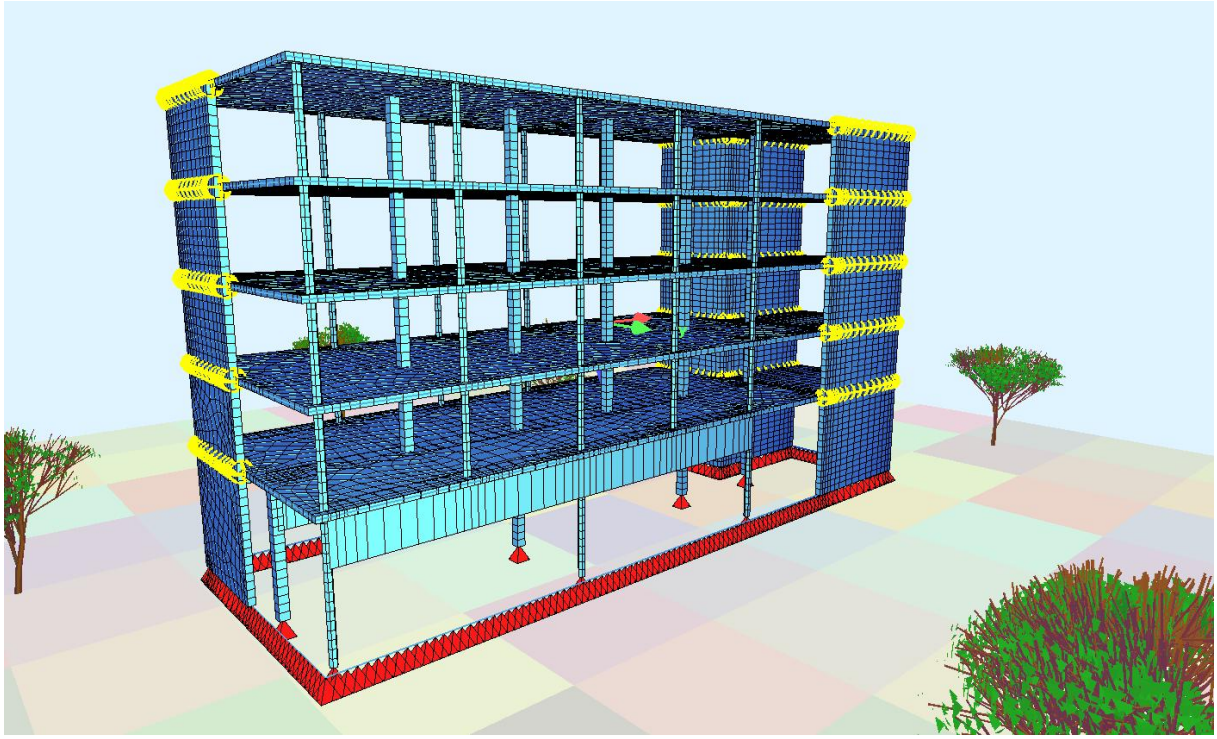


Figure 1: Overview building

This tutorial will explain how to analyse the multi-storey office building shown in figure 1. The main structure contains of shear walls, columns, beams and slabs as well as a stiffening core (staircase and elevator).
Shear walls and stiffening core support the structure for overall stability. Columns, beams and walls basically transfer vertical loads. The cladding transfers the wind loads to the floor slabs. It is one element from bottom to top and thus acts like a continuous beam. It also transfers the vertical loads on all floor slabs.

The building has a width of 12.0 m, a length of 34.6 m and is 19.5 m high. Walls are made of concrete C 30/37 and reinforcement steel S 500. The slabs (with T-beams) and alle other cross section are made of C 20/25 and reinforcement steel S 500.

The analysis will be done according to Eurocode 2.

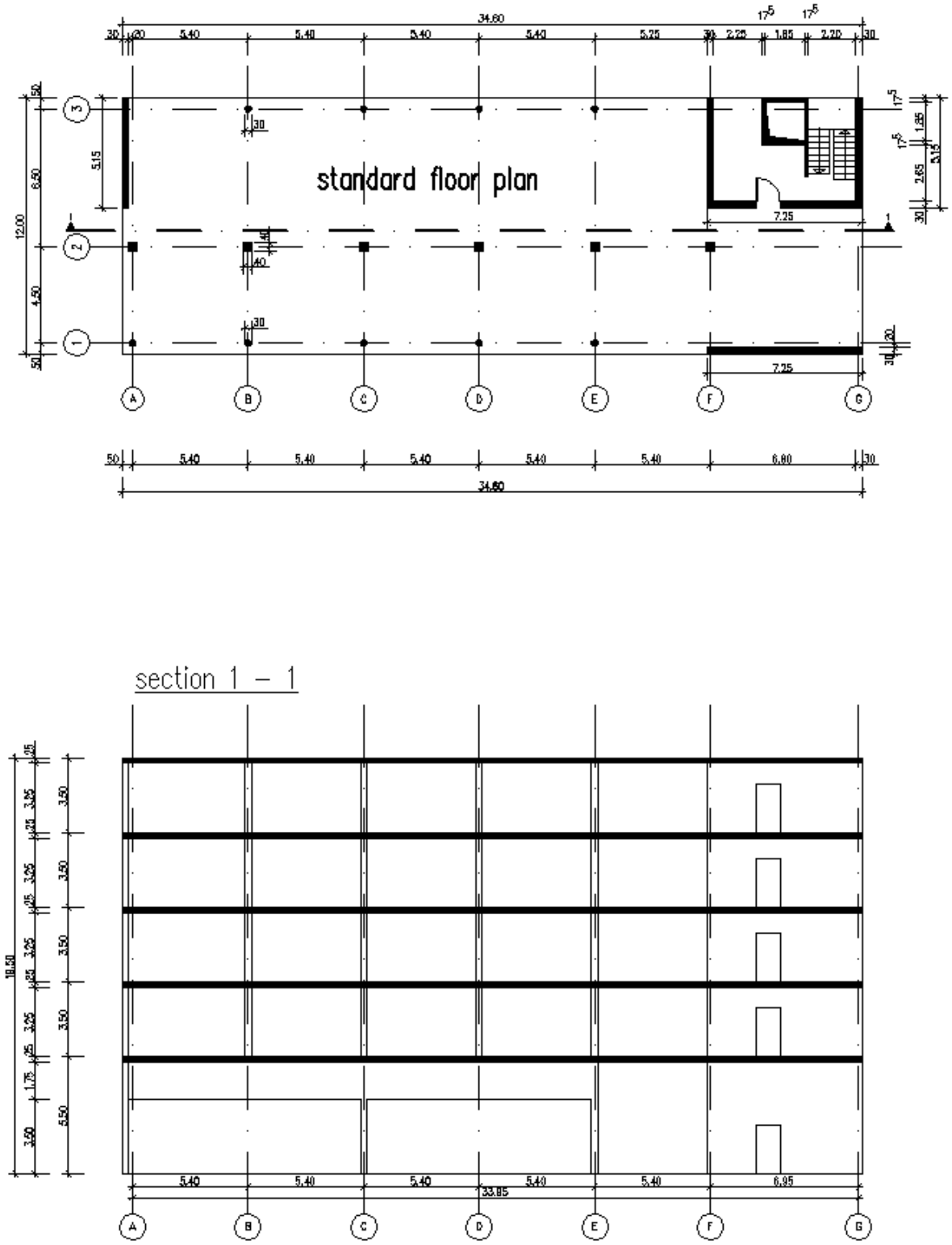


Figure 2: floor plan and section 1-1

According to EC1 the following loads have to be considered:

Type of load	**load value**
Self weight of the structure	calculated by the software
Cladding	0,50 kN/m²
Allowance for light weight dividing walls	1,20 kN/m²
Live load (offices, halls…)	2,00 kN/m²
Live load on stairways (here only slabs; staircases are not modelled; staircase loads are not included)	5,00 kN/m²
Snow	0,75 kN/m²
Wind	⇒ look at the table below

wind in global YY (on the long side)				
	area	cpe	q [kN/m²]	we [kN/m2]
walls	A	-1,2	0,75	**-0,900**
	B	-0,8	0,75	**-0,600**
	C	0	0,75	**0,000**
	D	0,8	0,75	**0,600**
	E	-0,7	0,75	**-0,525**
roof	F	-1,8	0,75	**-1,350**
	G	-1,2	0,75	**-0,900**
	H	-0,7	0,75	**-0,525**
	I	0	0,75	**0,000**

wind in global XX (on the gable side)					
**		area	cpe	q [kN/m²]	we [kN/m2]
walls	h ^ = b	A-1	-1,200	0,65	**-0,780**
		B-1	-0,800	0,65	**-0,520**
		C-1	-0,500	0,65	**-0,325**
		D-1	0,741	0,65	**0,482**
		E-1	-0,465	0,65	**-0,302**
walls	h v b	A-2	-1,200	0,75	**-0,900**
		B-2	-0,800	0,75	**-0,600**
		C-2	-0,500	0,75	**-0,375**
		D-2	0,741	0,75	**0,556**
		E-2	-0,465	0,75	**-0,349**
roof		F	-2,11	0,75	**-1,583**
		G	-1,31	0,75	**-0,986**
		H	-0,70	0,75	**-0,525**
		I	0,2/-0,2	0,75	**0,150**
…I with change of sign/ direction					
** because hight>widht wind load area must be divided over hight according to the code					

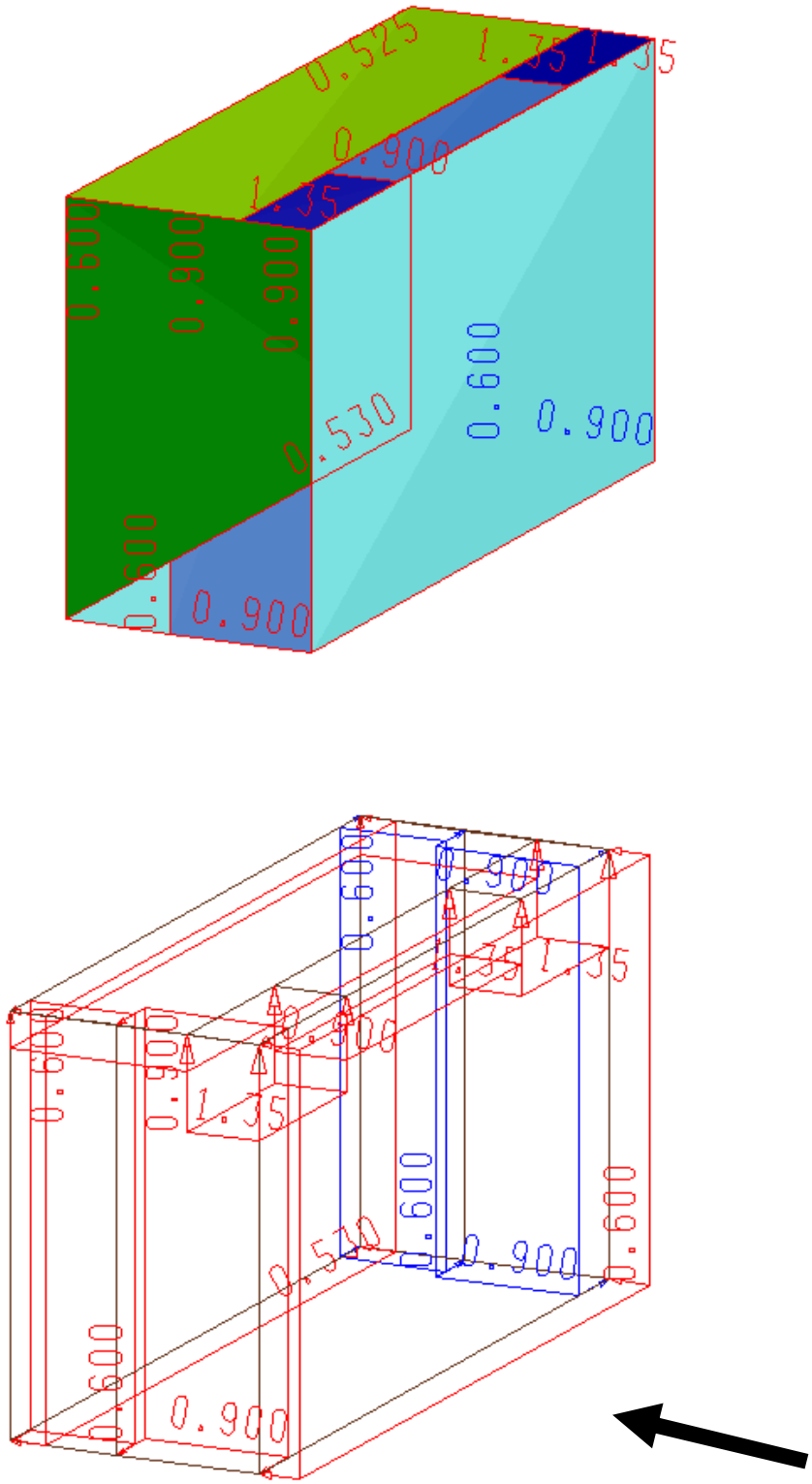


Figure 3: example – wind in global Y-direction (shown as filled area and as vector)

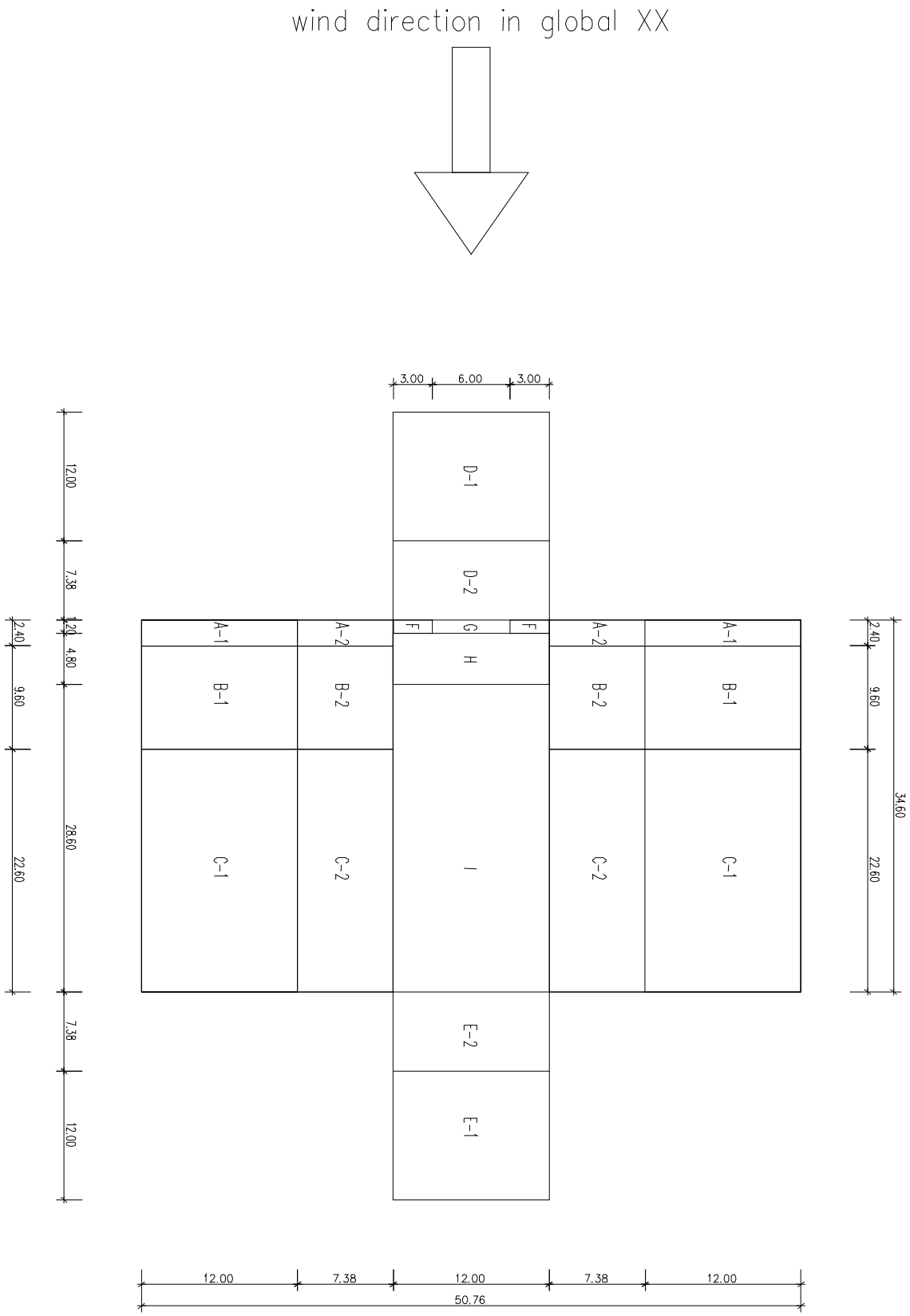


Figure 4: overview load areas for wind in global X-direction

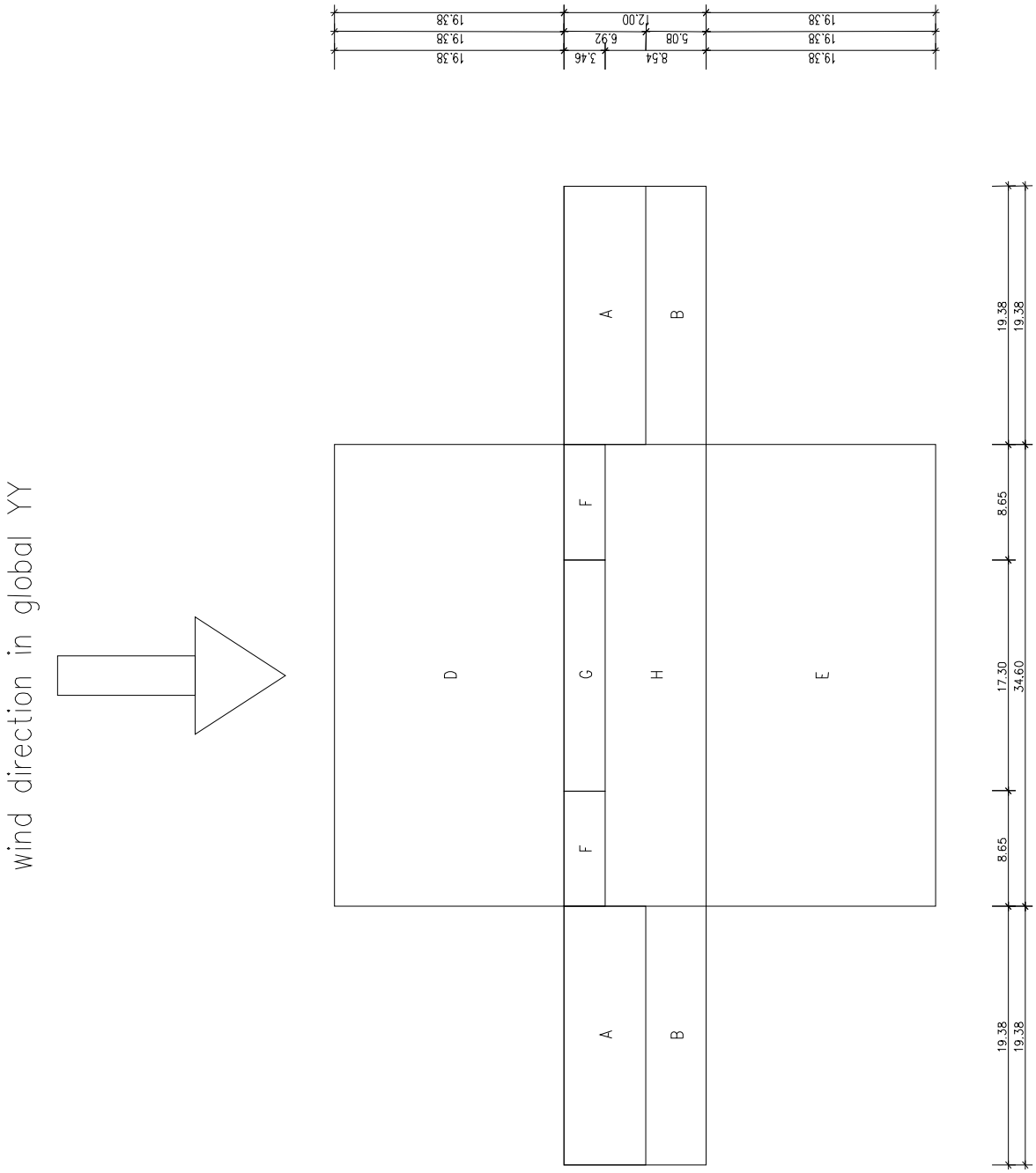


Figure 5: overview load areas for wind in global Y-direction

This is only a very short description how to define Windloads according to EC 1 and how to apply these loads to our examples. A fundamental knowledge of the relevant codes is also necessary.

3 Why making a 3d-model?

Before starting with the project, let's discuss the characteristics of 2D versus 3D modelling.

	2D Modelling	**3D Modelling**
Workflow for a structure	split construction into structural members; analyse each member separately	one complex model
Input/ handling	easy for each member; but often results in a lot of single, independent files	complex; but only one file for the whole structure
Level of abstraction	high	low
Modelling of details	good for modelling details, bad for coherence	modelling details not recommended, good for showing coherence
Time for system generation	little	plenty
Changes/ updates during working process	by hand for each member; danger to forget something; a lot of work	just once for the whole model
Complexity of model	low	High danger of black box effect
Verifiability (by hand)	simple	hard
Quality of the results	independent of the kind of modelling, but depending on the quality of the model	
Global behaviour of the structure	hard to predict - imprecise	more precise, e.g. redistribution of forces can be shown
Ability to model and show dependencies	bad	good
Analysis of local stability	easy	hard
Dynamic analysis (i.e. earthquake)	hard/ impossible	easy
Time for analysis	low for single components	high – the whole system has to be analysed
Focus on	local design (details)	global design (main structural elements)

The table shows that each method has its strengths and weaknesses.
2-D and 3-D modelling should be used complementary - or depending on the single job definition - apart.

