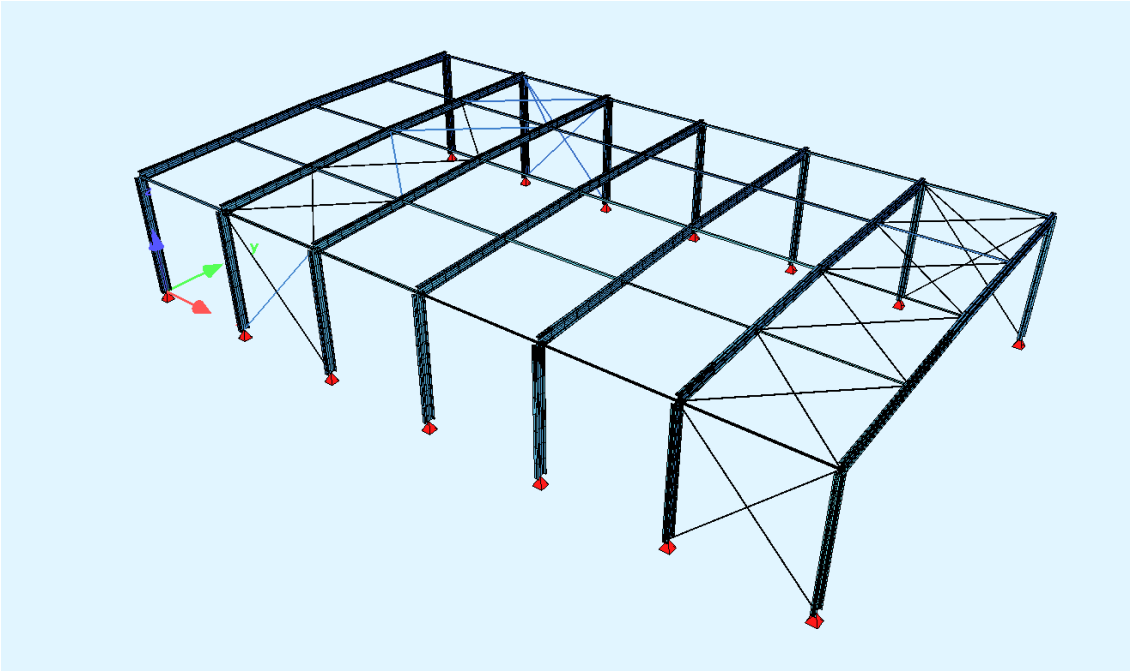
Tutorial

3D storage building

(Steel Design EC 3)

SSD

SOFiPLUS(-X)



Contents

1 Preface

1.1 What is the intention of this tutorial?

This tutorial is an introduction to 3-d modelling of a storage building. It will guide you through the whole process from modelling to design. Focused on the general approach of handling a 3-d model with our software, this example shows you the analysis and design according to EC 1 and 3.

Our graphical user interface, the SOFiSTiK Structural Desktop (SSD) will be used. It allows you to control pre-processing, processing and post-processing for the entire SOFiSTiK Software suite.

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 which is part of your SOFiSTiK manuals.

1.3 Tutorial Version

This tutorial is based on SOFiSTiK Software version 23 with service pack 12/2008 and SOFiPLUS-X 16.4 (build 23, AutoCAD format 2006)

2 Description of the project

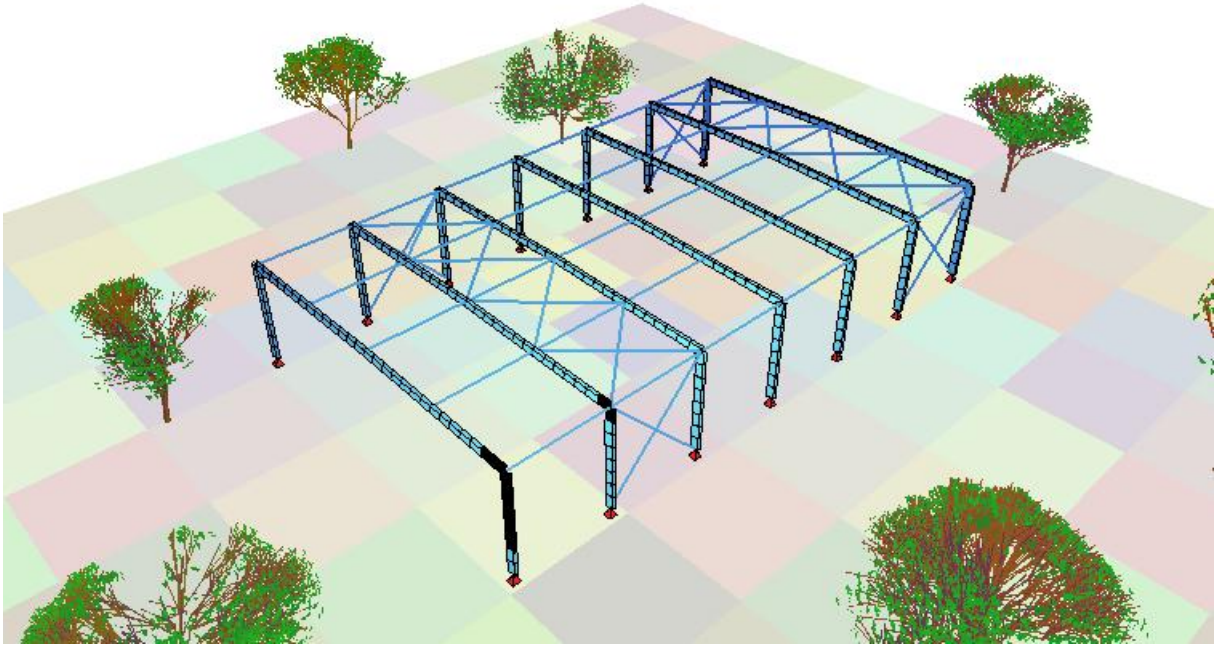


Figure 1: Overview building

In this tutorial we will explain how to analyse the storage building shown in figure 1. The building has a width of 18.4m, a length of 30.4m and it is 5.5m high. The exact measurements can be found in the corresponding *.dwg file.

The main structure consists of steel frames and a bracing system of tubes and cables.
The frames are made of steel girders IPE 400 and IPE 450 with haunches in the corners. Tubes with a cross section of 70x4.5 and cables with a diameter of 20mm are used for the bracing. All elements are made of steel S235.
Cladding and roofing will not be modelled, thus the loads will be applied on the structure directly.

The following loads will be considered:

Self weight of the structure	calculated by the software
Roofing	0,36 kN/m²
Snow (h<=300m NN, zone III)	0,75 kN/m²
Wind (simplified, not according to EC1: one loadcase with wind coming from y-direction)	roof: w=-0,3 kN/m² wall in x direction: 0,4 kN/m² wall in x direction, opposite to wind: 0,25 kN/m² wall in y-direction: 0,35 kN/m²

The analysis will be done according to Eurocode 3.

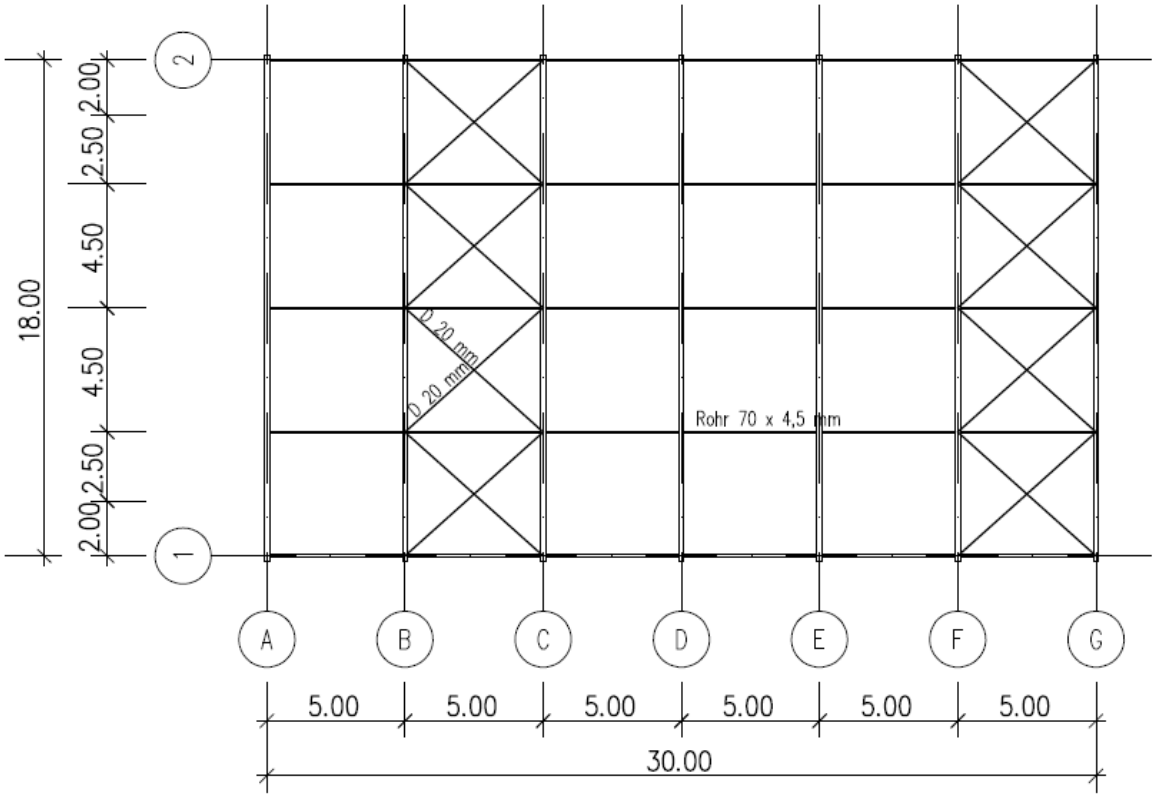


Figure 2: roof view

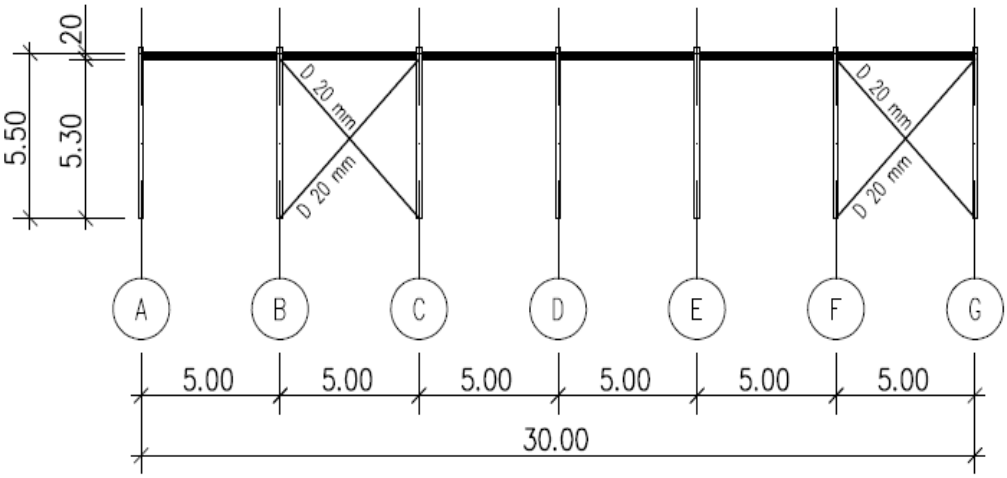


Figure 3: side view

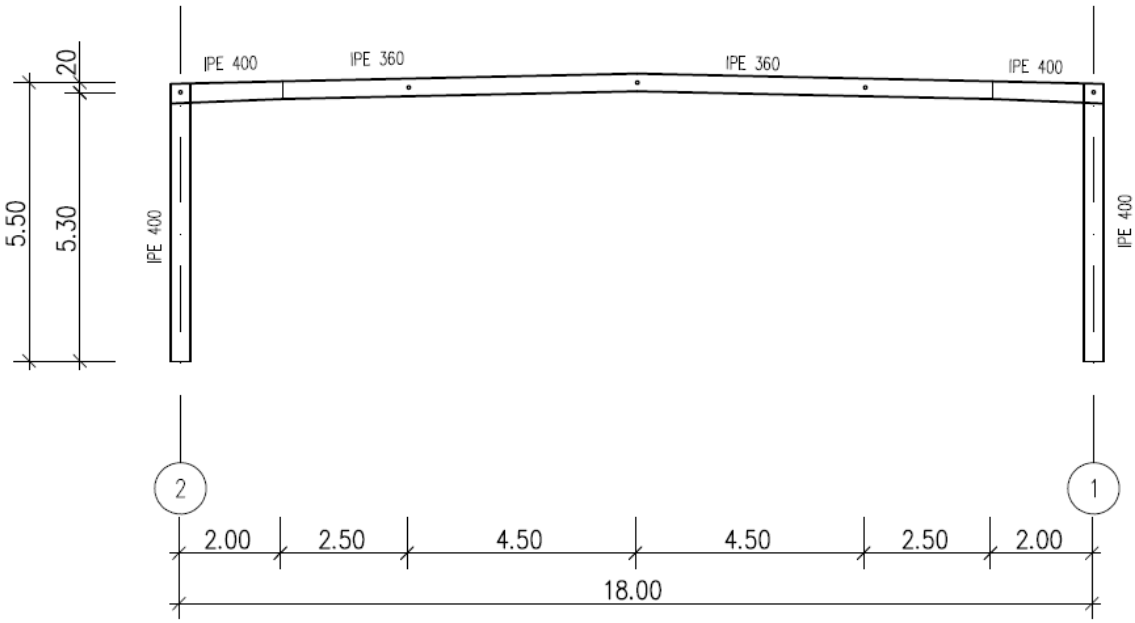


Figure 4: front view

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; separate analysis of each member	one complex model
Input/ handling	easy for each member; but often resulting in a lot of single, independent files	complex; but only one file for the whole structure
Level of abstraction	high	low
Modelling of details	useful for modelling details, less useful for coherence	modelling details not recommended, useful for showing coherence
Time for system generation	brief	long
Changes/ updates during working process	by hand for each member; danger of omitting something; a lot of work	just once for the whole model
Complexity of model	simple	high, danger of black box effect
Verifiability (by hand)	simple	difficult
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	limited	effective
Analysis of local stability	easy	difficult
Dynamic analysis (i.e. earthquake)	difficult/ impossible	easy
Time for analysis	brief for single components	long – 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 advantages and disadvantages.
2-D and 3-D modelling should be used complementary or apart - depending on the single job definition.

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 will be suited best for his project. It is similar to the decision to go to Munich by train, plane or car. By car, you can travel individually, going by train is very comfortable and going by plane is the fastest way to travel. Each choice has its advantages and disadvantages. Only if you know the whole circumstances, you can make the best decision. Making some manual checks in advanced to get a better understanding for your system is strongly recommended.

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 some planning before starting actual work with the software. As discussed in the last chapter it is not possible to make completely detailed design when using a 3d-model.

4.1.1 Considerations about the system

You first should list all the design checks you have to make. Based on this list you can decide which components of the structure have to be model and how much they can be simplify (rule: as simple as possible, but as precise as necessary).
Next you should check if some components could be merged into 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) will save you a lot of time during the design process and will allow you to check results easily. It can also help if you are not sure how to model details. You can assess the effect of the structural member on the main structure and decide whether it will be worthwhile modelling in a more detailed way or whether it will suffice to use 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 our building we decided to use the following load case concept:

Load case(s)	Content
1 –9	Dead loads
1	Automatically determined self weight
2	Additional dead load on roof
10 – 19	Snow
10	Snow on roof
20-29	Wind loads (simplified, not according to norm)
20	Wind in x-direction
21	Wind in y-direction

Keep your system flexible and upgradeable, i.e. don't use consecutive load case numbers only – if you skip some numbers 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 the last superposition that has been saved with this load case number. If it's annoying – just rename.

4.1.3 Considerations about groups

Defining a reasonable group concept for all structural components can be helpful for selecting a particular structural system, applying loads, analysing and designing elements as well as for graphical post-processing.

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.

There is no universal concept for the definition of groups, it rather depends on the problem that has to be solved. The following table shows how we classified the elements in our example:

Component	group number
Columns	1
Roof beams	2
Longitudinal trusses	3
Stiffening cables	4

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 while spending some time thinking about the details to avoid mistakes and to get the most efficient model. 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 we will discuss some details and decisions for our storage building.

The main frames will be modelled of centric beams. This means that the girders of the columns and roof beams overlap in the corners. This rigid joint has to be considered when designing the connection.

You have to use two structural lines for the roof beams to be able to model the haunches.

The longitudinal trusses will be modelled as truss elements (meshed as one element).

The stiffening system is made of cable elements that have to be meshed as one element. This way the automatic mesh generator will ignore the intersection of the cables.

4.3 Meshing

Normally the meshing will be done automatically by our program SOFiMSHB. Nevertheless, there are some means to influence the mesh generation:

- if possible, use elastic supports to avoid singularities.
- don’t model too detailed – otherwise you will get lots of elements and thus large calculation times, but not necessarily better results.
- define the origin in your model (i.e. don’t use Gauss-Krueger coordinate-system or similar systems) to avoid numerical problems – big coordinates require 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 points), then globally.

5 Workflow in SSD

5.1 Create new SSD project

Start a new SSD project by entering a name for the database, selecting a design code and the kind of system. The calculation is done with the module ASE.

The pre-processing will be done graphically with SOFiPLUS(-X)