This tutorial will show you the workflow we suggest if you want to use a 3-D model. Nevertheless, this building could also be modelled in 2-D. It is up to the engineer to decide, which model would be best for his project. It is similar to the decision if you want to go to Munich by train, plain or car. With the car, you can make your travel individually, going by train is very comfortable and by plane is the fastest way to travel. Each possibility has its advantages and disadvantages. Only if you know the whole circumstances, you can make the best decision.

4 From the static system to the FEA-model

4.1 Preliminary considerations

To avoid problems during the analysis and design of a 3d-structure we recommend doing some planning before starting to actually work with the software. As discussed in the last chapter it is not possible to make a complete design in all details using a 3d-model.

4.1.1 Considerations about the system

You first should make a list of all the design checks you have to make.
Based on this list you can decide which components of the structure you have to model and how far you can simplify these (rule: as simple as possible, but as exact as necessary).
Next you should check if some components could be merged to one structural element (e.g. one cross section for similar columns).
Making a pre-design of the main structural members (e.g. on a simple beam-model) can save you a lot of time during the design process and allows you to easily check results. It also can help if you are not sure how to model details. You can see how big the influence of the structural member is on the main structure and if it is worth modelling detailed or sufficient using a coarse model.

4.1.2 Considerations about loads and actions

Make a list of all actions and loads (see chapter 2 Description of the project).
Define a concept for the load case numbers. SOFiSTiK recommends using load case numbers smaller than 1000 for single load cases, because numbers larger than 1000 are used for load case combinations by default. It is useful to divide the load cases in small sections according to their actions. For analyzing this building the following load case concept will be used:

Load case(s)	Content
1 – 99	Dead loads
1	Automatically determined self weight
2	Dead load in offices/ halls etc.
3	Dead load cladding
100 – 199	Live loads on slabs/ roof
101 – 113	Live loads
200 – 299	Wind loads
201	Wind –Y
202	Wind +Y
203	Wind +X; roof +
204	Wind +X; roof -
205	Wind –X; roof -
206	Wind –X; roof +
300 – 399	Snow
300	Snow on roof

Keep your system flexible and upgradeable, i.e. don't use only consecutive load case numbers – if you let some numbers in-between you will be able to add e.g. a "forgotten" load case without changing the whole concept.

Number zone	Load case combinations by default
1100 – 1200 (default)	SLS – permanent
1400 – 1500 (default)	SLS – permanent (here: nodal displacements)
2100 – 2200 (default)	ULS

In some cases the program uses the same load case to save the results of different superpositions. Nevertheless, the description only shows the name of last superposition that has been saved with this load case number. If it's annoying – just rename.

4.1.3 Considerations about groups

What means group concept – why should I use groups in my model?

Group concept is an classification system to keep your model clear. You can connect parts of your structure by similarities; i.e. one group for each construction stage or one group for each cross section….
If you define your groups in a resonable way (=group concept) you'll be able to (de)select a particular structural system, apply loads, analyse and design elements or make graphical post-processings very fast and effective.

There is no universal concept for the definition of groups, but it rather depends on the problem that has to be solved. In one case e.g. it makes sense to define all walls in one group and all slabs in another. In another case it might be better to group the elements by floor level.

Using SOFiPLUS-(X) 16.4/17.1 the group-divisor is the same for all groups. With the default setting of 10.000 you may use a maximum of 999 groups. Generally speaking: the group number multiplied with the group-divisor has to be less than 10.000.000.
Group divisor means the max. number of (finite) elements in one group. (quads, beams, springs,…)
Elementnumber of the finite element consists of group number (1st part) and element number (2nd part).
(example: group divisor 10000; finite element 345; group 23 elementnumber> 230345)

The following table shows how the elements in this example are classified into groups:

Component	Formula for group number
Slabs/ roof	level number x 100 i.e. 1st floor: group number = 1 x 100 = 100
Columns (assuming not more than 50 different cross sections)	cross section number + Level number x 100 i.e. column with cross section 1 – ground floor: group number = 1 + 0 x 100 = 1 exception: all dummy beams are in group 49
Beams	same group number as the respective slab i.e. T-beam in slab of 1st floor: group number =100
Walls (assuming not more than 50 different walls on each floor)	wall number + 50+ level number x 100 i.e. wall number 1 in 3rd level: group number = 1 + 50 + 3 x 100= 351

4.2 Modelling the details

Although it is the primary purpose to model a realistic behaviour of the structure, it is important to keep your model as simple as possible. It is worth spending some time thinking about the details to avoid mistakes and get the most efficient model. Having a model with fewer elements can not only save a lot of calculation time but will also help to understand results.

Modelling the details is not only depending on the specifics of FEA but also on the construction sequence as well as on good engineering practice. In the following some details and decisions for the multi-storey office building will be discussed.

4.2.1 Connection walls/ columns – slabs

Following comments are made for walls, but in general meaning they are also hold for true for connetions column-slab.

Basically there are two possibilities how to model the connection between walls and slabs:

a) rigid connection
b) hinged connection

For the input of the model it is easier to choose a rigid connection because this is the default setting in SOFiPLUS(-X). But you have to consider this effect when planning the reinforcement.

If using a hinged connection there is no analytical bending moment between the slab and the wall. Therefore planning and building the reinforcement is a lot easier.

The true structural behaviour is somewhere in-between case a) and b). Thus it is up to the structural engineer how to model the connections. Tables Table 1 (vertical modeling details; page 14) and Table 2 (vertical modeling details; page 15) show what decisions were made for the multi-storey office building of this tutorial.

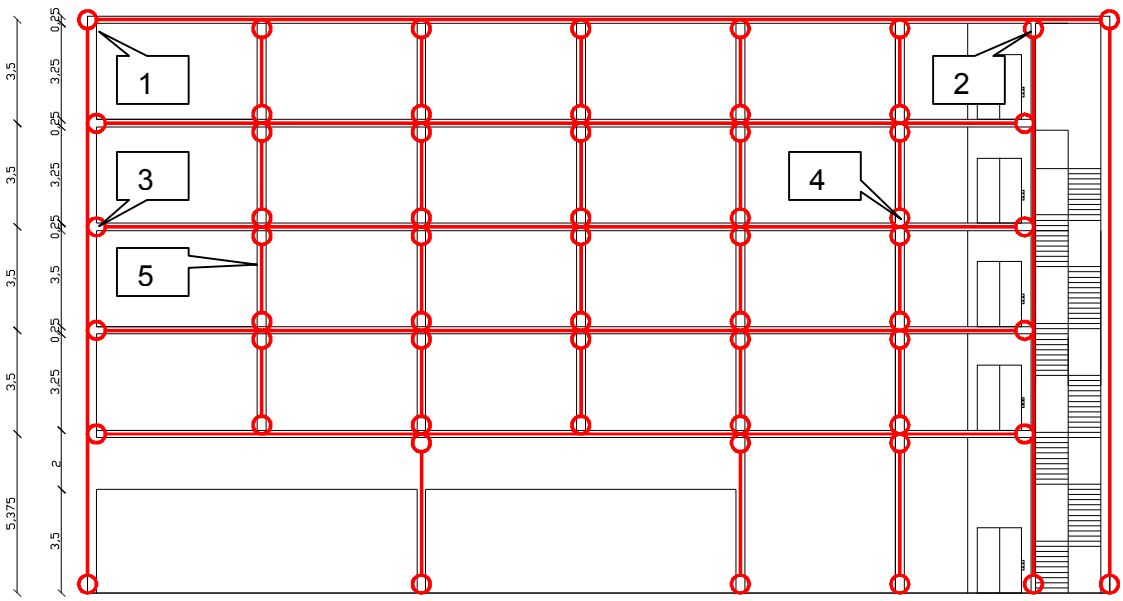


Figure 6: overview vertical connection details

Description (number of details – please look at Figure 6: overview vertical connection details)
Case 1: exterior wall –roof • hinge on top of wall • the middle axis of walls is moved to the real borderline of the building; so the wall elements in the model have their nodes in the middle, but wall is slightly moved (on the conservative side)
Case 2: interior wall - roof • hinge on top of wall
Case 3: floor slab - exterior wall • hinge in floor slab • wall acts like a continous beam
Case 4: interior (core) wall on slab • hinges on top and bottom wall • floor slab acts as continuous beam
Case 5: columns • all columns are modelled as pin-ended columns from slab to slab

Table 1: vertical modeling details – connection walls/columns ó slabs/ roof

4.2.2 Horizontal details

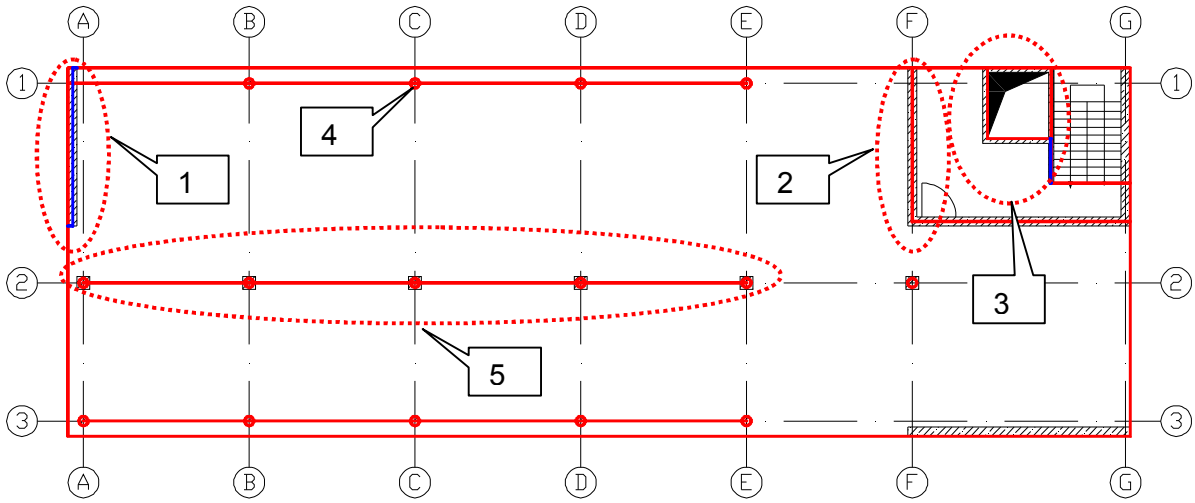


Figure 7: overview horizontal modelling details

Description
Case 1: exterior wall ó real slab dimensions/ mesh • to avoid bends in system lines (which are not existing in reality) and singularities in the mesh, the system line of the wall is set to the boundary of the slab (instead of 1/3 line of the wall)
Case 2: interior wall • system line = middle line of the wall
Case 3: interior walls around opening • avoiding bends in system line by choosing middle line for the wall at right, because it is small compared to the adjacent wall • other walls analogous to case 1
Case 4: column support • Model as a single structural point (with input of dimension for punching)
Case 5: downstand beam • downstand beams are modelled directly inside the slab using a structural line with beam properties (the program takes care of the downstand beam - for further information please see our paper t-beam philosophy).

Table 2: horizontal modeling details

Another usual case is a column close to a border of a slab (i.e. real column dimension is equal to the border of the slab), but the center point of the column is in the slab. In those cases you should model the edge column directly on the boundary. This will result in a better FE mesh (the minimum increase in span width is usually negligible)

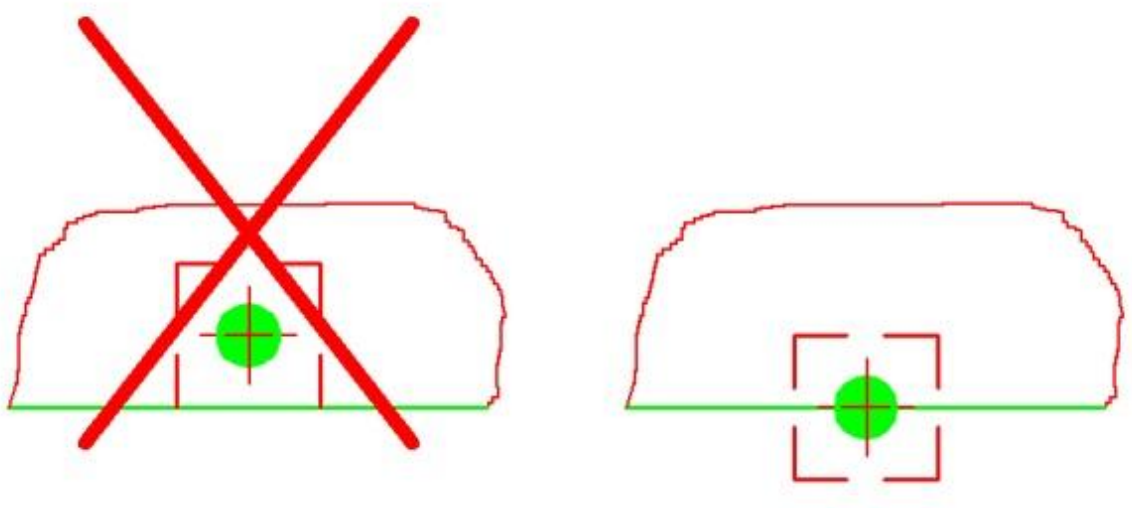


Figure 8: Modelling edge columns close to the borderline

4.2.3 Modelling wall pillars

For wall pillars, you have to decide whether it is better to model a column using beam elements or quad elements. This question cannot be answered in general. The engineer has to decide for each case separately, depending on the system and the applied forces.

The problem is not only the design of the wall or column itself. But depending on the model you will also get different results for the adjoining slab, especially for punching design.

Using beam elements might seem easier for the design of the wall pillar. But for column dimensions width/depth >>2 the distribution of the forces might not be realistic. In addition, the large point load caused by the beam element is a singularity and will result in high drilling moments in the slab.

On the other hand, if a small wall is modelled with quad elements you have to check that the punching areas do not overlap, because the program might integrate the forces of the wall ends twice and then the reinforcement will be too large.

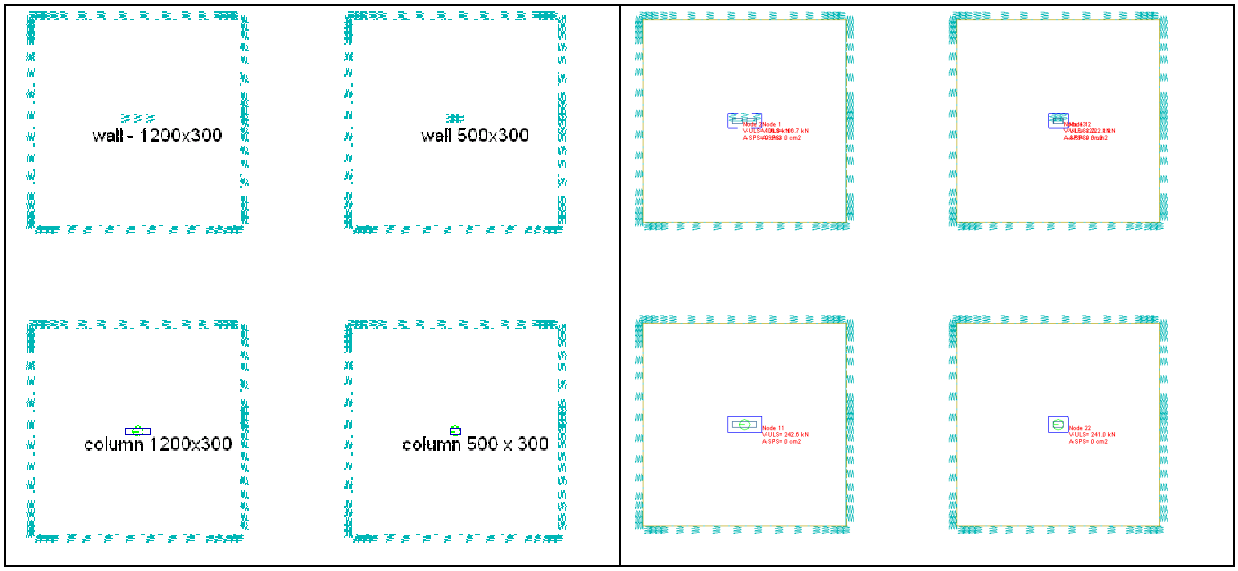


Figure 9: example in principle comparison different wall models - results

4.3 Meshing

Normally the meshing will be done automatically by SOFiMSHB. Nevertheless meshing is influenced by user because border conditions are user defined. So the quality of mesh (and results) depends on mesher and quality of model.

4.3.1 General hints for system generation

- define group divisor as small as possible
- if possible, use elastic supports to avoid singularities
- don't model too detailed – otherwise you will get a lot of elements and large calculation times, but not necessarily better results
- create your system with equal attributes for all elements at first, afterwards you can modify them.

4.3.2 Hints for meshing with SOFIMSHB

- define clever boundary conditions (e.g. insert some structural lines to "help" the automatic mesh generator on edges with an angle >90°)
- use features like column macro (refines the mesh around a column if there is no structural line on structural point)
- define real dimensions of the column head (may avoid punching problems if real dimension differs too much from the default value; for further information look at the bemess manual - chapter punching)

- define the origin in your model (i.e. do not use Gauss-Krueger coordinate-system or similar) to avoid numerical problems –big coordinates reserve many digits which can cause computational inaccuracies if calculating a system with small dimensions
- if you need a finer mesh, first refine locally (e.g. by adding some structural lines), then globally.

5 Workflow in SSD

1. system definition (Task Systen Information) in SSD
2. define materials
3. define cross sections
4. GUI for Model Creation (SOFiPLUS-(X))
5. Linear Analysis
6. Define Superpositioning (if necessary)
7. Compute Superpositioning
8. Design Parameters of area elements
9. Design ULS - area elements
10. Design SLS - area elements
11. Design ULS – Beams
12. Design SLS – Beams
13. [further tasks i.e. for 2nd order theory analysis] (not part of this tutorial)
14. create documention (not part of this tutorial)

Because working on a project is an iterative process, you'll sooner or later have to change something in your project. Then make your changes an rerun the system from the changed task till to the end.

6 Tutorial example 3D mulit storey office building

6.1 Create new SSD project

To make the input of the model easier start with a 2d-system and change to 3d later.

Select the option “Graphical Pre-processing”.

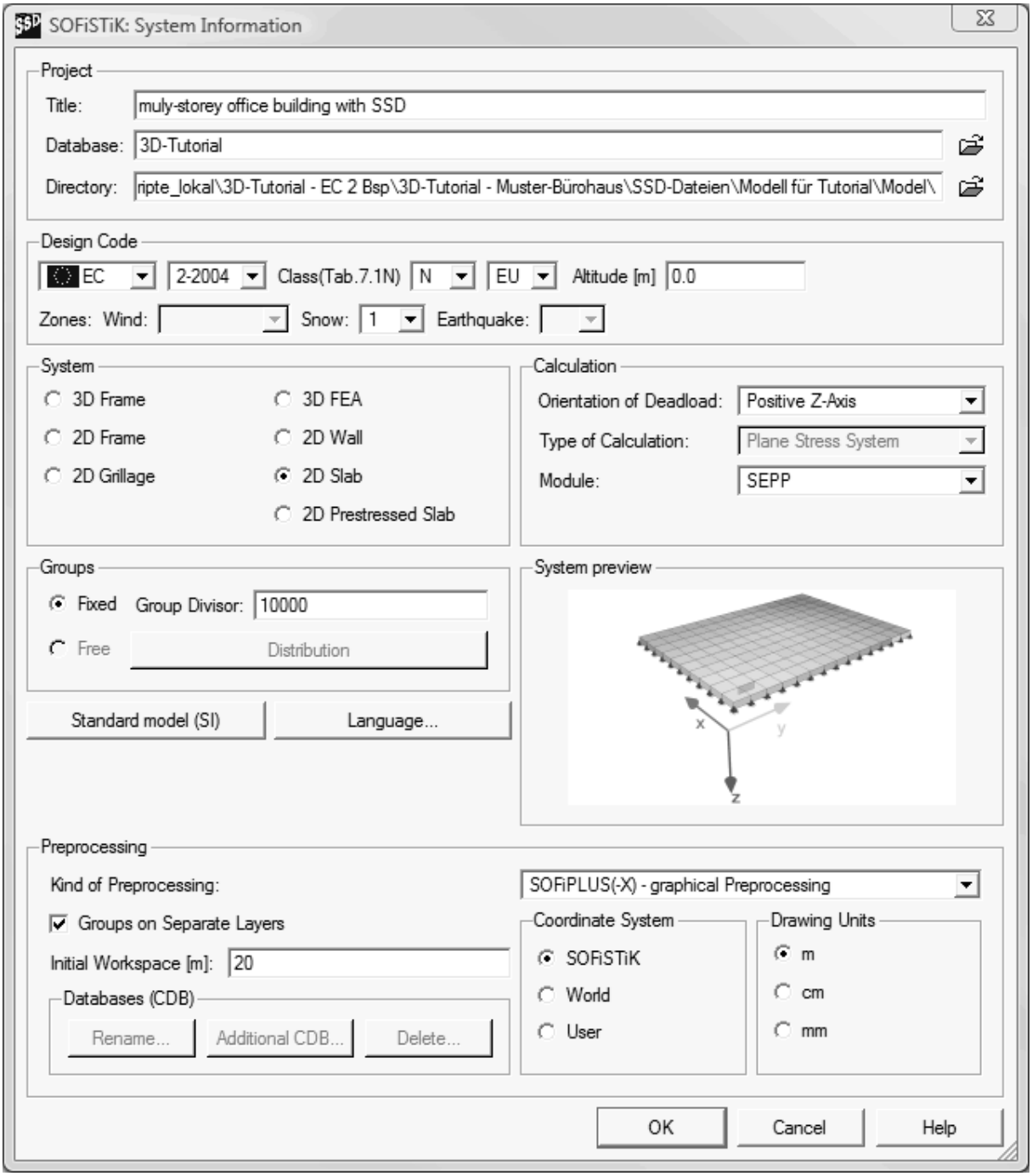


Figure 10: system information (dialogue)

Take care, that the name of the database is the same as the [name].dwg, else a new *.dwg with the name of the database will be created.