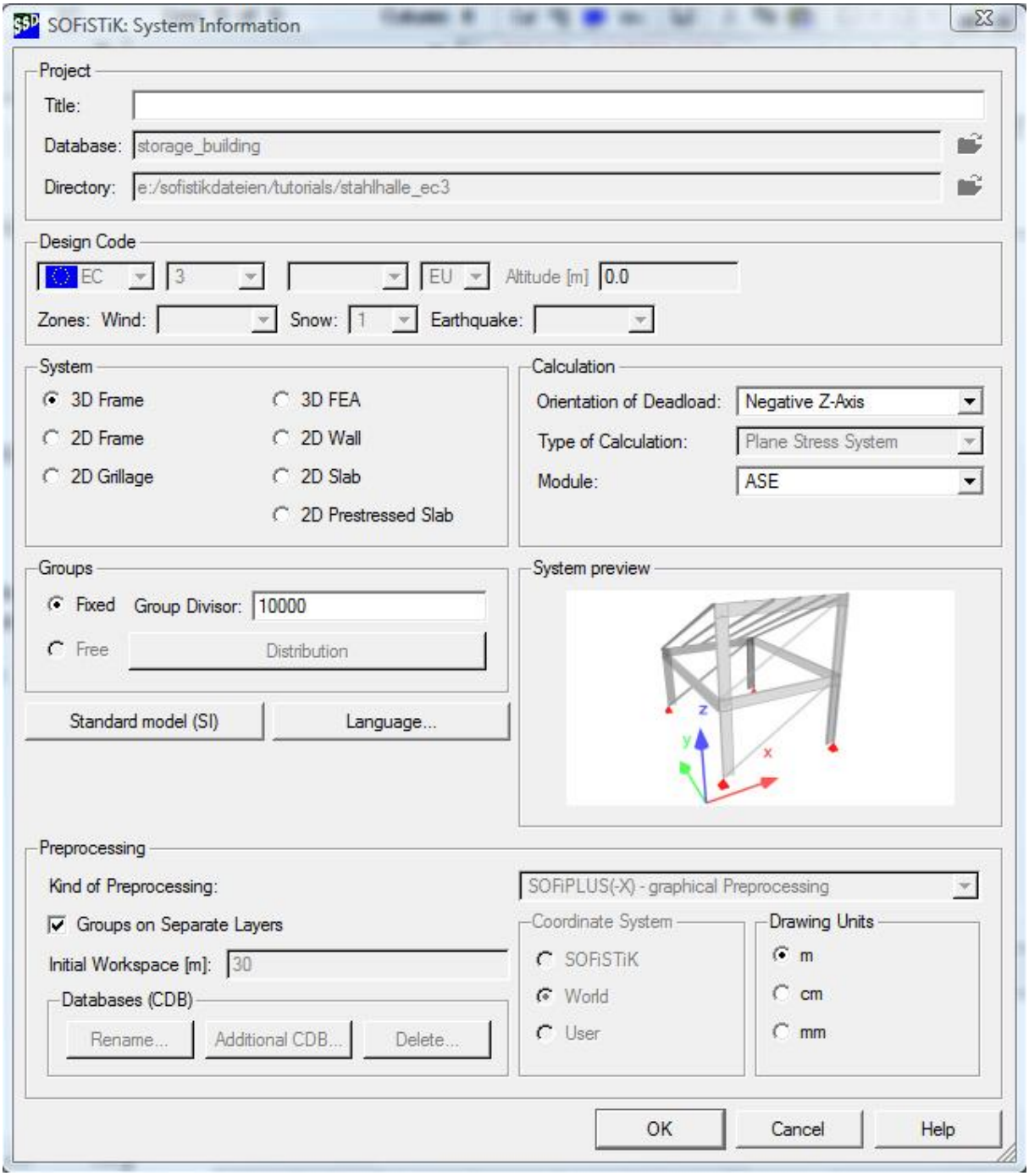


Figure 5: system information

If you want to use an architect's plan, 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.

5.2 Define materials and cross sections

Define the materials and cross sections you need for the project. The cross section IPE 450 has to be defined twice, once for the columns and once for the roof beams. This will help if you need to change one of the cross sections during the design process.

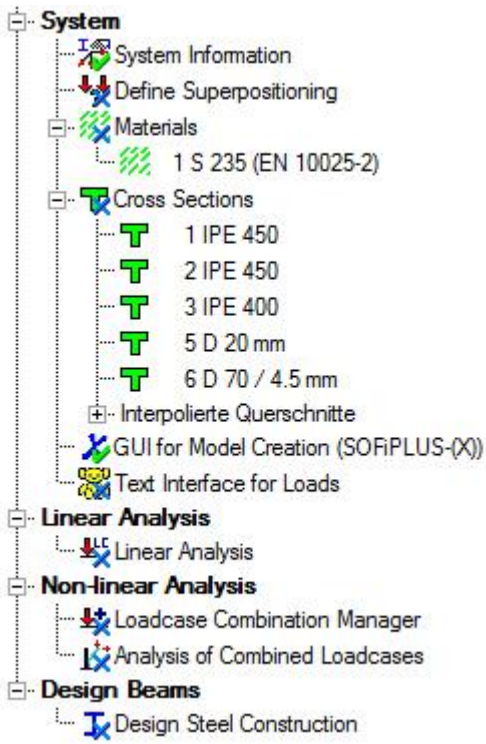


Figure 6: **Materials and cross sections**

Materials and cross sections can be added and modified later.

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

Click on „GUI for Model Creation“ to open SOFiPLUS(-X). If a *.dwg with the same name exists, it will open up automatically. The *.dwg for this project includes the system lines for one frame, some help lines and the axes of the building.

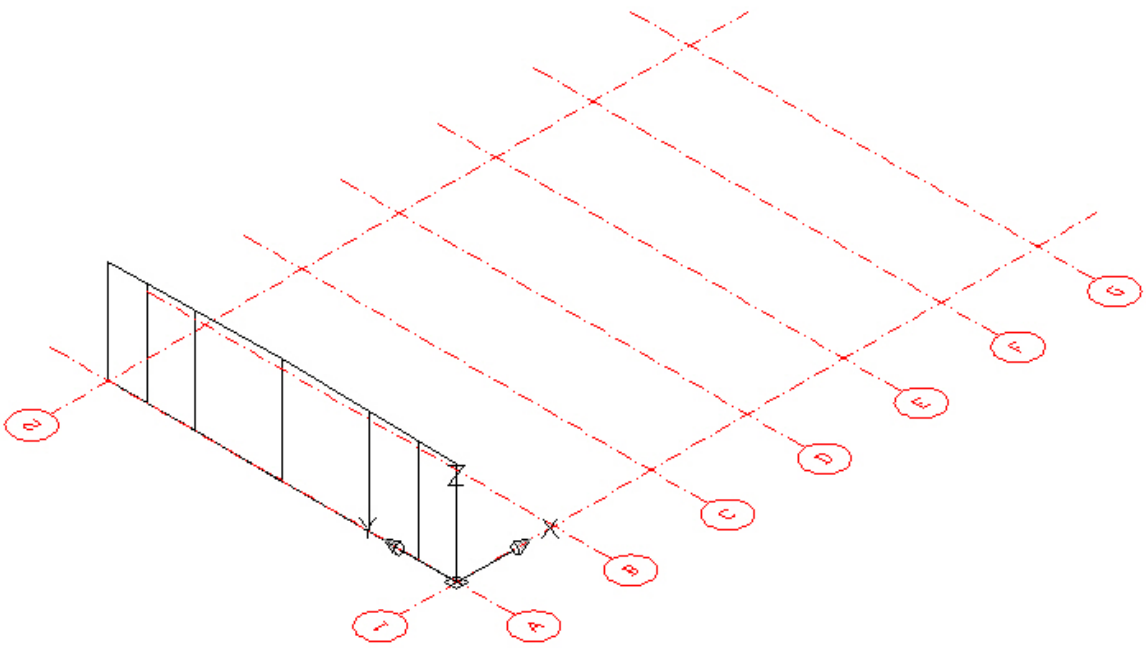


Figure 7: System and help lines of one frame and axes of the building

Before starting to work, please check the origin of the system. If it is not yet in one of the corners of the frame, please move the frame 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).

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

Now you can start to create the structural elements.

5.3.1 Create elements of one frame

Command: Structural Edge

First define all the structural lines of the frame as element type “no section”.

Then you can turn off the layer with the AutoCAD lines.

Figure 8: System with structure lines

Command: Modify Structure Edge

Now you can assign the cross sections and group numbers to the structure lines.

The columns are IPE 450 and in group one. The roof beams (IPE 400) belong to group number two. For the haunches, which have a span of two meters on each side of the roof beams, you have to enter a start and an end section. Pay attention to the local x-direction of the beam and choose start and end section correctly (IPE 400 and IPE 450).

Hint: if you have selected a beam element, the local coordinate system is displayed.

You can check if the cross sections are assigned to the beam elements the way you want by visualizing the cross section contour.