6.2 Define materials and cross sections

Define the materials and cross sections you need for the project:

Number	Kind of cross section	Concrete Nr.	Reinforcement
1	Rectangle CS	1	S500 two-sided
2	Circle	1	S500
3	T-beam	1	S500 two-sided

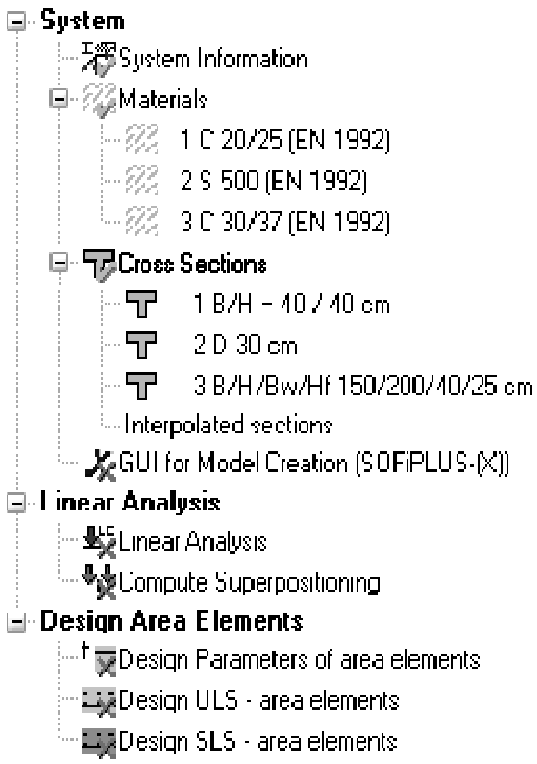


Figure 11: Materials and cross sections

Materials and cross sections can be added and modified later.

6.3 Graphical input of system and loads with SOFiPLUS(-X)

6.3.1 Input of the first floor in 2D

Click on „GUI for Model Creation“ to open SOFiPLUS(-X) and the *.dwg with the architect’s plan.

Before starting to work, please check where the origin of the system in the architect’s plan is. If it is not yet in one of the corners of the slab, please move the slab into the origin, using the AutoCAD **command: move**. This will not only make work easier, but also prevent large numbers for the coordinates, which might cause trouble (see chapter 4.3.2).

Create all system lines for the ground plan with the commands **command: Copy to structural Layer, command: Center Line** and **command: 1/3 Line** from the draw menu.

Take care that all system lines are on the layer X___AUFL (SOFiPLUS(-X) Layer for system lines).

Note that system lines are not equal with structural lines! System lines are used as help lines for creating the structural elements.

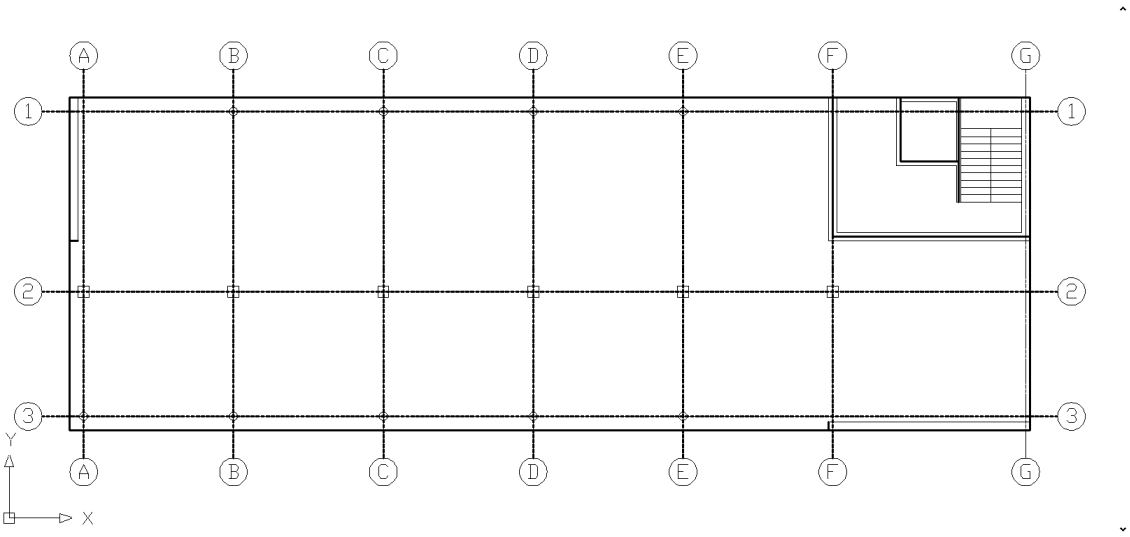


Figure 12: ground plan with system lines and axes

Next, switch off all layers except the one with the system lines (X__AUFL).

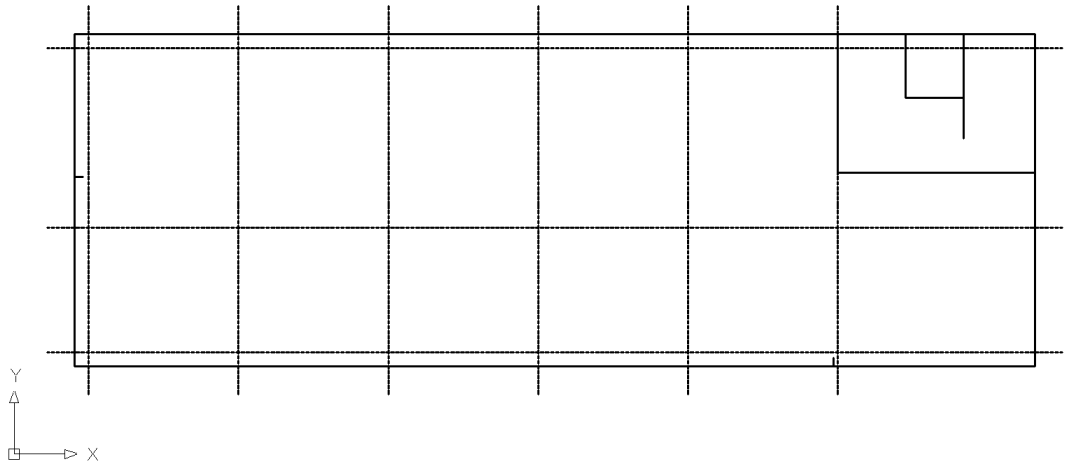


Figure 13: layer with system line only

Now you can start to create the structural elements, starting with the structural lines.

This tutorial wants to show you, how to work efficiently. It is just a recommendation. There are many other ways you can get the same results as well.

Command: Structural Edge

First select all system lines and then deselect (shift + left click on elements) the few ones you do not need, like the lines in axis 1,3,A and the excess of the axis at the exterior walls. When you are done, apply with enter.

Alternatively, you can first create structural lines of all system lines and then delete the ones you do not need.

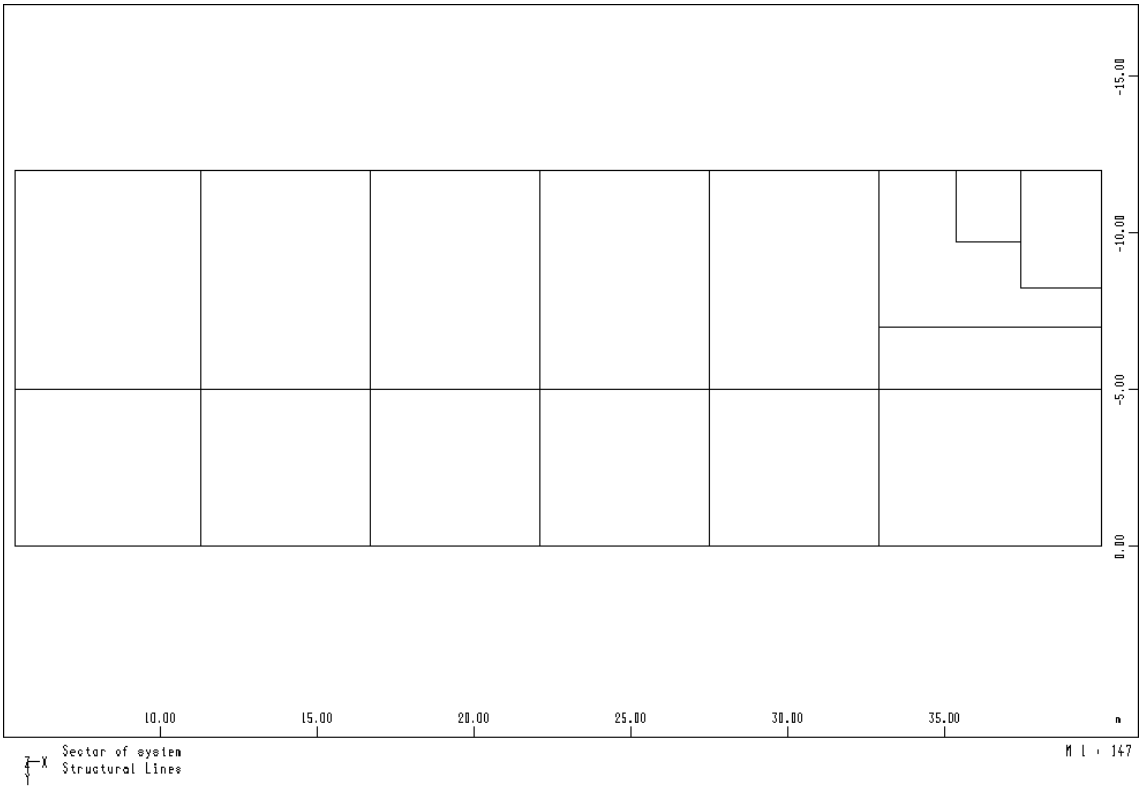


Figure 14: System with structural lines (wingraf plot)

Command: Cure Edge

This command is used to merge two structure lines with a common structure point to one structure line. The common structure point will be removed. Check your system if all structure lines you have created are the way you want them or if you have to remove some intermediate points. If you have unnecessary structure points in your system you will get a higher mesh density than needed (results higher calculation times).

> Because the facade will not be modelled, you have to use load distribution areas (called LARs) to apply the wind loads on the structure. LARs only act on beams and therefore you have to create dummy beams, which transfer the loads to the slabs.
>
> The load distribution areas applys the free loads onto the beam system. The free loads are converted into equivalent beam loads automatically during export to .cdb. The LARs act like a layer of quad elements with no stiffness.

Command: Cross Sections

Define a new rectangular cross section with nearly no stiffness (very small cross section). Fill in the data shown in Figure 15.

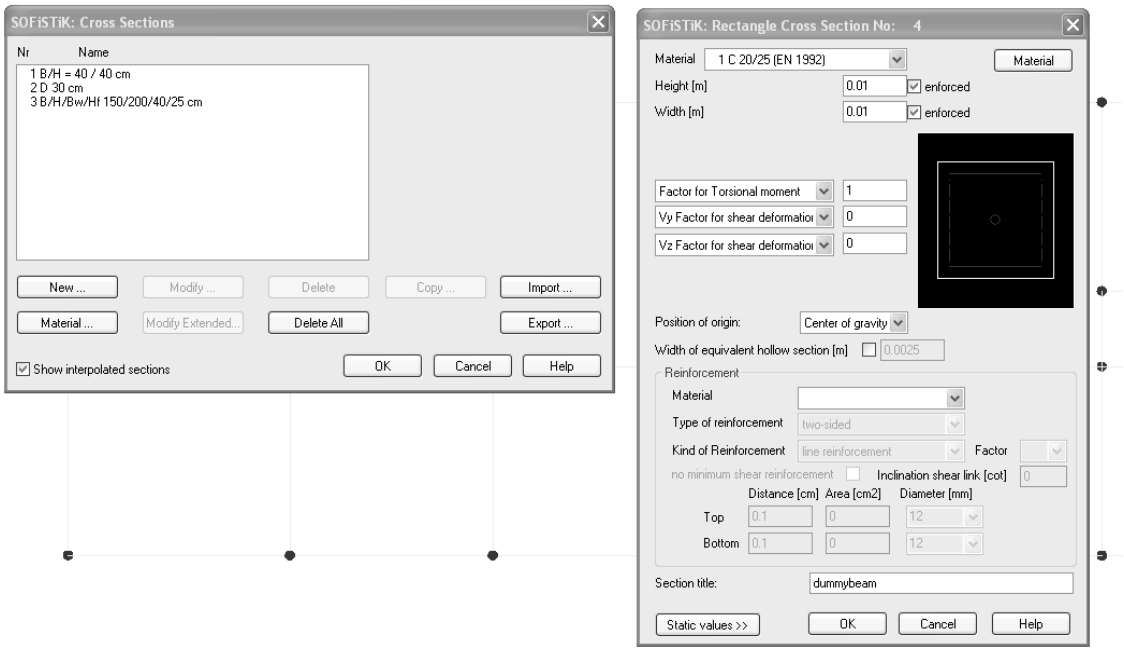


Figure 15: creating cross section for dummy beams

Command: Modify Structure Edge

Now you can assign the dummy cross section to all outside edges. Click on command „modify structure edge“ and select all outside edges. On the tab „beam/ cable“ select the option „Centric Beam“ and select the dummy cross section. Click on tab „General“ and change the group number (in this case to 49[1]) to have all dummy beams within one group. This is important, because later you have to tell the LARs on which group they will act. Confirm your input with OK.

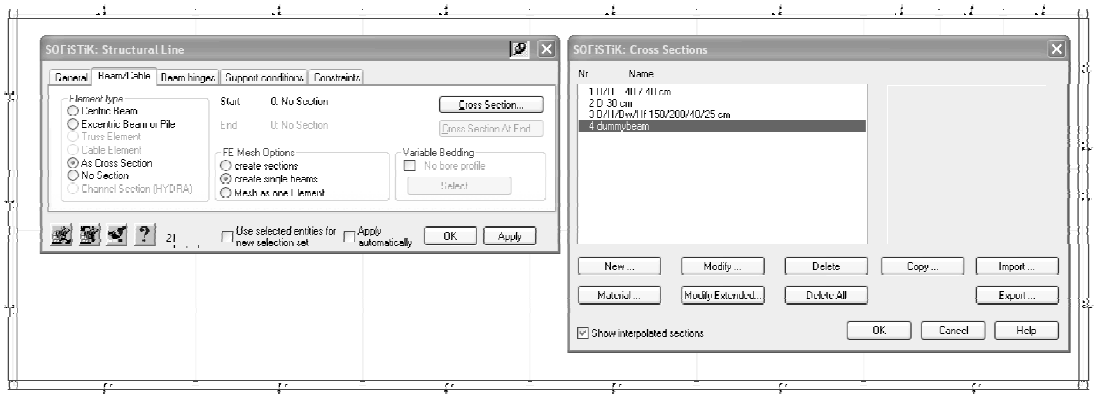


Figure 16: Assign dummy beam to structure lines

Command: Loadcase Manager

In the loadcase manager generate load case 1 and set the factor „EGZ“ for this load case to 1, i.e. the self-weight of the structure will be calculated automatically by the program using the geometry of the model and the material values.

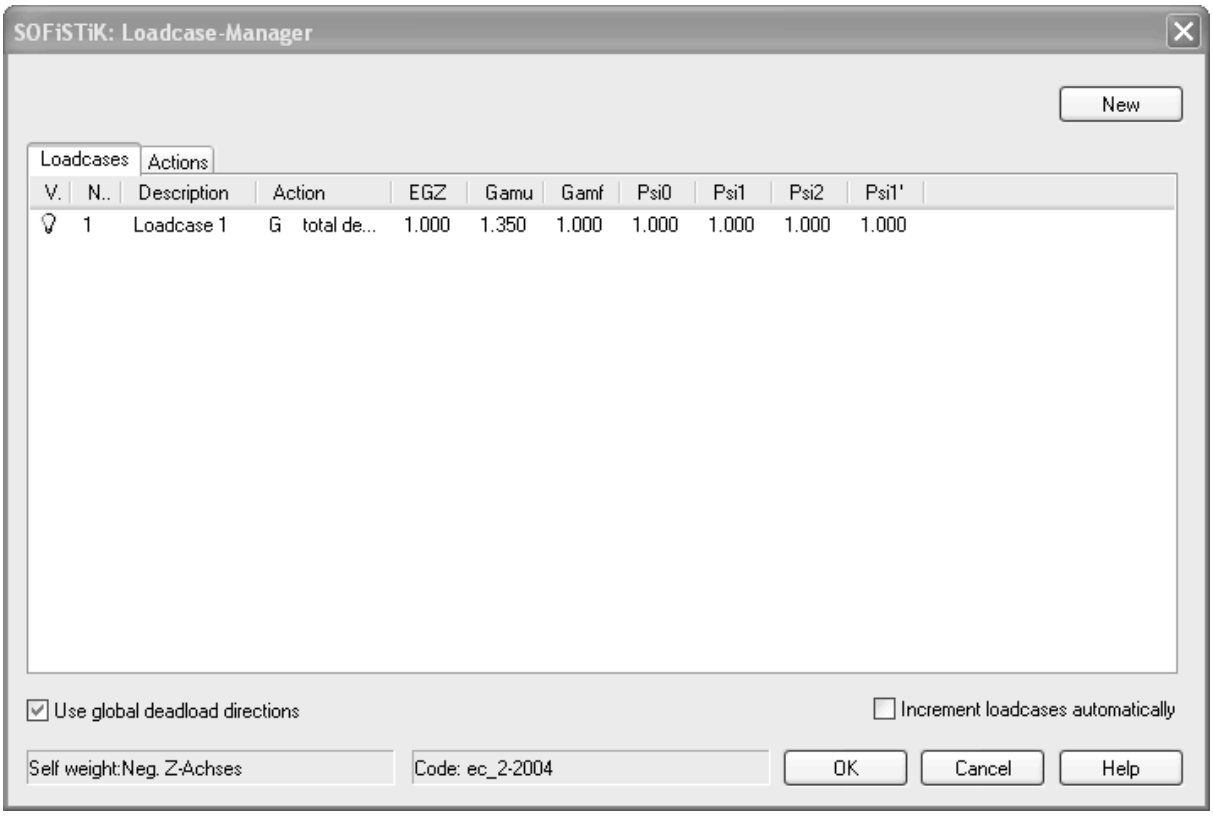


Figure 17: Loadcase Manager

Recommended: Create all necessary action (and load cases) at once.

[1] Number 49 because highest group number for cross sections is 50; so 49 is a free number and not used for “real” cross section groups

Command: Structure Area

On the tab „General“ put in all the data as shown in Figure 18. Then click with your mouse in the drawing and select the option „pick point in area“ and click into area 1.

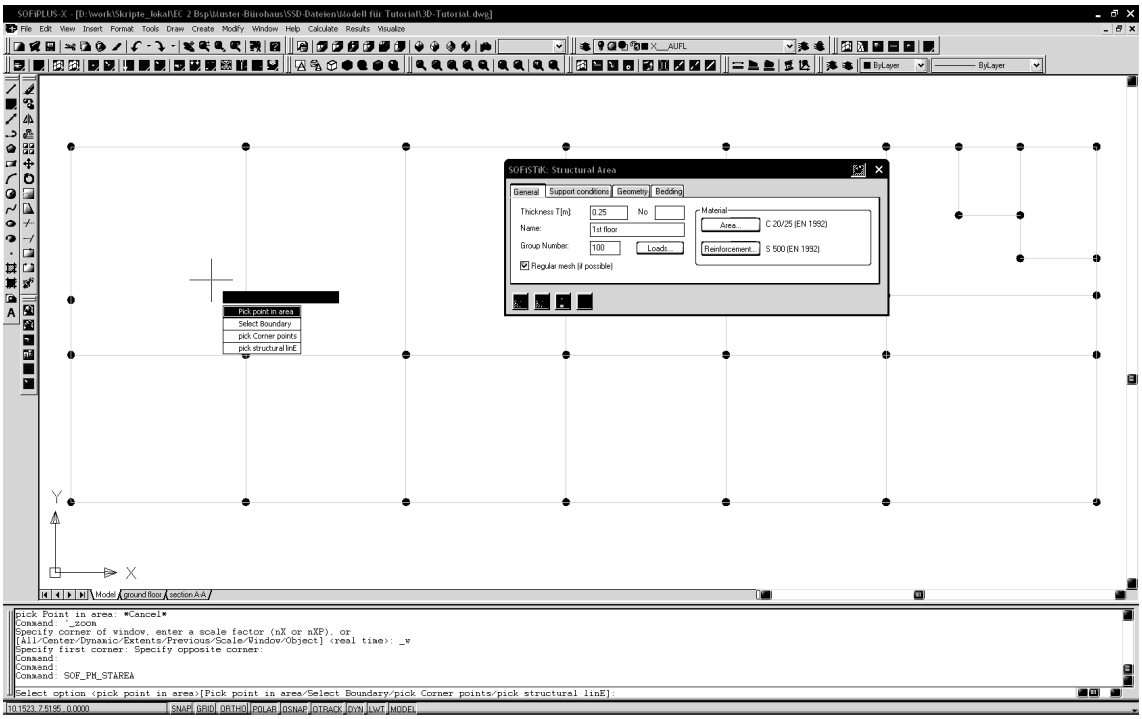


Figure 18: Creating a structural area

> The option “pick point in area” for the command “structural area” can be used for 2D-systems. In 3D-systems it is only available for the current plane.

A window called „Loads on Slabs“ will open automatically. Click on „Select Load Case“. Create and select load case 2 (action G) for dead load. Then insert the load of 1.2 kN/m² as load in gravity direction. Do the same for the imposed load: load case number is 101, action Q, load of 2 kN/m². Please make sure, that the fields „Increment loadcases automatically“ and „Do not show this again“ are NOT selected, before confirming your input with OK.

> This automatism for the load cases is only available if you click into the structural area first. (and not on the load button)

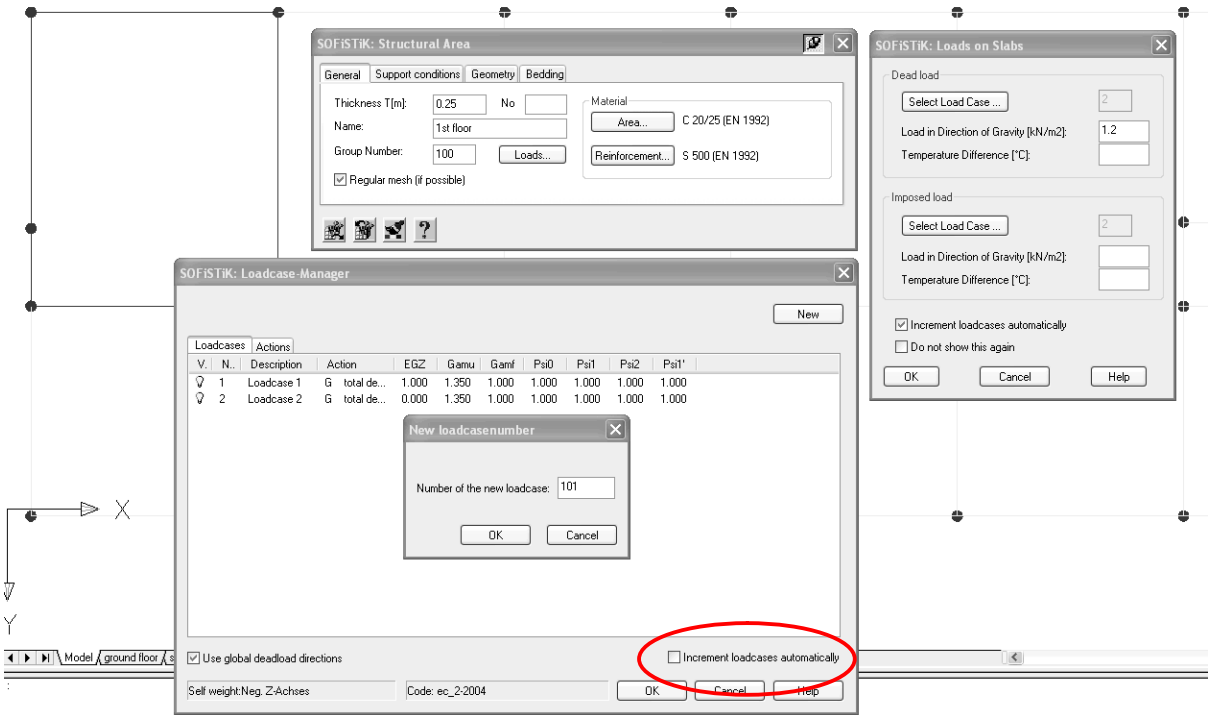


Figure 19: Loads on Slabs

The command „Structure Area“ is still active (you can check that if you look at the command line – Sofiplus(-X) is waiting for further input in this command). Now you can create all other structure areas analogously (Clicking into a new area and so on…). Create a new loadcase for the imposed load of each area, i.e. lc 102 for area 2 and so on. The imposed load on the slab of the stairway is 5 kN/m² instead of 2 kN/m².

The loads that are defined on the structure areas of this floor will be copied to all other floors together with the structure elements. This will result in less load combinations, assuming that the leading distribution will be the same on all floors.

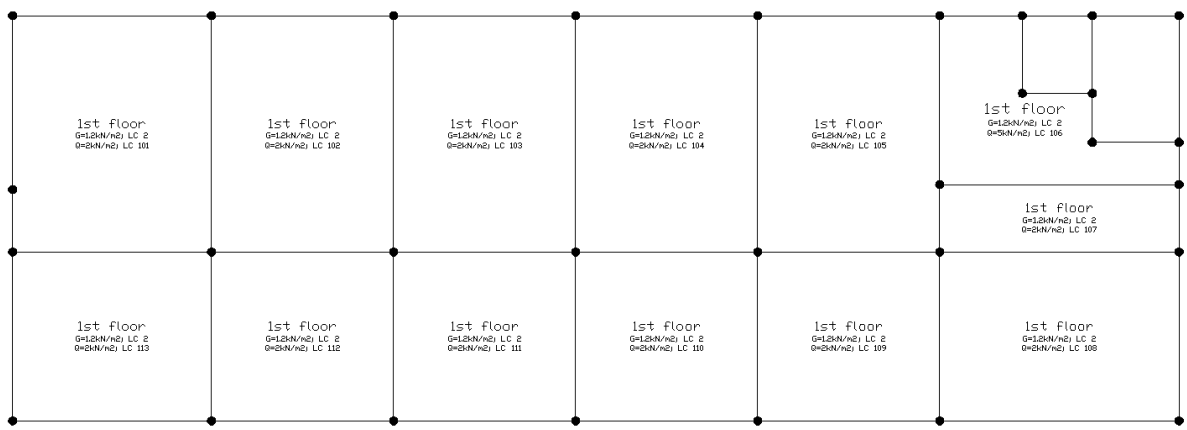


Figure 20: System with structure areas

Command: Structural Point

Next, you should define structural points at those places where there will be columns in the 3d-model. Therefore, switch on the layers X_AUFL and Axes. Select command structural point and click on the intersections of axis 1, 2, 3 and A-E, F2; but not A1. Finish the input with enter. Switch of the two layers X_AUFL and Axes.

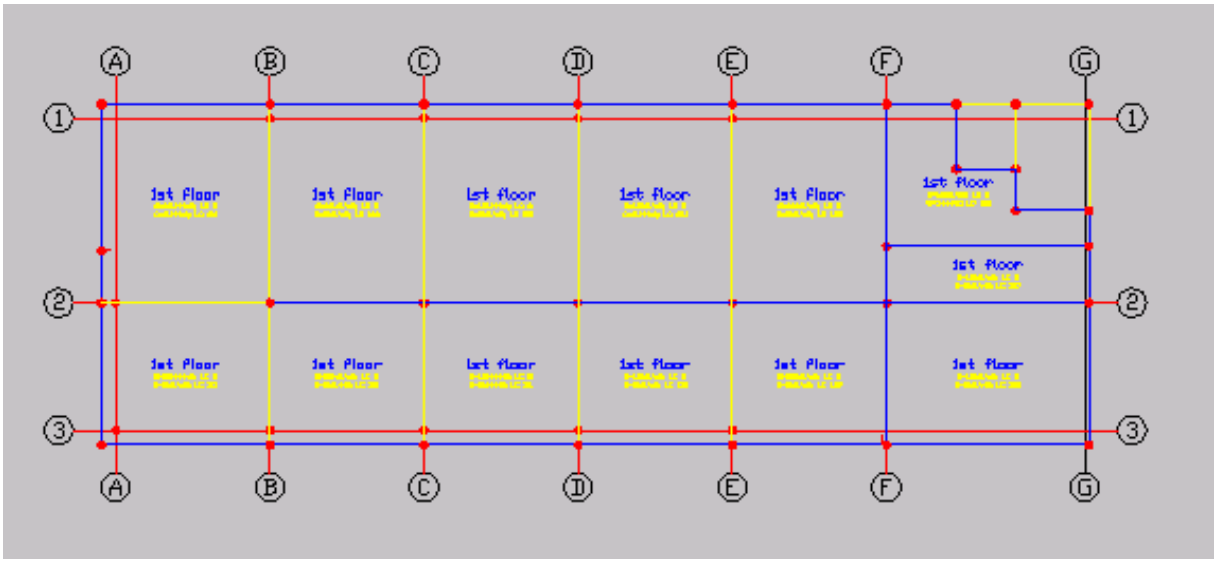


Figure 21: Defining structural points for columns

Because the input for the 2D-system is finished, you should check your file for errors and export it to .cdb before changing to a 3D-system.

Command: audit

In the command line type _audit and then confirm the question with y(es). The drawing will be examined and some detected errors will be fixed.

Command: Export

When exporting the system to the SOFiSTiK-database *.cdb the program runs an automatical mesh generation. Usually you do not have to make adjustments and use default settings. Afterwards you should check your system and the mesh with the ANIMATOR.

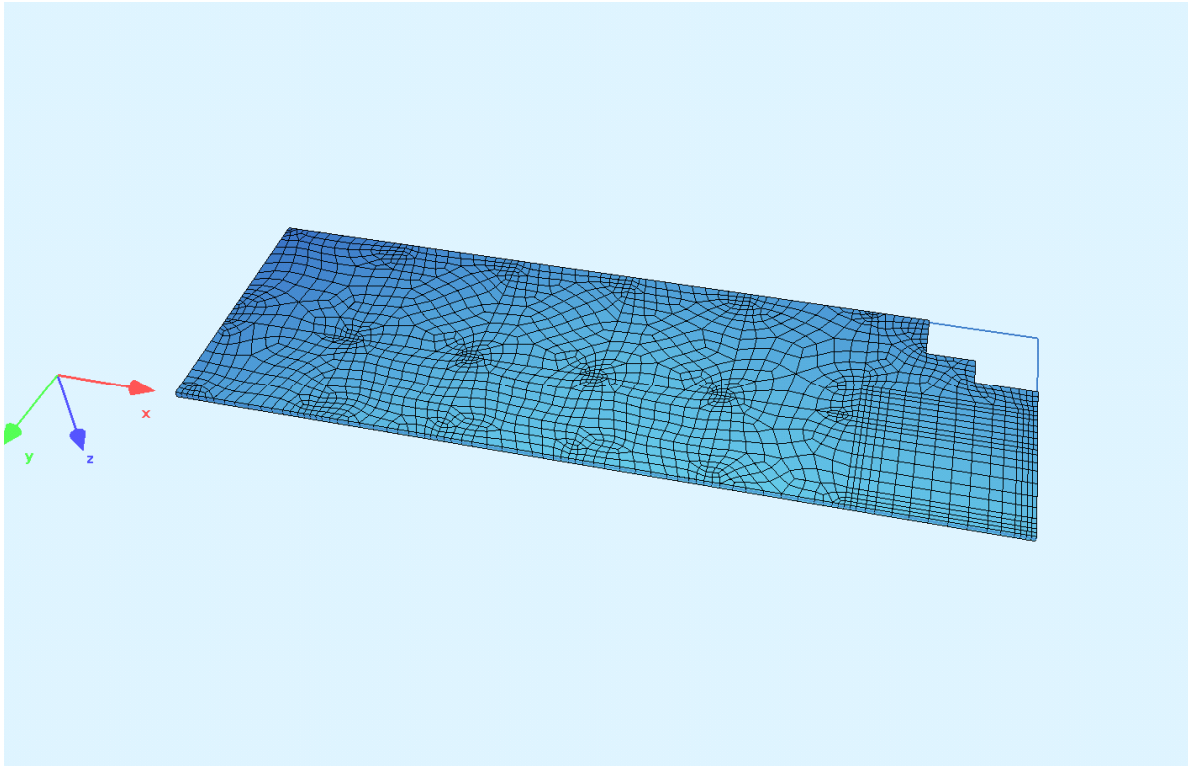


Figure 22: Generated mesh of first floor

If the export of the 2D-system worked properly, you can now change to the 3D-sytem.

6.3.2 3D Modelling

Command: System Information

Change the system information in SOFiPLUS(-X) from 2D slab to 3D FEA.

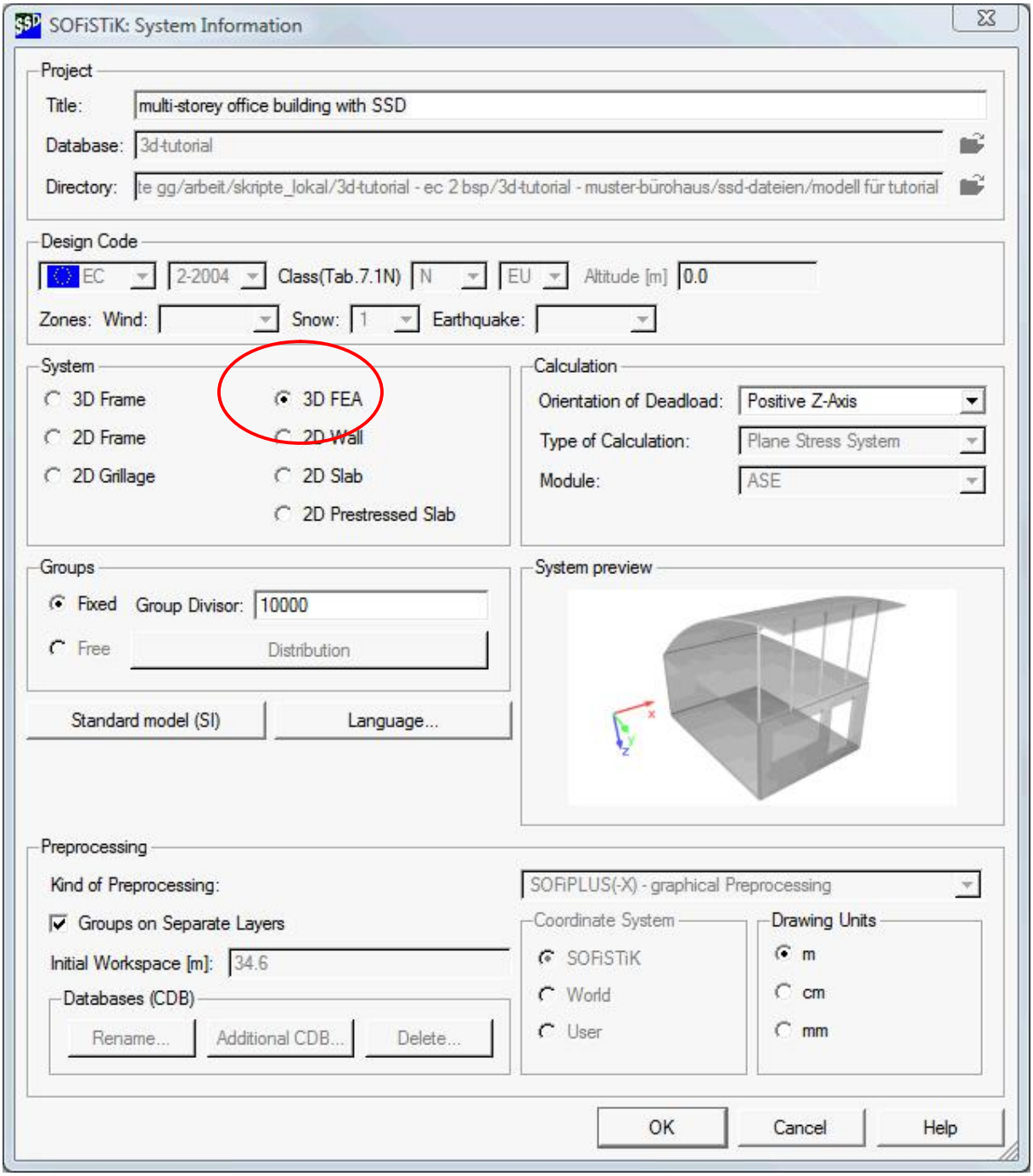


Figure 23: Change system information

For the 3D-system it is recommended to use 3 viewports simultaneously. Go to Menu View > viewports > 3 viewports and choose „right“. Set viewport 1 to „top“, viewport 2 to „left“ and viewport 3 to „SW isometric“.

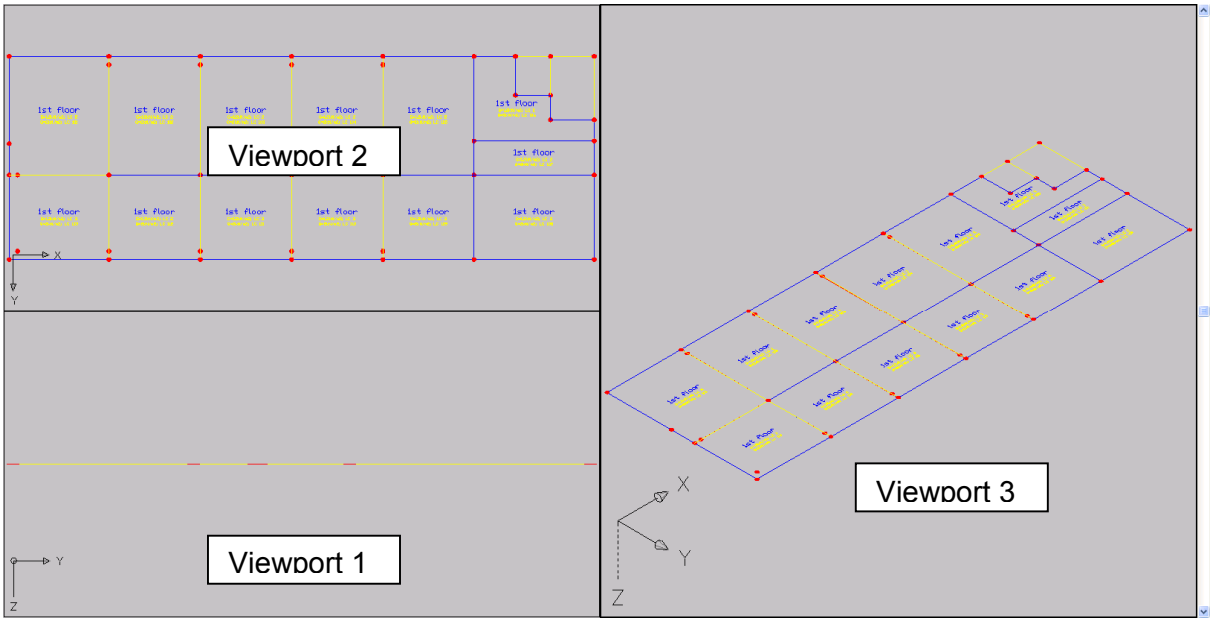


Figure 24: Workspace with 3 viewports; numbers of viewports

6.3.2.1 Creating upper floors

Command: copy

Next you can copy the whole system of the first floor to all other floors. To do so, use the AutoCAD **command: copy**. Select the structure and specify the origin as base point. Specify all further points by their coordinates (0,0,-3.5; 0,0,-7; 0,0,-10.5; 0,0,-14). Finish the input with return key.

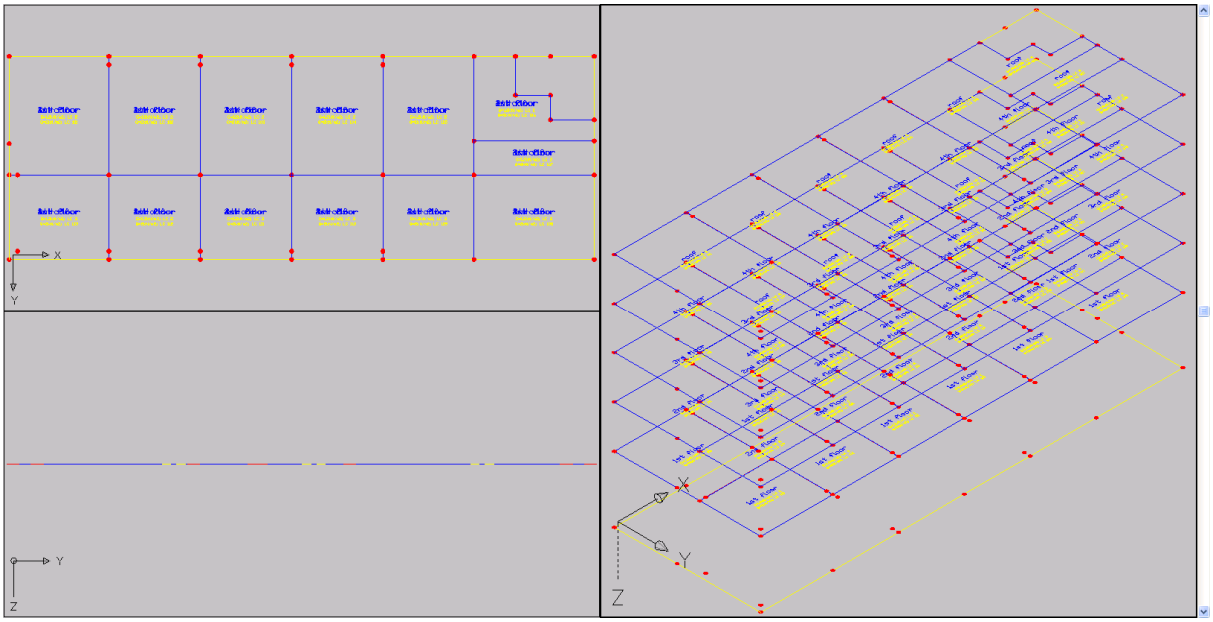


Figure 25: System with all floors

Command: Modify Structure Area

In the viewport 1 you can easily select the second level. Change the name of the structure areas to 2nd floor and the group number to 200 as shown in Figure 26. Modify all other floors analogously.