Command: Visualize -> Create cross section contour

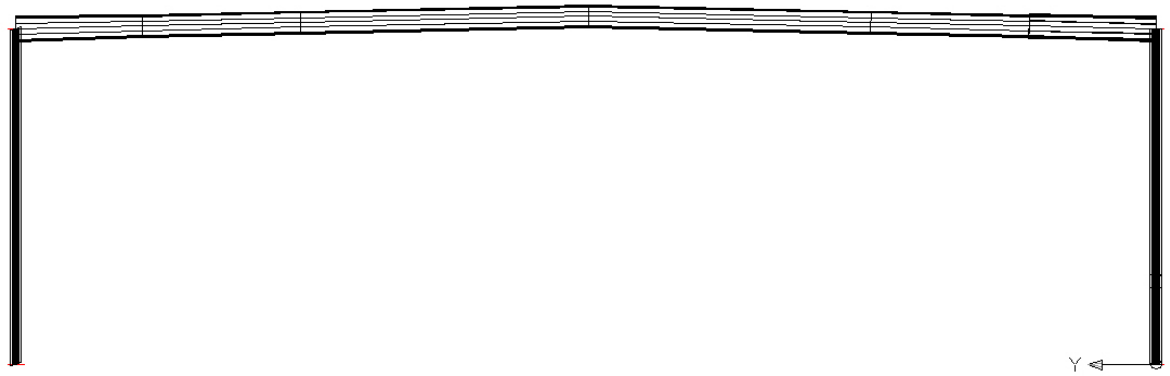


Figure 9: Cross section contours of beams

In this case the cross sections of the columns are perpendicular to the way they should be. Thus we have to turn the local z/y-axis of the beams.

To switch off the cross section contours, you have to switch off the layer X___SOLIDS in the layer manager.
Alternatively, you can export the system and check it in ANIMATOR.

Command: Turn edge

First switch on the layer with the help lines. In the command “Turn edge” select the option “show z-direction”. Select the structural line and click on two points of the z direction.

Command: Modify Structure Column/ Point

Select all points of support. On the tab “support conditions” check the boxes for global PXX, PYY and PZZ.

Because the input for the whole frame is finished, you should check your file for errors and export it to .cdb before copying the frame.

Command: _audit

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

Command: Export

When exporting the system to the SOFiSTiK-database *.cdb the program runs an automatic mesh generation. Usually you do not have to make adjustments, thus you can leave all pre-

settings. The program defines interpolated cross sections for the beam elements of the haunches. Afterwards you should check your system and the mesh with the ANIMATOR.

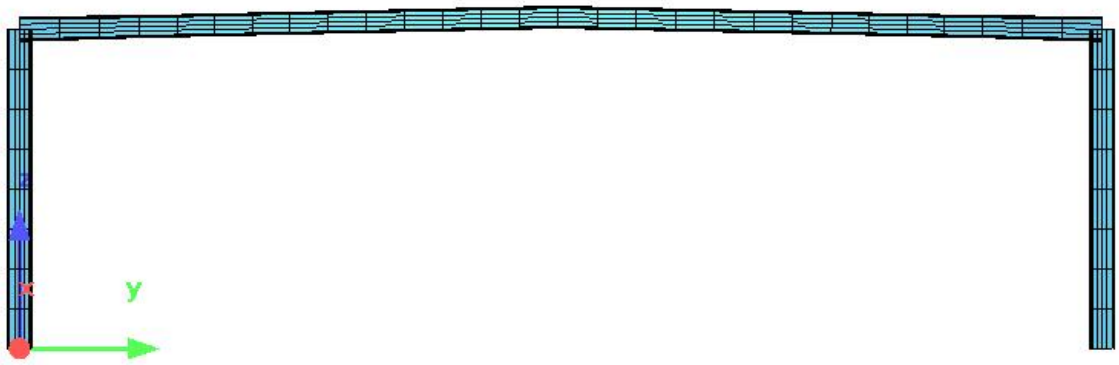


Figure 10: Generated mesh for the frame

5.3.2 From one frame to the whole building

If the export of the frame worked properly, you can now copy it. Choose a side view on the frame.

AutoCAD command: Copy

Select the whole frame. Choose one support point as base point. The distance between each of the 7 frames is 5 meters. You can enter all end points as relative coordinates (@5,0,0; @10,0,0; @15,0,0; @20,0,0; @25,0,0; @30,0,0).

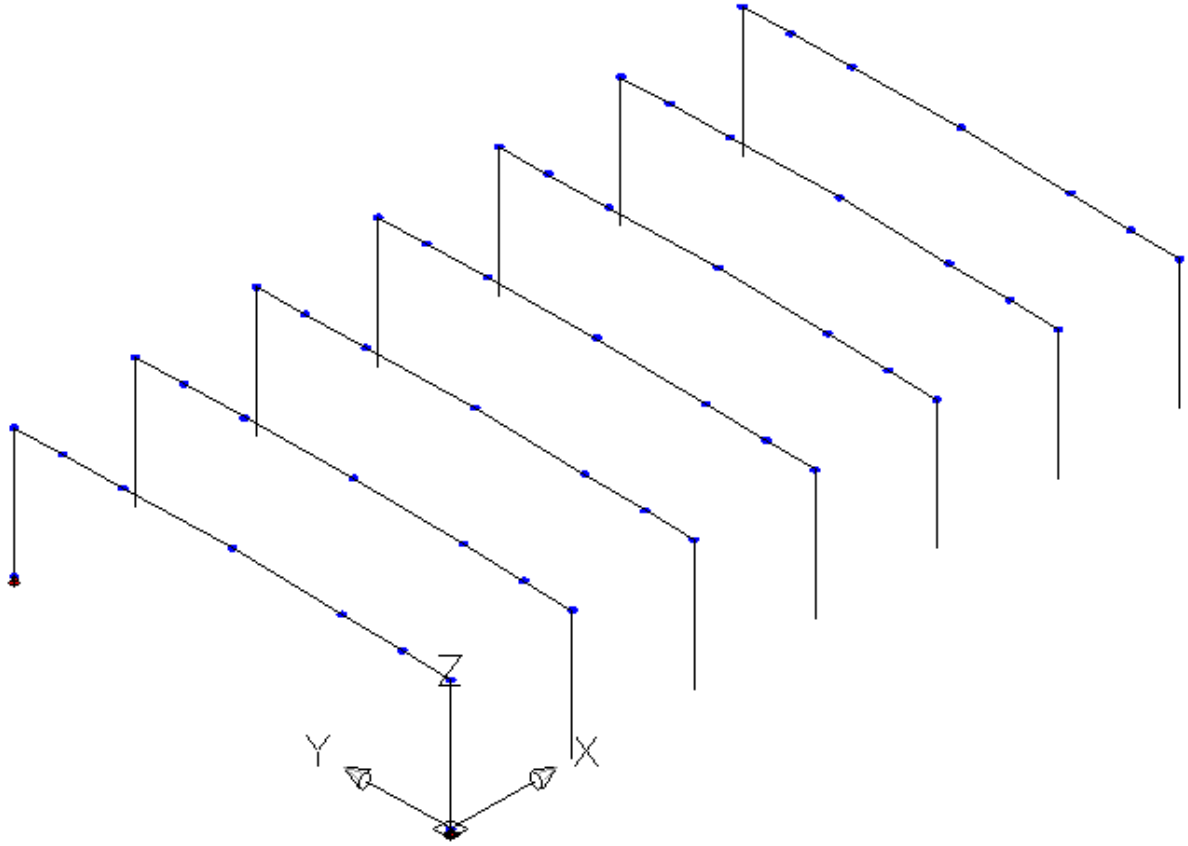


Figure 11: System with all frames

Command: Structural Edge

The longitudinal trusses are in group number 3. They are meshed as one element only.