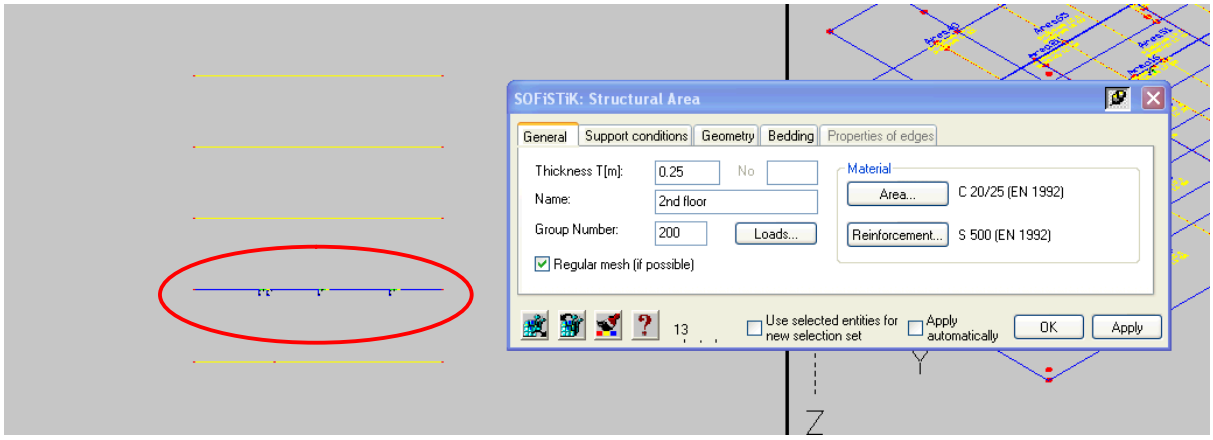


Figure 26: Modify structure areas

6.3.2.2 "Modelling" bottom plate

In this example, the bottom plate will not be modelled. To make sure that the wind loads can be applied on the system correctly, the dummy beams have to be copied to the ground floor.

Command: Display Selection Set

Select the 1st level in viewport one. Then use Quick Select option in the right click menu to select all dummy beams of the first floor (structure lines, group 49).

Command: Copy

Copy selected dummy beams to the ground floor (base point = origin, second point = 0,0,5.375).

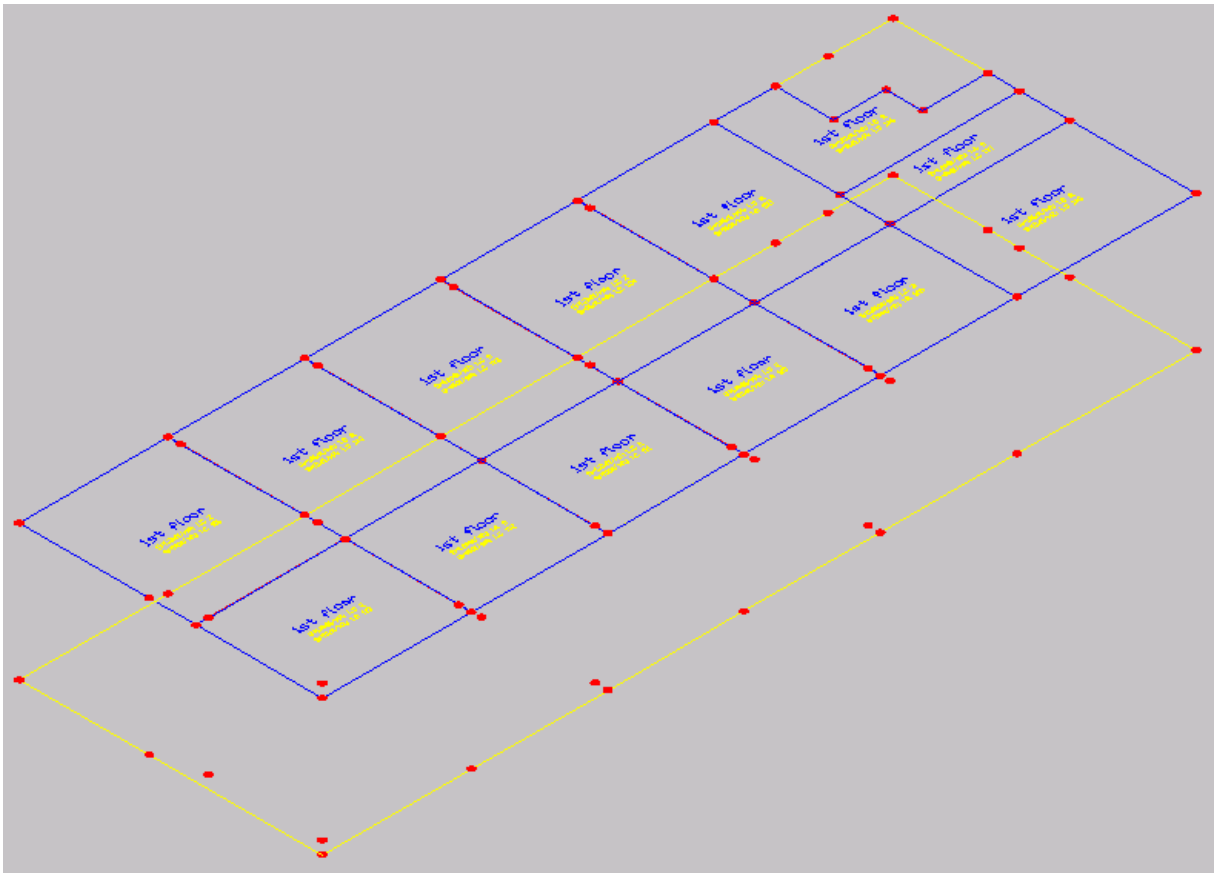


Figure 27: 1st floor and ground floor with dummy beams

Command: Display All

To undo the command display selection set, just click on the “display all” button.

6.3.2.3 Creating columns

In the next steps the columns of the building will be created: first the ones for the upper levels and than the ones in the ground floor.

Command: Display Selection Set

Select 1st level and the roof in viewport 1.

Command: Structural Edge

Draw structural edge from 1st floor to roof and assign cross section number 2.

Command: _copy

Copy the column to all required places with the AutoCAD command _copy.

Repeat the last two steps to create the quadratic columns with cross section number 1 in the middle axis analogously. Please note that the program automatically splits the structural edges on all floors.

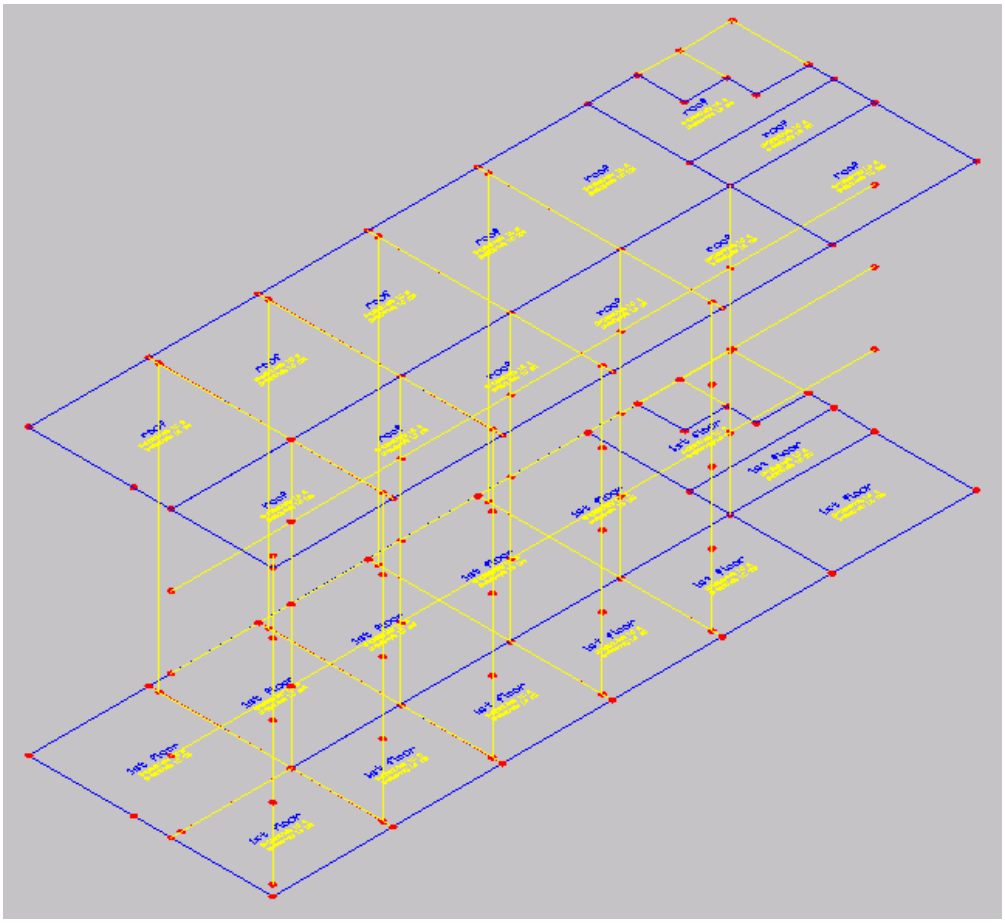


Figure 28: Create columns in all upper floors

When you are ready, you can display all again and run _audit command.

> We strongly recommend using the _audit command every now and then to check (and repair if possible) the input for errors. Alternatively for using the command line, you can call the command in the menu File > Drawing Utilities as well.

Command: Display Selection Set

In the viewport down left select the 1st level and the ground floor.

Command: _copy

In viewport 2 you can easily select all points where columns are below (see blue grip points in Figure 29). Copy them to the ground floor.

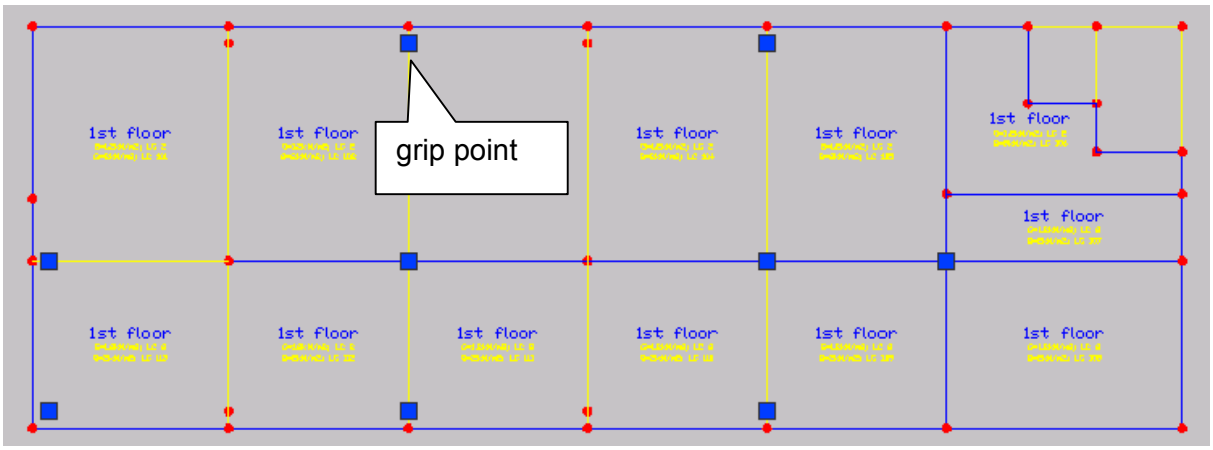


Figure 29: Copy points for columns on ground floor

Now you can create the columns on the ground floor analogously to the ones on the upper floors. When you are done, you can display all again.

Then you should assign the correct group number to the columns. According to the group concept the group number is equal to the cross section number plus the level number x 100. (see also at chapter 4.1.3 Considerations about groups)

Command: Quick select

In the right click menu is a button for selecting objects graphically. Select one floor. Choose object type “structure line” and with the property “cross section ” you can select all columns of one cross section for this floor.

Command: Modify Structure Edge

Enter the appropriate group number on the tab “general”.

Repeat these two steps for all columns on all floors.

Next you should make the settings for the beam hinges.

Command: Quick select

All columns have a cross section number >= 0 and <3. To select them, you can use the command quick select twice in succession by using these properties.

Command: Modify Structure Edge

To model the columns with a hinged connection to the slabs, please select MY and MZ on the tab “beam hinges” for both, start and end of beam.

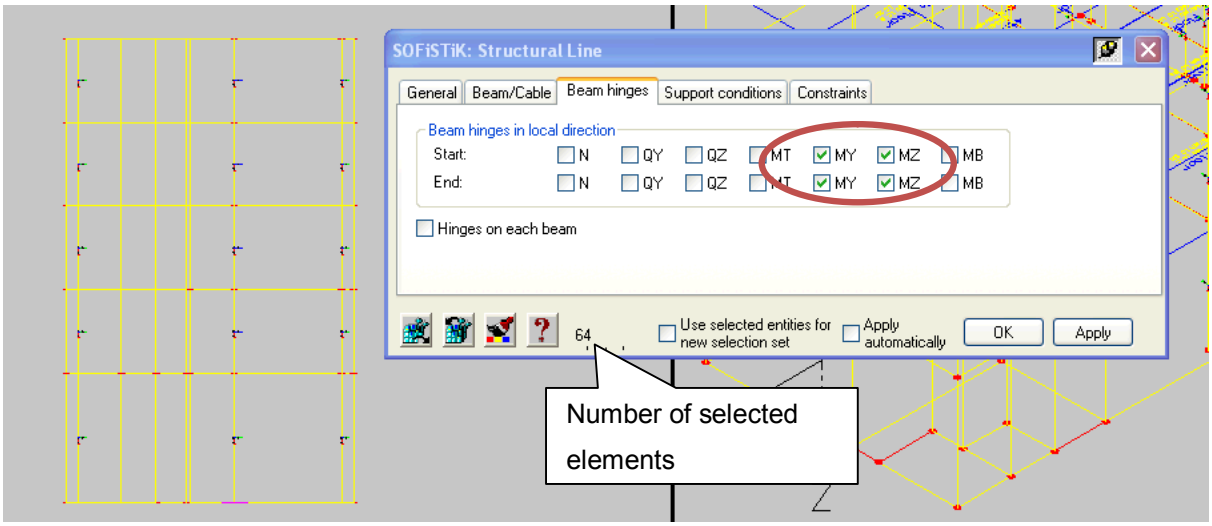


Figure 30: beam hinges

On the bottom of the window “structural line” the number next to the question mark tells you how many elements are selected.

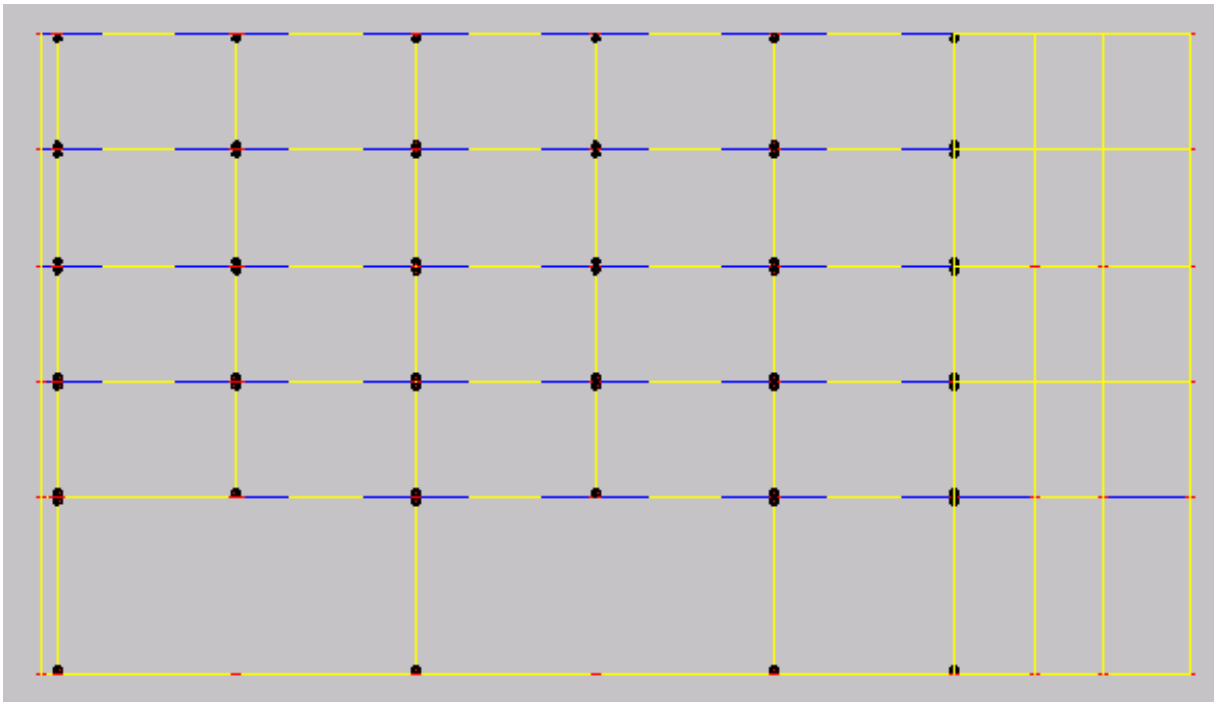


Figure 31: quick graphical check of column hinges

Now you can assign the data for the column heads to the structural points. In the viewport up left you can easily select all columns with the same cross section. To save time, it is recommended to do these settings for all structural points first, and then delete them for the ones on the bottom.

Command: Modify structural point

For punching design add the shape of the column and its dimension.

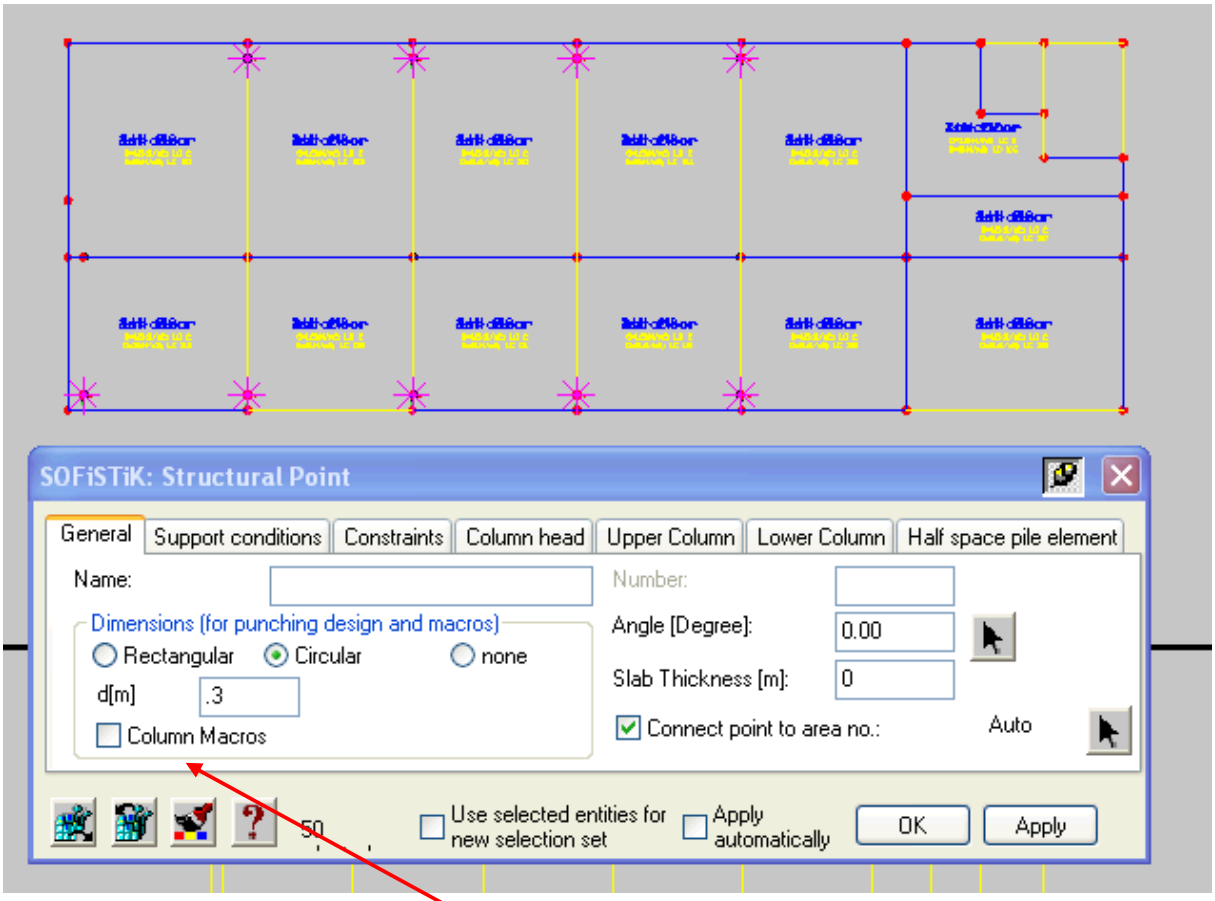


Figure 32: define punching dimensions

You do not need to activate the option “column macros” if the structural point is on a structural line.

6.3.2.4 Creating walls

The next steps will show you how to create walls for the building. Start with the walls on the ground floor.

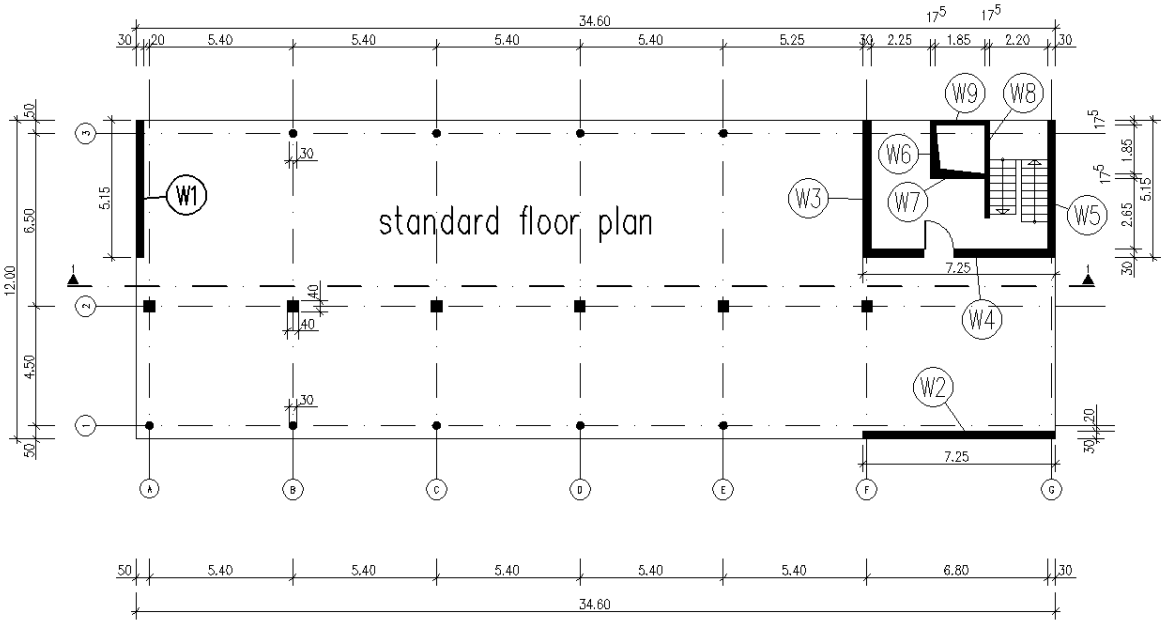


Figure 33: Wall numbers

Command: Create structural area

Use the option “pick corner points” of the right click menu to define the area.

If you pick the points anti-clockwise the local z-axis of the elements will always point away from you. This is important for the interpretation of the reinforcement (upper/ lower layer).

In the input window, fill in the thickness of the wall (wall 1 to 5: 30cm, wall 6 to 8: 20 cm), its name and the group number (key word group-number: see also chapter 4.1.3. Considerations about groups)

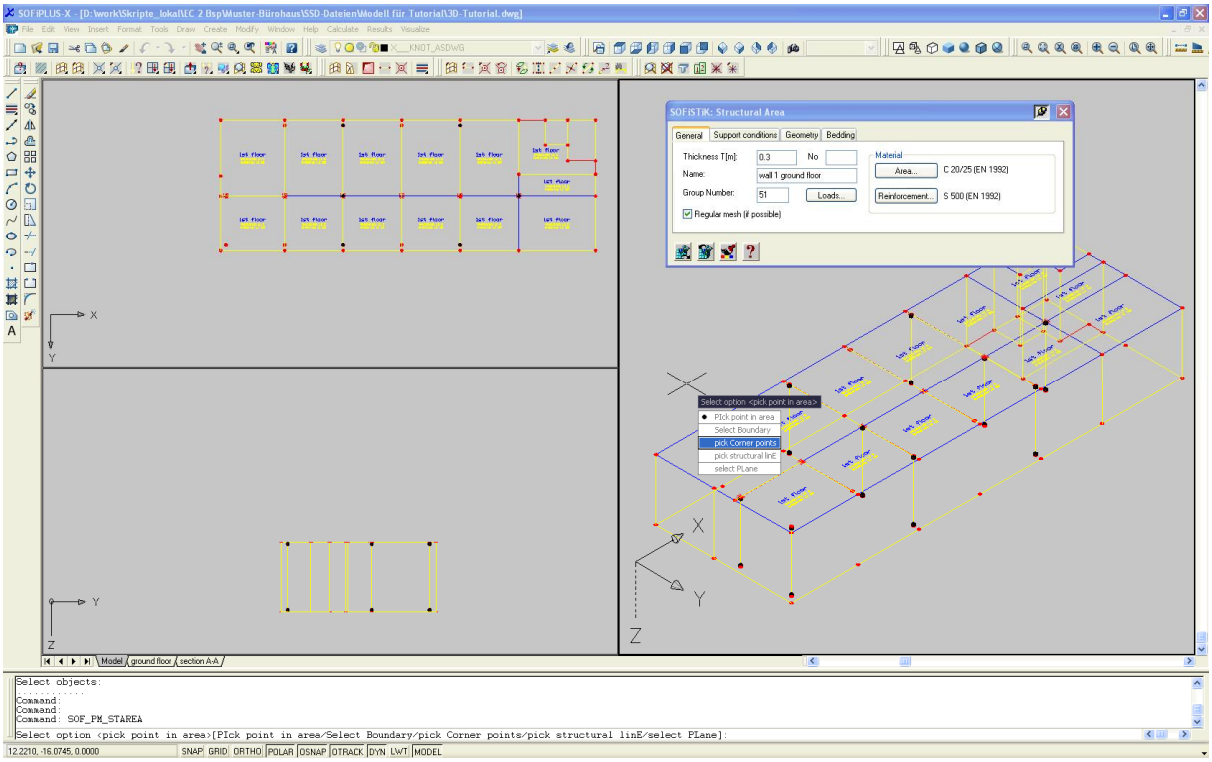


Figure 34: Creating wall number 1 on the ground floor

Command: Modify structural area

On the tab “Properties of edges” (see Figure 35) make the settings for the cinematic constraints at the intersection between walls and floors. (for details look at chapter 4.2.1 Connection walls/ columns – slabs)

It is important to make these settings before copying the walls, because if you want to adjust it later, you can only do it for each wall separately; if you copy it each wall comes into the right contraint.

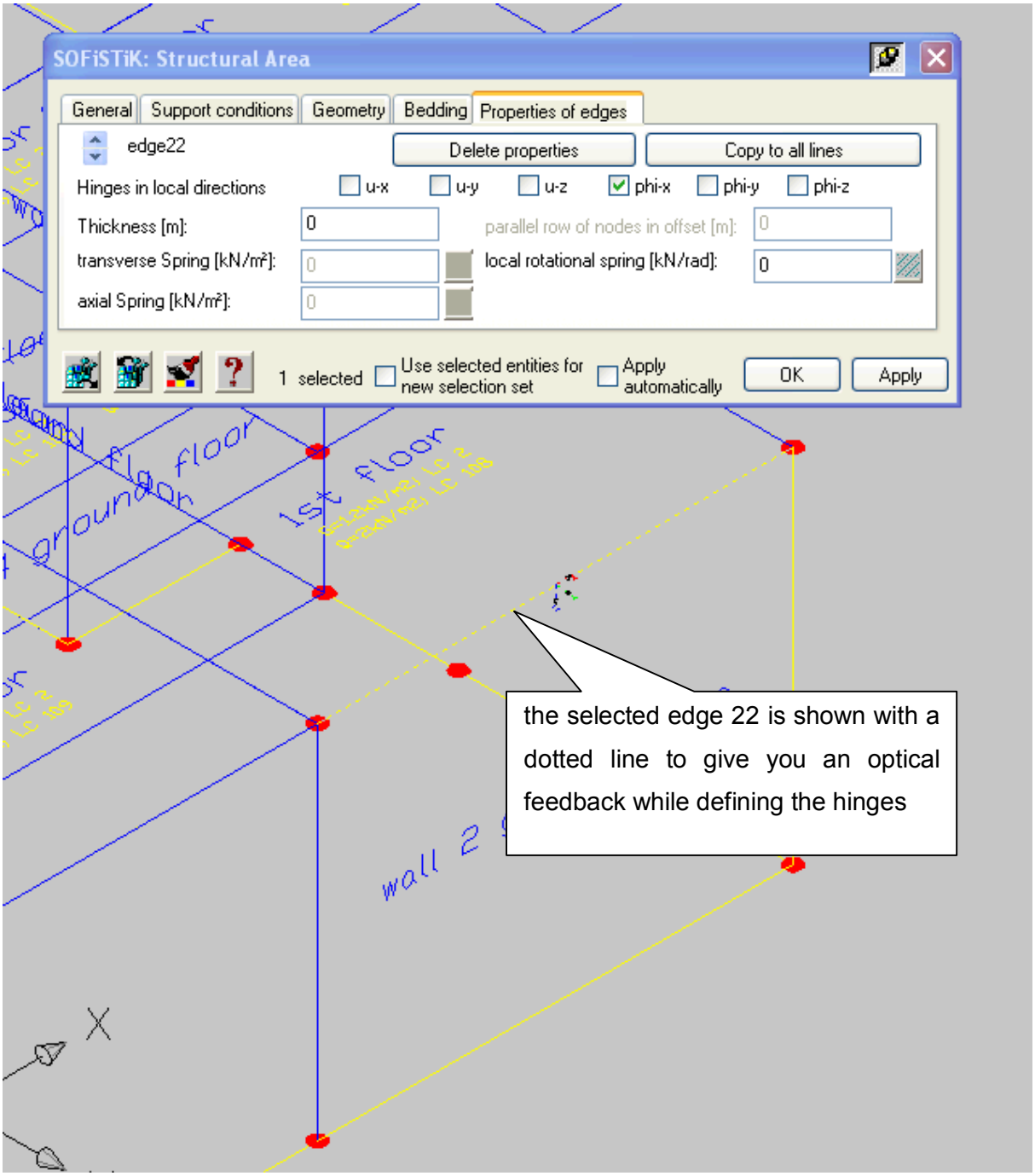


Figure 35: Cinematic constraints at the intersection of wall and floor

The walls on the first floor can be modelled analogously. You can copy the walls from the first floor to all other floors, because they have the same height. Don't forget to adjust the names and group numbers of the elements.

It is strongly recommended to check the model with _audit after the input of the walls.

Make an export and check the system with animator. There you can see, if all walls have different group numbers (select "view control > colour options > change of colour > per

group" that each group is displayed in a different colour). Also the hinges (yellow symbol) on the top and bottom of the walls are visible.

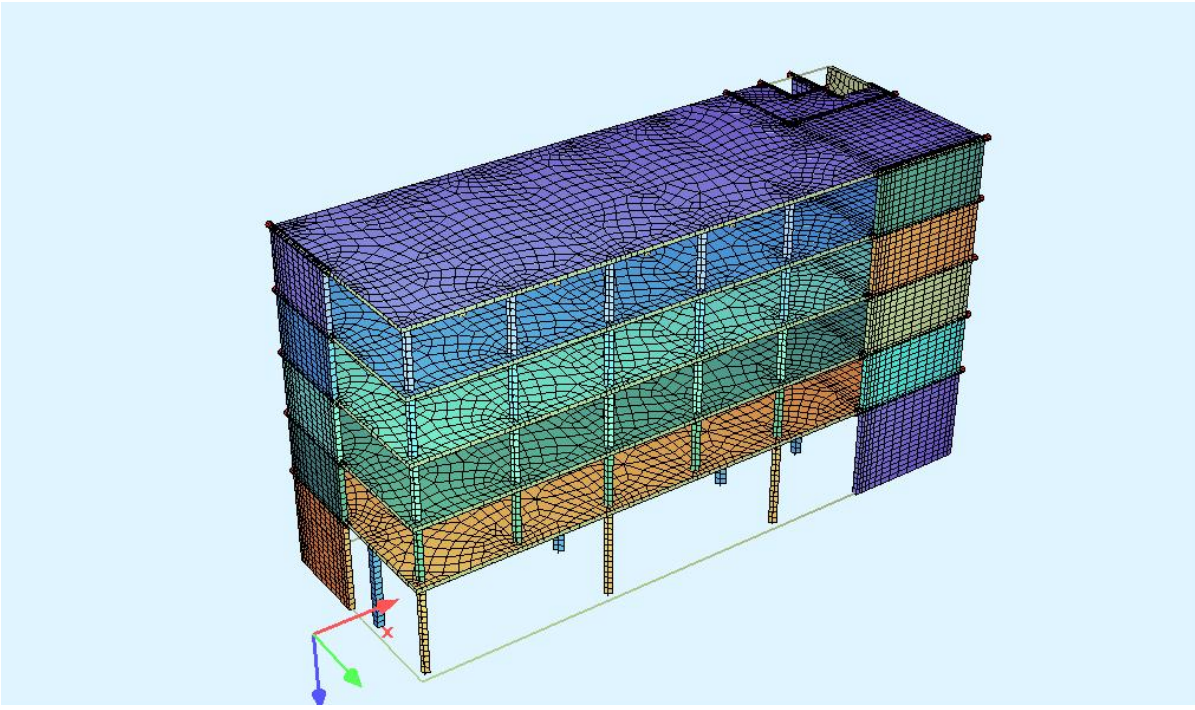


Figure 36: Check system with ANIMATOR

6.3.2.5 Ground floor - supports

As mentioned before, the foundation will not be modelled but the supports are assumed to be rigid. (for this tutorial this is OK; in reality usually not)

Command: Modify structure edge

Select all lines on the ground floor and select PXX, PYY and PZZ on the tab "Support conditions".

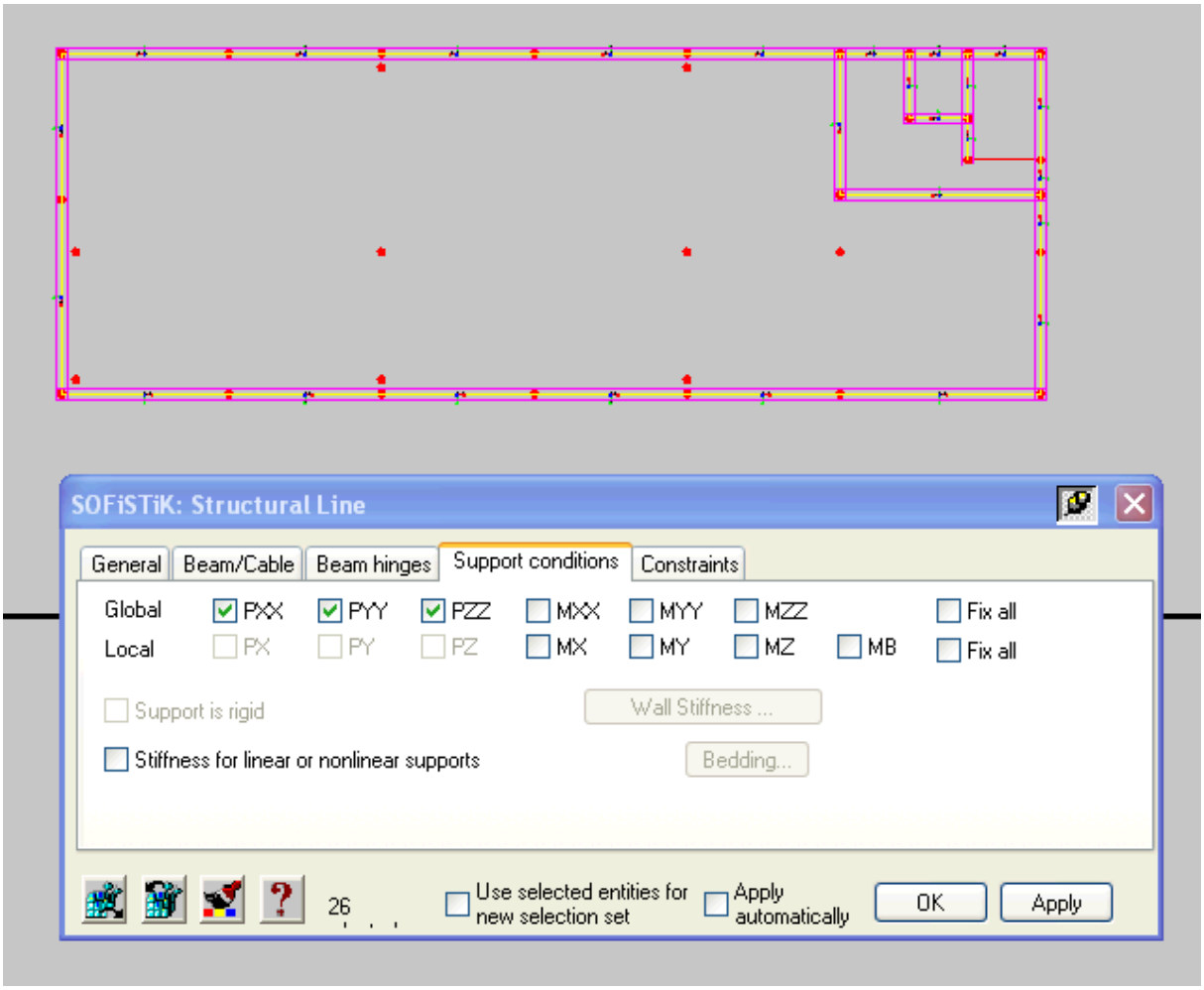


Figure 37: Support conditions – structural lines

Command: Modify structure point

Select all points on the ground floor and select PXX, PYY and PZZ on the tab “Support conditions”.

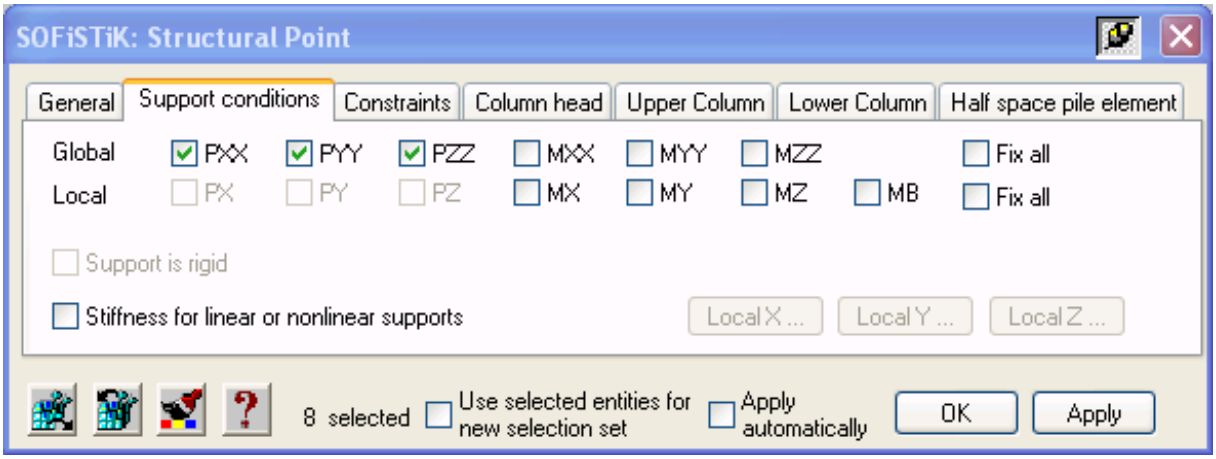


Figure 38: Support conditions – structural points

6.3.2.6 Defining T-beams in slab over ground floor

The slab above the ground floor is stiffened with t-beams.

Command: Display selection set

Select the 1st floor in viewport 1.

Command: Modify structure edge

Select the structure lines on axis 2 (see Figure 39: T-beam on axis 2). In the tab “beam/ cable” chooses “Centric beam” and assign cross section number 3.

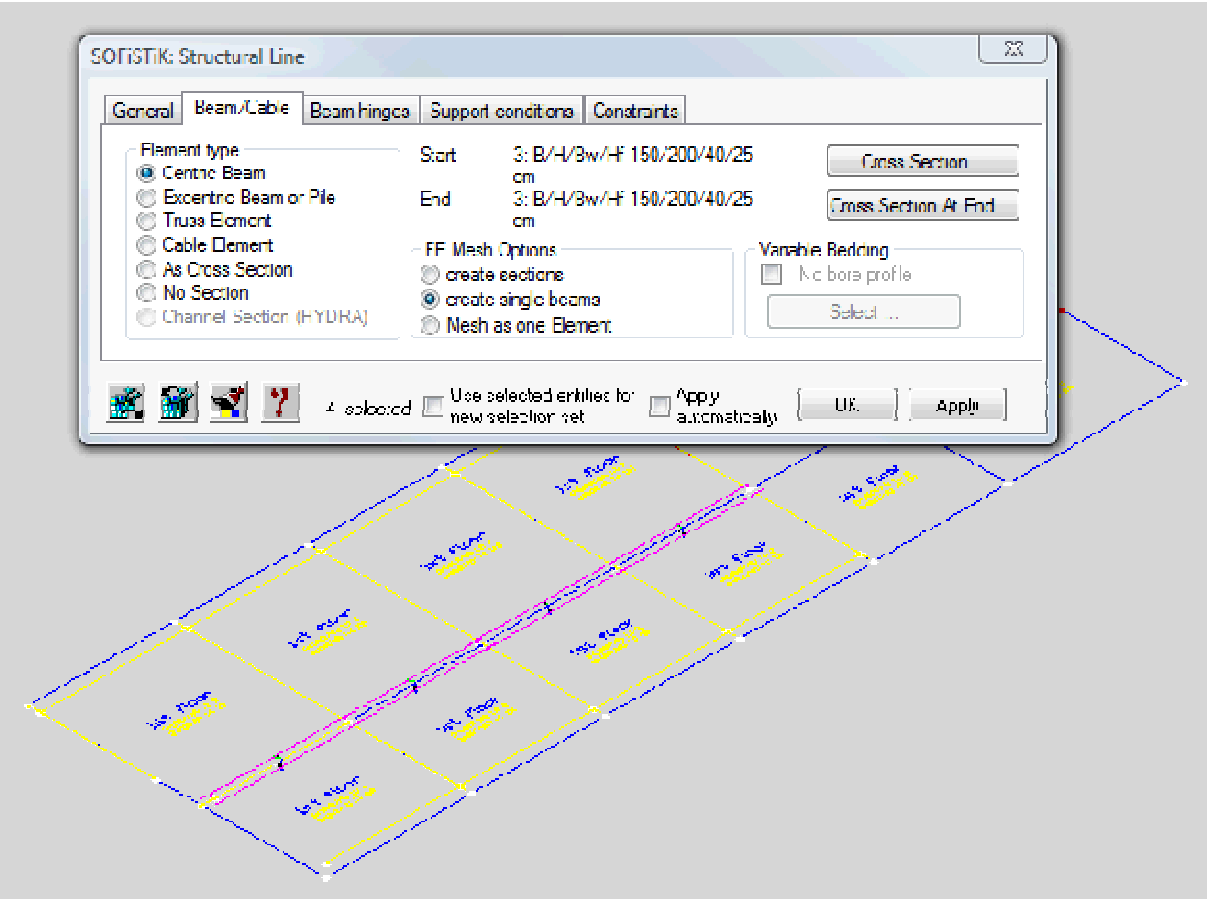


Figure 39: T-beam on axis 2

Command: Modify structure edge

Select the structure lines on axis 1 and 3 A-E (start on axis 1 from wall 1).