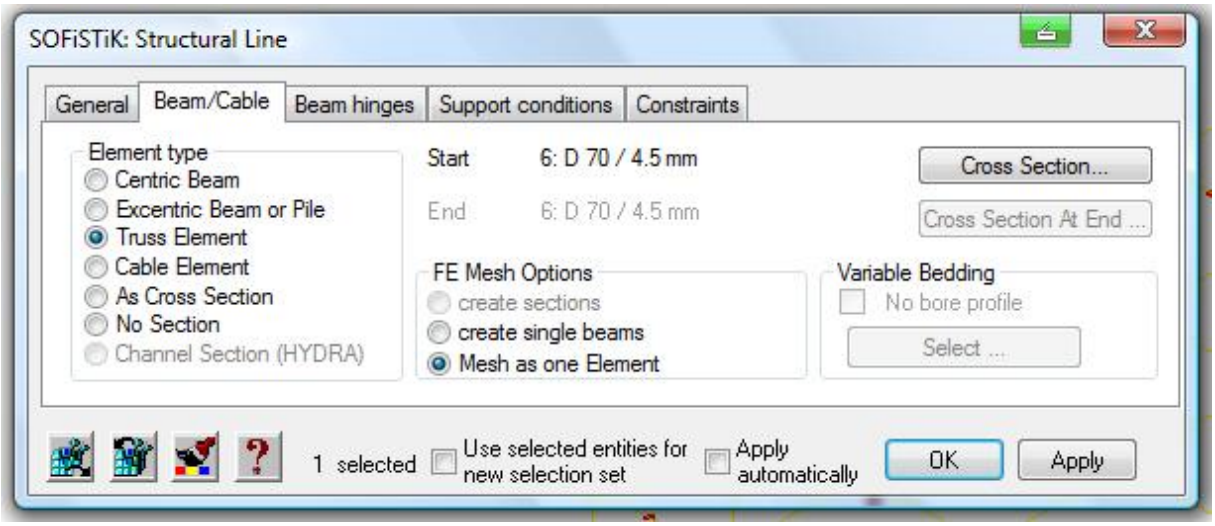


Figure 12: Truss elements

It is sufficient to show the start point on the first frame and the end point on the last frame. The program will automatically realize all intersecting points on the other frames.

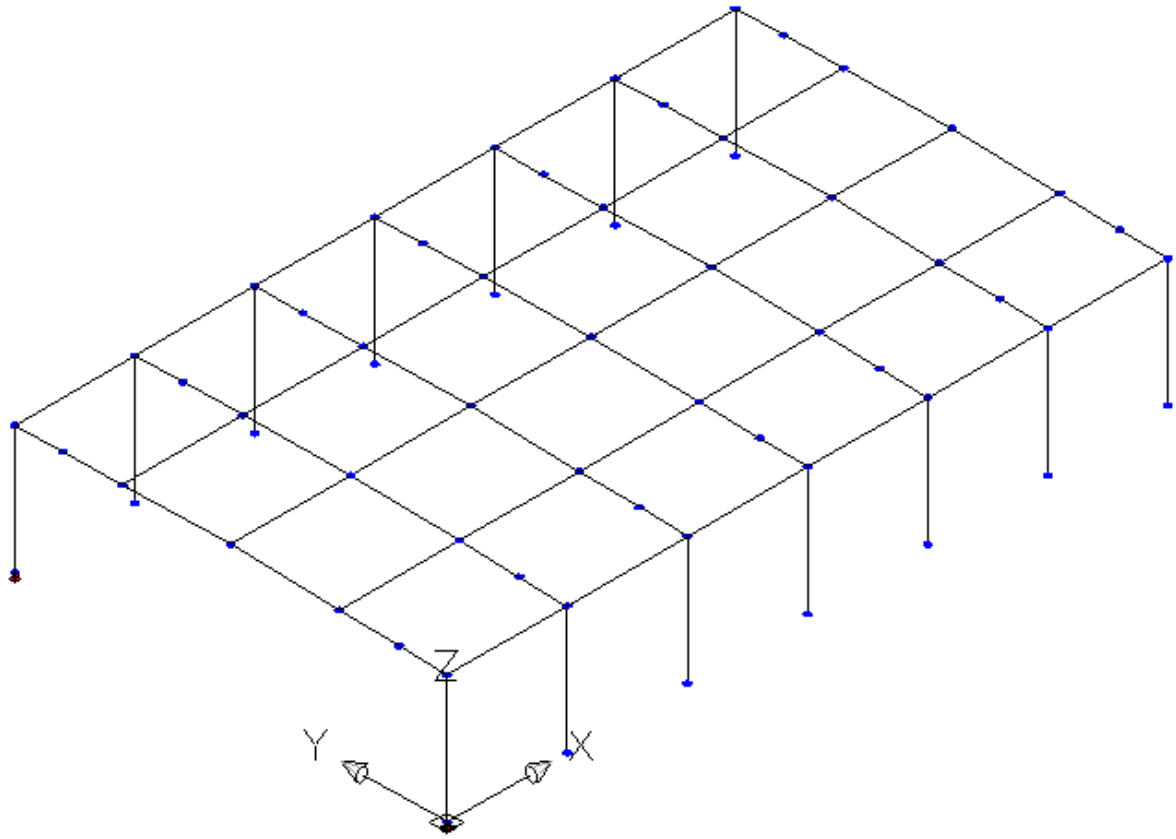


Figure 13: System with truss elements

Command: Structural Edge

The cable elements are in group number 4. They are meshed as one element only. Thus the mesh generator will ignore the intersection points.