Because the T-beams are at the edge of the slab, you cannot use cross section number 3. You have to create a new cross section with a width of 1m only. This can be done directly, when selecting the cross section in the structural lines dialog. Just use the buttons “copy” and “modify” to create this new cross section.

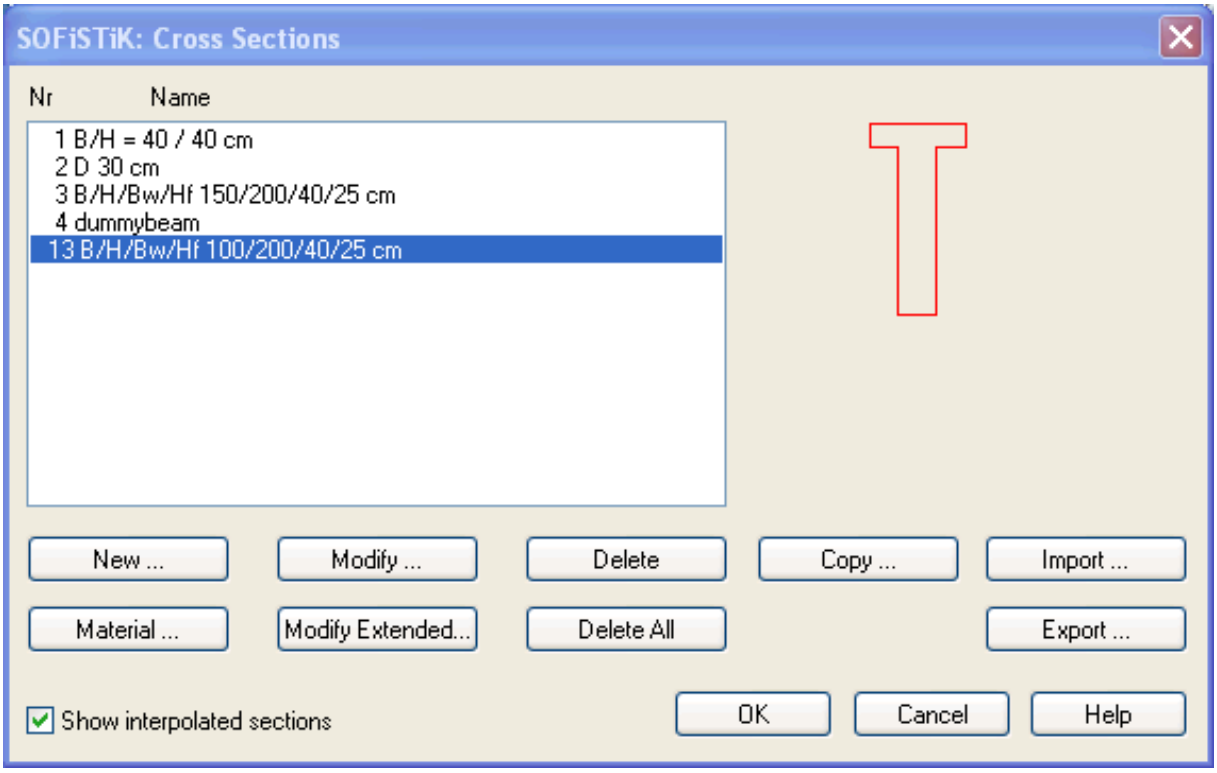


Figure 40: Create new cross section for edge T-beams

Command: Display all

After an export you can check the T-beams in ANIMATOR.

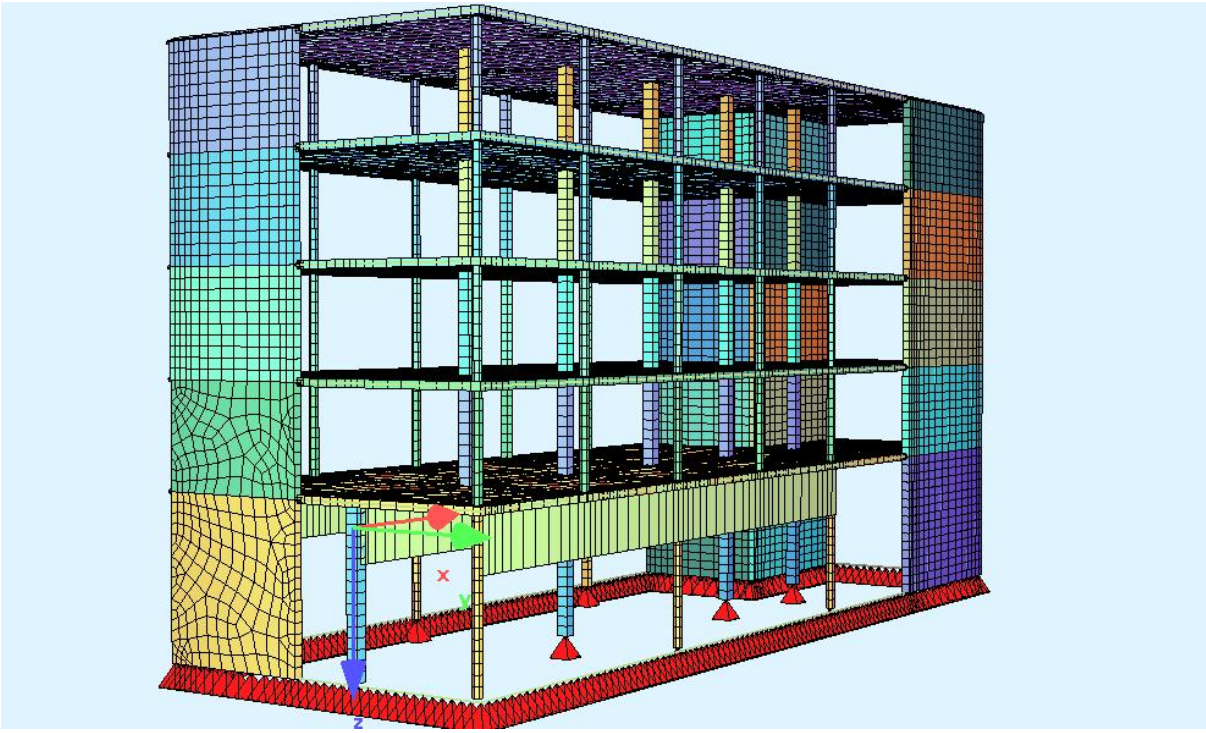


Figure 41: System with t-beams

6.3.2.7 Replenish roof over staircase

Above the staircase a part of the roof is still missing.

Command: Display selection set

Select the roof in viewport 1.

Command: Create structural area

Create the area above the staircase.

To make work easier, you can use the button with the paint brush to get the properties for the elements from the other element by clicking on another roof area.

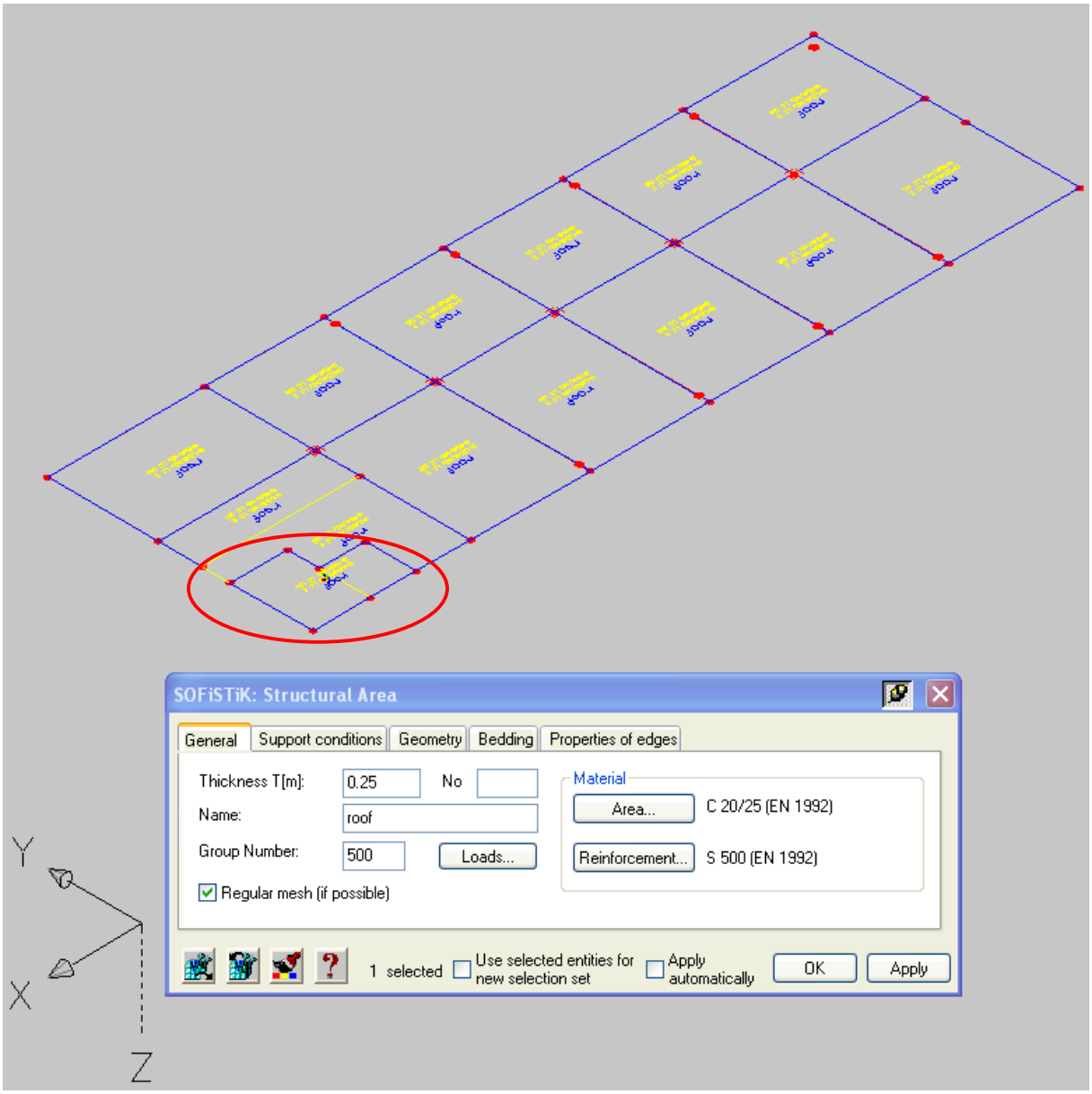


Figure 42: Structure area above staircase

Command: Modify structural area

Select the walls that are connected to the area you created in the last step.

Adjust the properties of the edges between the wall areas and the roof (hinge in direction of phix).

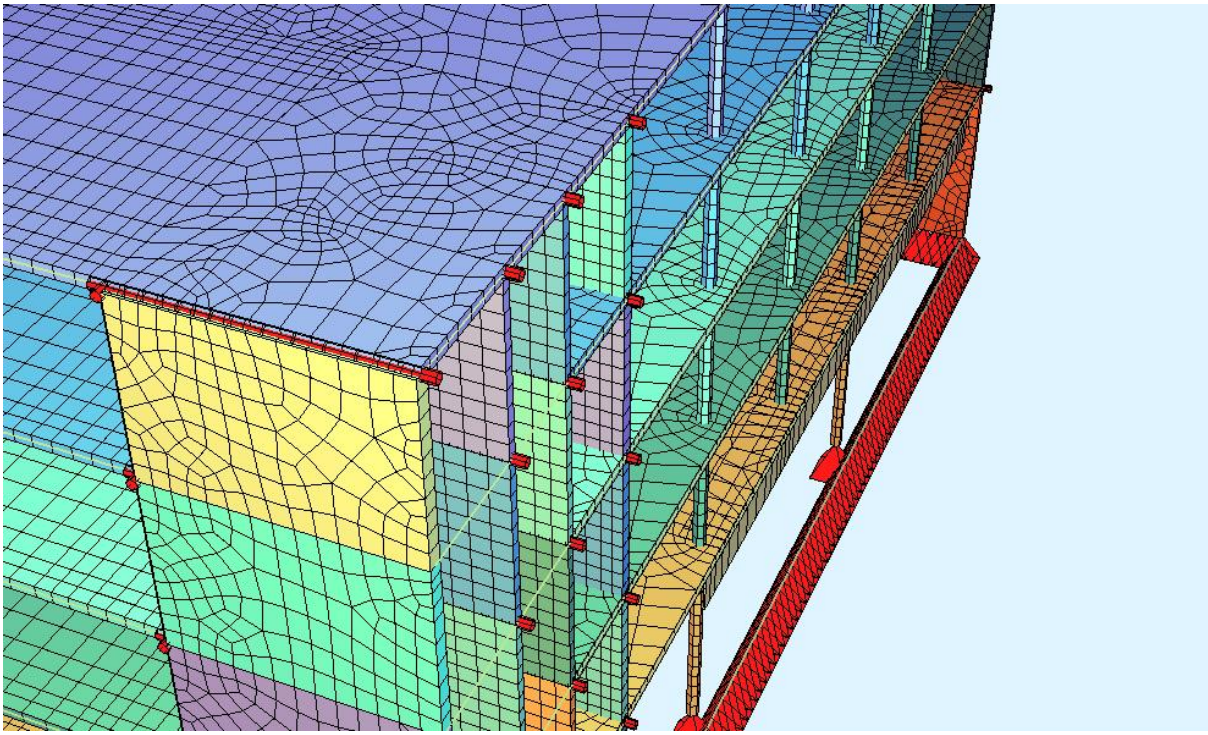


Figure 43: Cinematic constraints at the intersection of roof and walls

6.3.2.8 Replenish beams in staircase for wind load transfer

The last step in modelling the building is an adjustment for the structure lines with the dummy cross section in the staircase area. As described before (see also chapter 6.2 Define materials and cross sections), the dummy beams have nearly no stiffness and just transfer the loads to the slabs. But in the staircase there is no slab to which the dummy beams can transfer the loads. So the program would run into trouble while calculation. Therefore you have to model beams in staircase with a realistic cross section and stiffness.

Command: Display all

Command: Modify structural edge

Define and assign a new cross section to beams in staircase area.

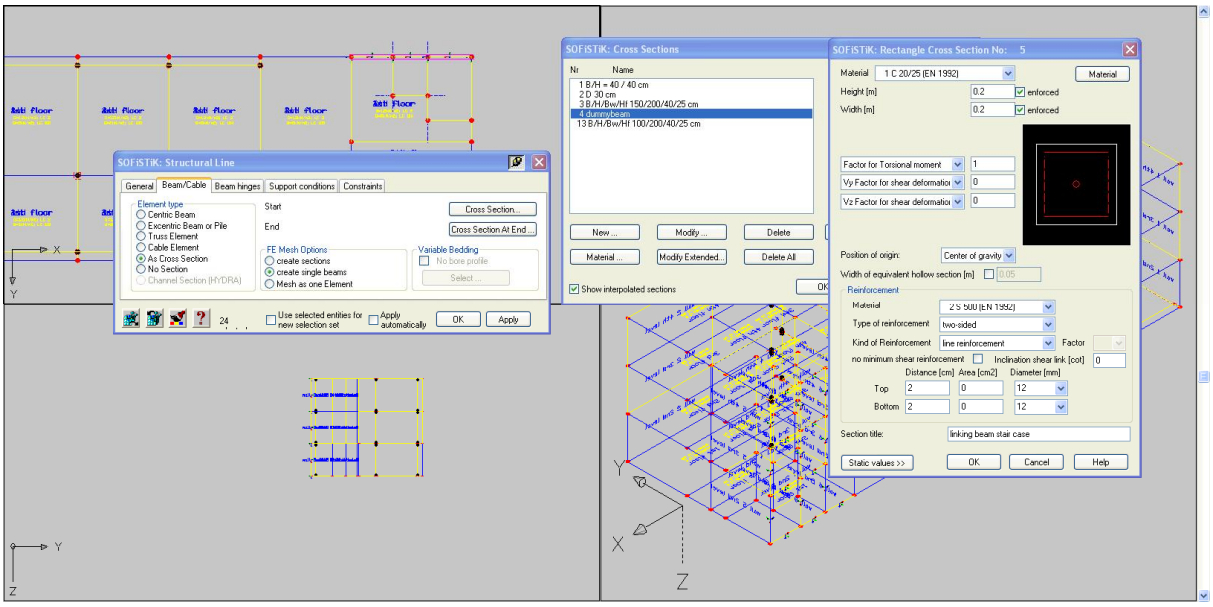


Figure 44: Beam in area of staircase

6.3.3 Additional loads (free loads)

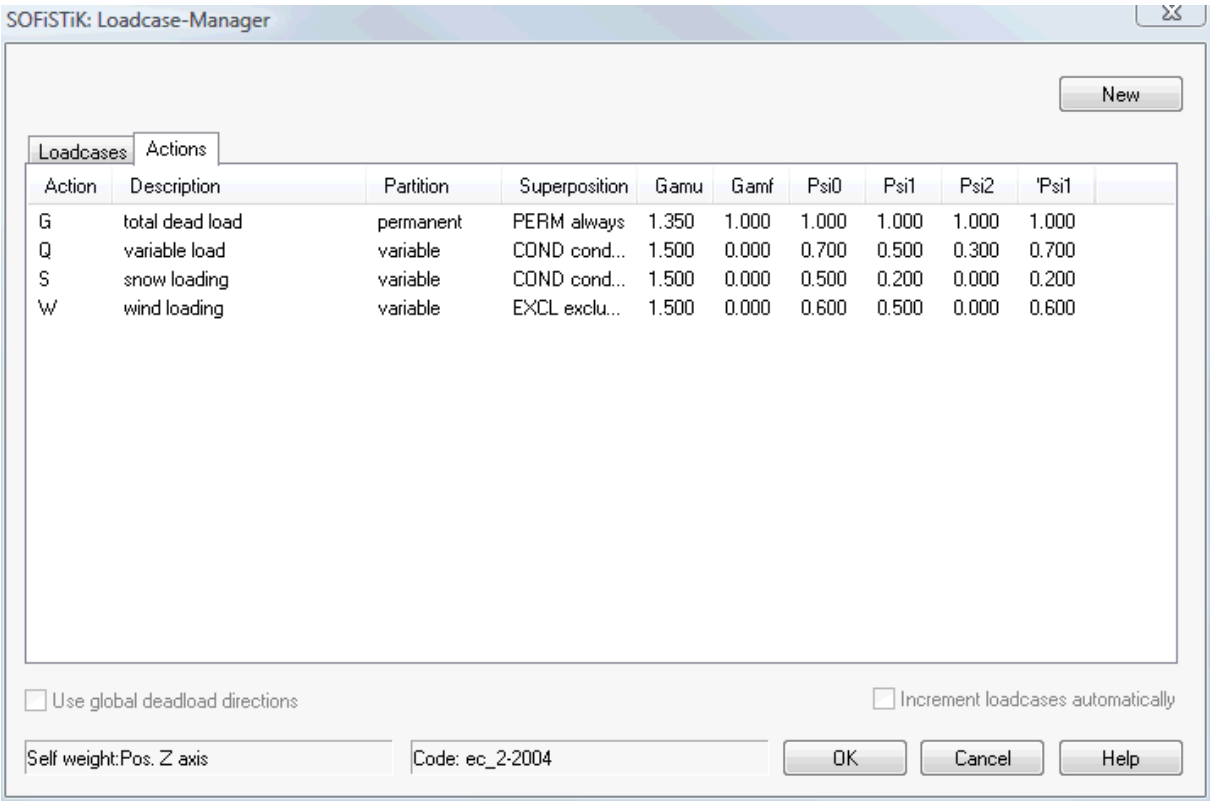
So far you have just defined and life loads on the slabs. Wind loads, snow loads and loads from cladding are still missing.

6.3.3.1 Define actions

If not already defined, it's recommendable to define actions for wind and snow loads now.

Command: load case manager

Click on register card "Actions" and define the (missing) actions like shown in Figure 45.



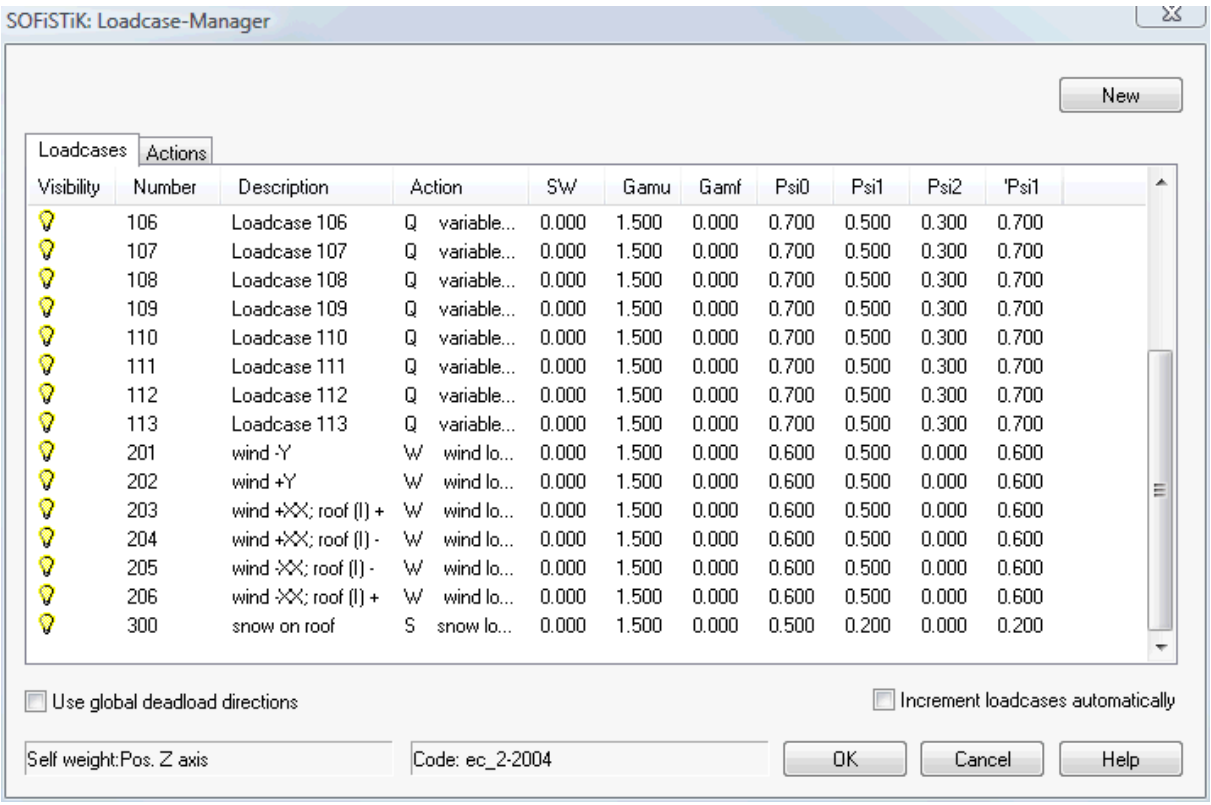
Action	Description	Partition	Superposition	Gamu	Gamf	Psi0	Psi1	Psi2	'Psi1
G	total dead load	permanent	PERM always	1.350	1.000	1.000	1.000	1.000	1.000
Q	variable load	variable	COND cond...	1.500	0.000	0.700	0.500	0.300	0.700
S	snow loading	variable	COND cond...	1.500	0.000	0.500	0.200	0.000	0.200
W	wind loading	variable	EXCL exclu...	1.500	0.000	0.600	0.500	0.000	0.600

Figure 45: load case manager - definition of action

6.3.3.2 Define load cases for wind and snow

Before you create wind and snow loads you have to define the according load cases. The individual loadcase number depends on your loadcase number scheme. (see also at chapter 4.1.2 Considerations about loads and actions)

Switch to register card Loadcases and define load case for wind and snow like shown in Figure 46).



Visibility	Number	Description	Action		SW	Gamu	Gamf	Psi0	Psi1	Psi2	'Psi1
	106	Loadcase 106	Q	variable...	0.000	1.500	0.000	0.700	0.500	0.300	0.700
	107	Loadcase 107	Q	variable...	0.000	1.500	0.000	0.700	0.500	0.300	0.700
	108	Loadcase 108	Q	variable...	0.000	1.500	0.000	0.700	0.500	0.300	0.700
	109	Loadcase 109	Q	variable...	0.000	1.500	0.000	0.700	0.500	0.300	0.700
	110	Loadcase 110	Q	variable...	0.000	1.500	0.000	0.700	0.500	0.300	0.700
	111	Loadcase 111	Q	variable...	0.000	1.500	0.000	0.700	0.500	0.300	0.700
	112	Loadcase 112	Q	variable...	0.000	1.500	0.000	0.700	0.500	0.300	0.700
	113	Loadcase 113	Q	variable...	0.000	1.500	0.000	0.700	0.500	0.300	0.700
	201	wind -Y	W	wind lo...	0.000	1.500	0.000	0.600	0.500	0.000	0.600
	202	wind +Y	W	wind lo...	0.000	1.500	0.000	0.600	0.500	0.000	0.600
	203	wind +XX; roof (l) +	W	wind lo...	0.000	1.500	0.000	0.600	0.500	0.000	0.600
	204	wind +XX; roof (l) -	W	wind lo...	0.000	1.500	0.000	0.600	0.500	0.000	0.600
	205	wind -XX; roof (l) -	W	wind lo...	0.000	1.500	0.000	0.600	0.500	0.000	0.600
	206	wind -XX; roof (l) +	W	wind lo...	0.000	1.500	0.000	0.600	0.500	0.000	0.600
	300	snow on roof	S	snow lo...	0.000	1.500	0.000	0.500	0.200	0.000	0.200

Figure 46 defining load cases for wind and snow loads

> Make sure that your new loadcases are defined with the right action. (you can change it via leftclick in the row Action and the according loadcase line)

6.3.3.3 Cladding loads

The loads from the cladding are acting on the slabs beneath. Thus the roof does not get any cladding load.

Because the foundation is not considered in this example, also the loads of the cladding on the ground floor do not have to be modelled.

Command: Loadcase Manager

Define a new loadcase number 3 with action G called “cladding”.

Command: Display groups

Switch off all groups except group 49.

Command: Display selection set

Select all floors except the ground floor and the roof.

Command: Free line load

Define a load of 0.5 kN/m as load in gravity direction. Select “BGRP - group beam elements” as reference type and apply the loads on group 49 only.

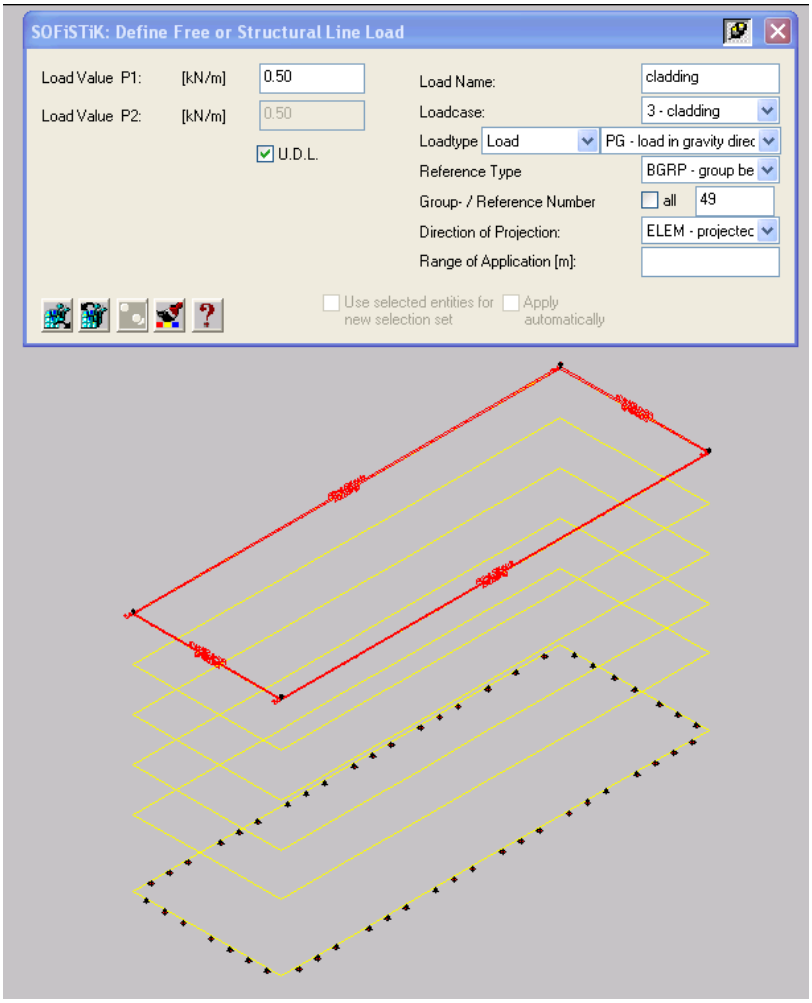


Figure 47: Load from cladding

6.3.3.4 Snow load

Command: Display all

Command: Free area load

Define a load of 0.75 kN/m² as load in gravity direction. Select “QGRP - group quad elements” as reference type and apply the loads on group 500 (roof) only.

> In the dialogs of free loads you can select a reference type and a group-/ reference number. Making these settings will make sure that the loads will be applied on the right structural elements.

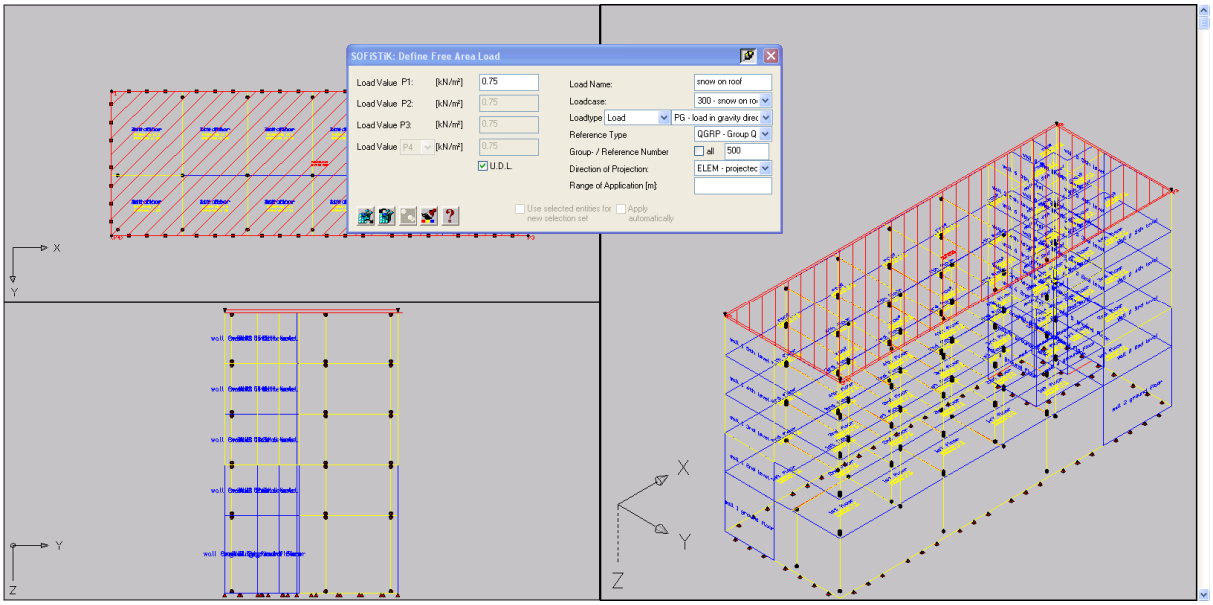


Figure 48: Definition of snow load

6.3.3.5 Wind loads

Defining the wind loads is a bit more labour intensive. As you have read in chapter 4.1.2, the wind loads are divided in different areas. To make work a little bit easier, define (new) help layers in AutoCAD/ Sofiplus(-X) - one for each wind load case. On these layers you can draw all lines/areas that you need to define wind loads. By that way you can easy snap the corner points for later wind load input.

> Drawing these help lines 0.5 m away from the real structure can help not to loose the overview on the system. If you do so you should set the depth on the LAR on 0.5m; in this tutorial it’s drawn direkt on the borderlines of the structure.

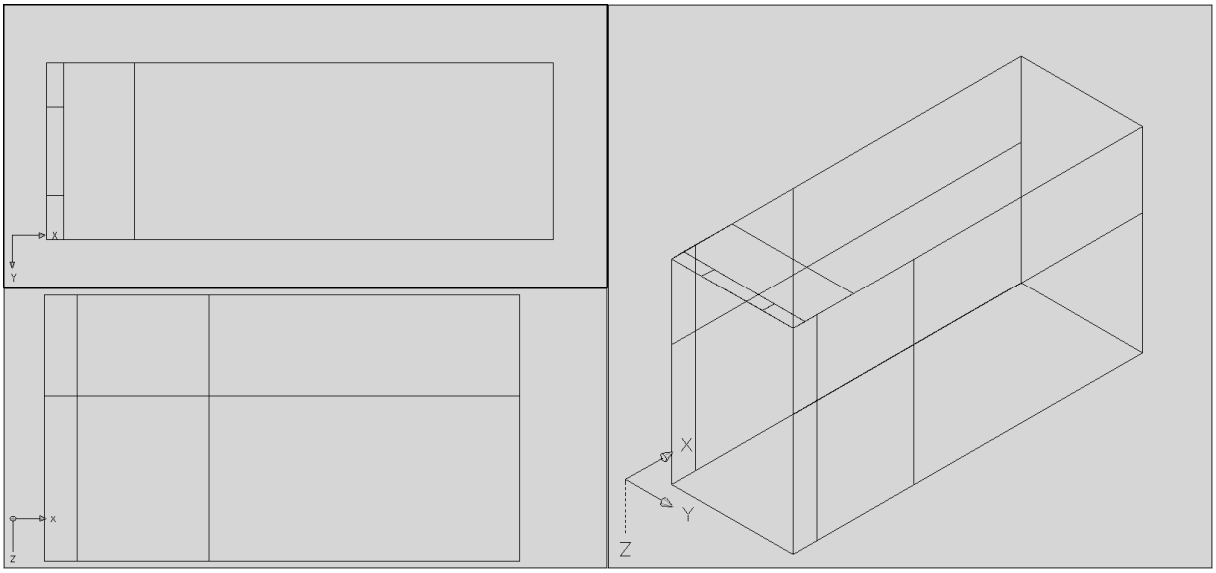


Figure 49: helplines for wind load input in global x-direction (structur is not shown)

Command: load distribution area

Create one load distribution area on each side of your structure on the border lines. Roof doesn't need an LAR. The "affected group" is group 49. Depth is 0.00 to avoid overlapping effects with bordering walls.

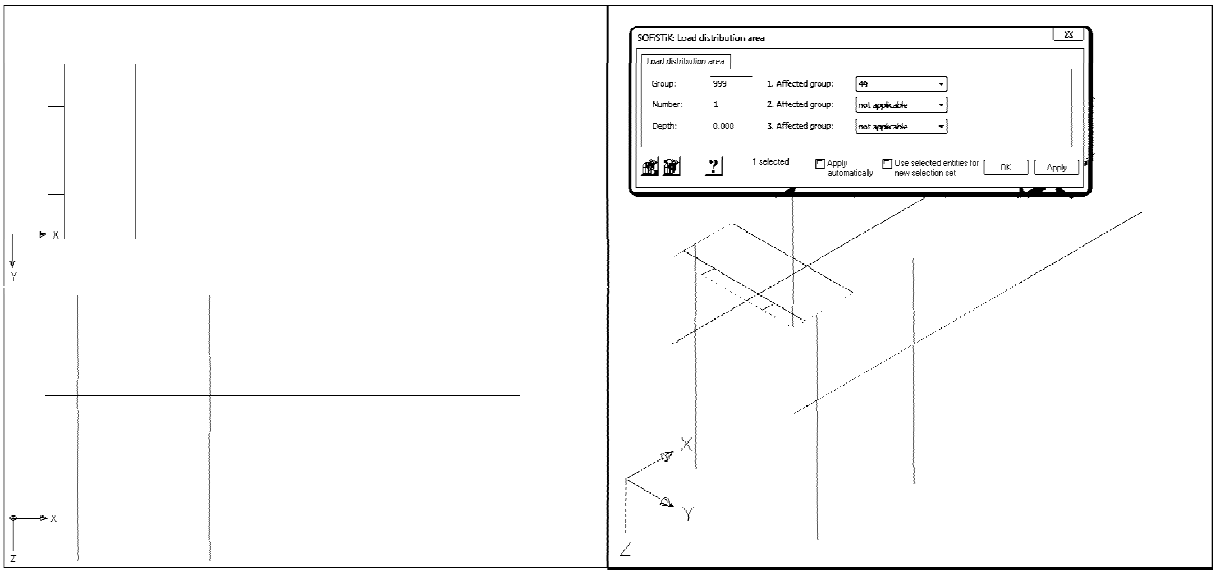


Figure 50: LAR's on all cladding sides

Command: free area load

Input the data for the load values and select reference type "LAR load distribution area" and the number of your LAR you created one step before. Create one area load for each wind load area (A-I) for each wind load case by snaping the corner point from the created help layers. You'll find all neccesary wind load data in chapter 4.1.2.

You can use Autocad commands/ options like _move, _copy, _mirror, _scale etc. to copy or modifying free loads. In this case it's provided to mirror all free loads from one wind load case to the according wind loadcase with the opposite direction.

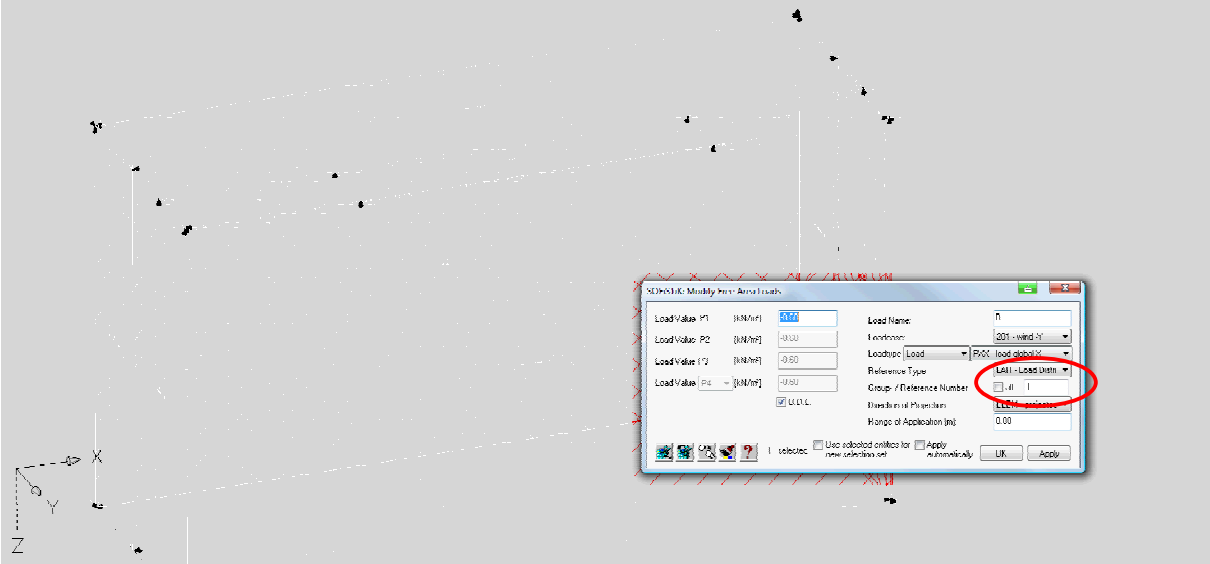
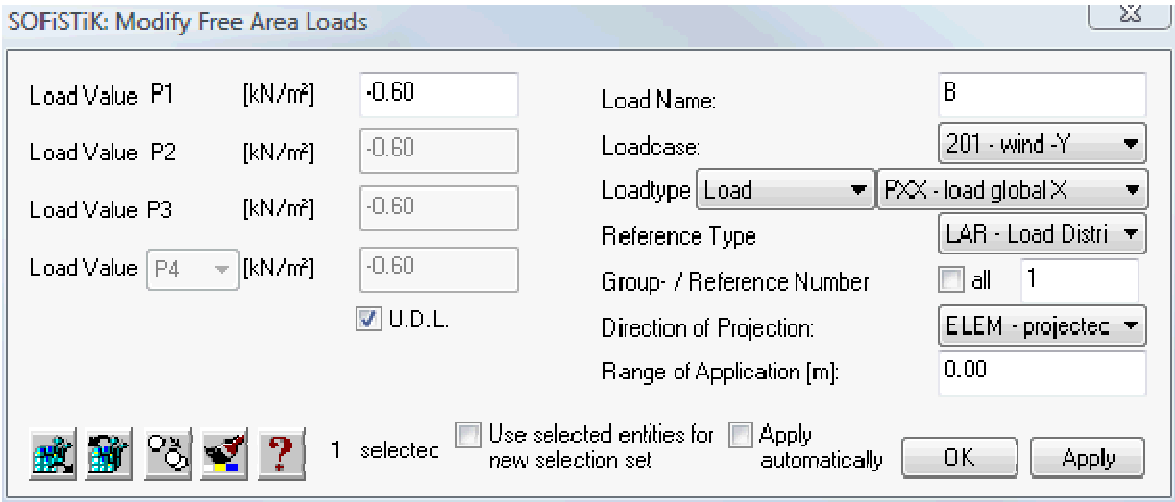


Figure 51: input/modify wind loads – here area B is referenced to LAR 1



6.4 Export/ Checks

Now your input is complete. Please export your system and change back to the SSD. Now you can proceed as usual with the linear analysis, the superpositioning, the design and the post processing.

In case the export runs into trouble, you should check for error messages in URSULA as well as in SOFiPLUS(-X) directly (press F2).
Check the system (mesh, supportings,…) and the loads (related to elements) with ANIMATOR/ WinGRAF before starting the analysis. If the mesh has deformed elements please try the recommendations given in chapter 4.3.2.

If you calculate the “Linear Analysis” in SSD, the program will return a warning message because of the “unstable” dummy beams in the system. It is just a hint, which does not require any changes. (dummy beams are unstable because they have a small stiffness)

7 Index of Figures