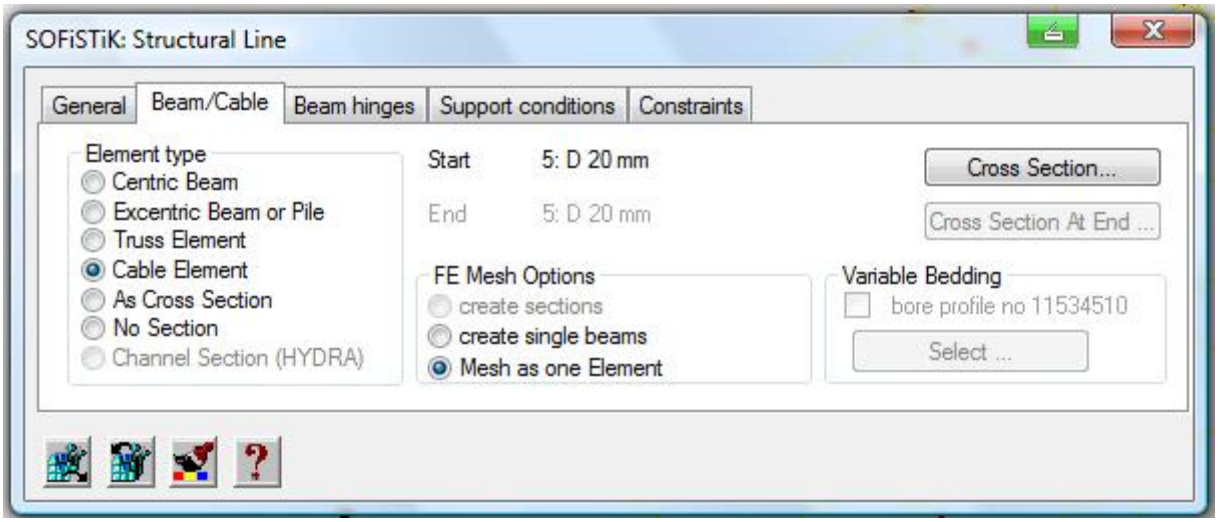


Figure 14: Cable elements

The stiffening cables are between frame 2 and 3 and between frame 6 and 7.

It might be the easiest way to choose a view from above first and then define the cables in the roof area. At last select some side views to define the cables between the columns.

> As long as the command “structure edge” is active you still have all settings in the dialog and can go on defining elements.
> If you left the command with a double click and want to use the same settings again, you can use the “show properties” button  in the structure edge dialog to copy the settings from another element.

If you are done with the input of the system you can again run an _audit and export the system.

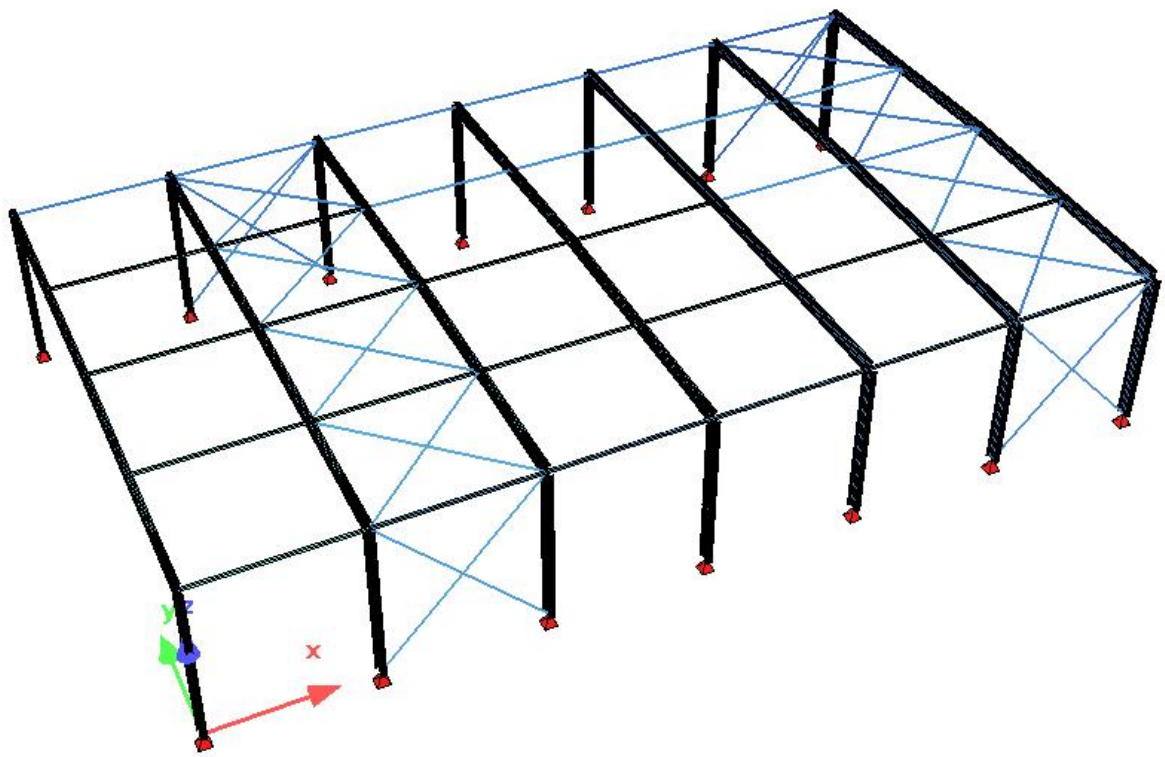


Figure 15: System with cable elements

5.3.3 Define loadcases and loads

Command: Loadcase Manager

Define the loadcases according to the table in chapter 4.1.2. To be able to choose the numbers for the loadcases, make sure that the box “increment loadcases automatically” in the lower right corner is not checked. Before defining the loadcases wind and snow, you first have to define the corresponding actions on the “actions” tab

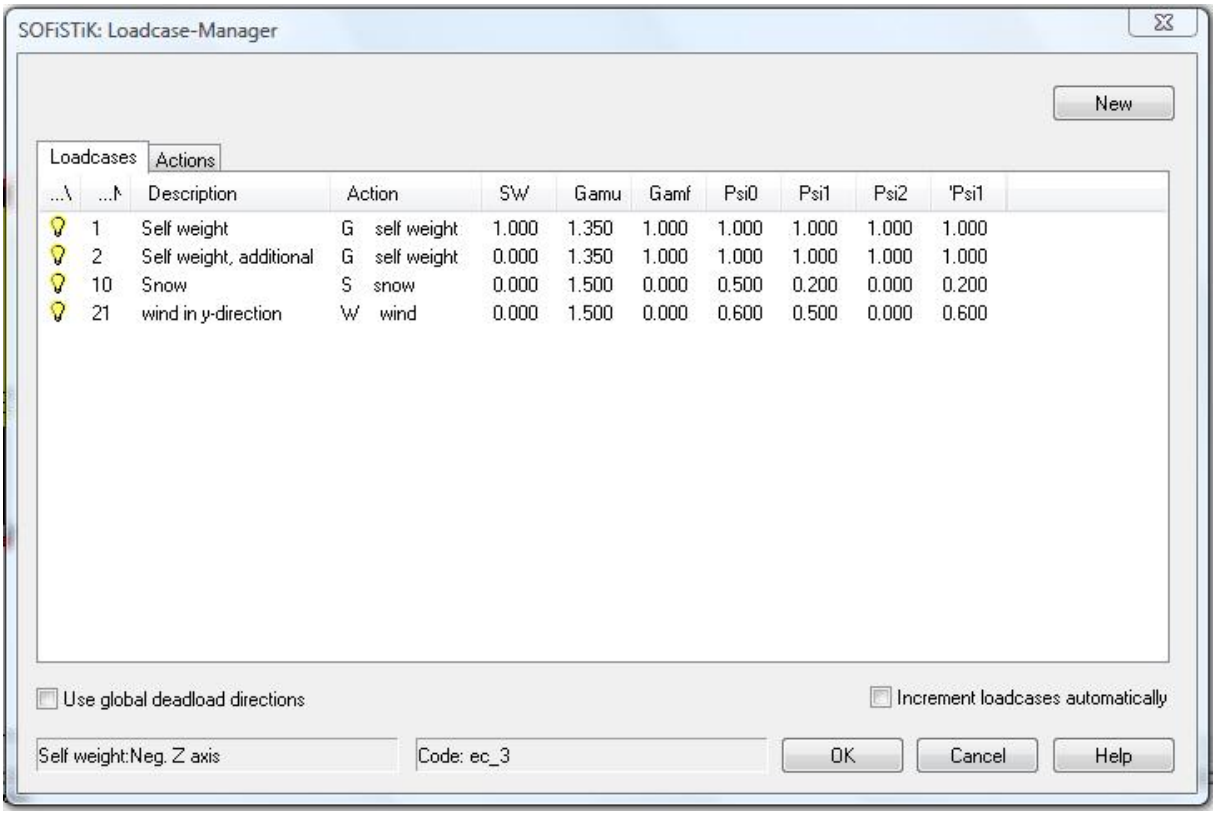


Figure 16: Loadcase manager

Command: Free line load

Define a load of 1.8 kN/m (=0.36 kN/m² x 5m) as load in gravity direction for the additional self weight of the roof. Remember that the weight of the steel girders is calculated by the software itself. The load has to be assigned to loadcase 2. Then apply it to all structural lines of the roof.

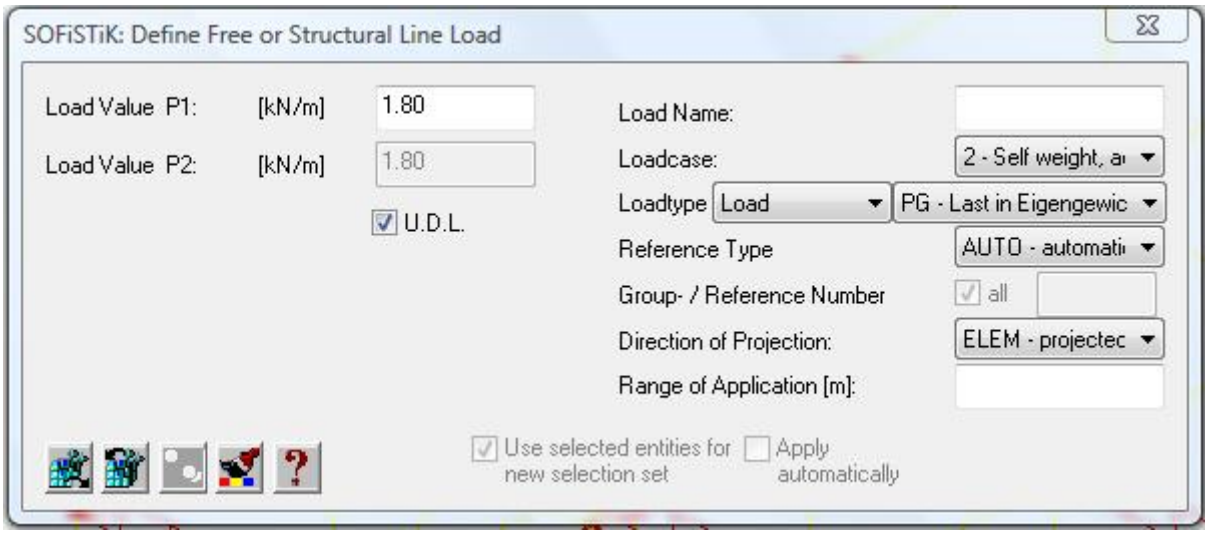


Figure 17: Free line load – additional self weight of roof

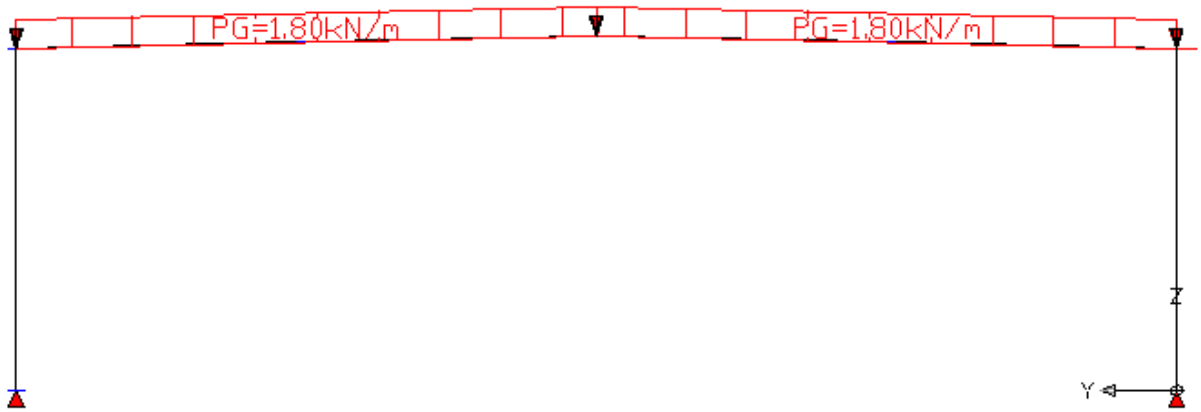


Figure 18: Free line load – additional self weight

Command: Modify line load

Because the influence area of the first and the last frame of the building is only half the size of the others, correct the load value of these line loads to 0.9 kN/m.

Command: Free line load

Define a load of 3.75 kN/m (=0.75 kN/m² x 5m) as load in gravity direction for the snow load on the roof. It has to be assigned to loadcase 10. Then apply it to all structural lines of the roof.

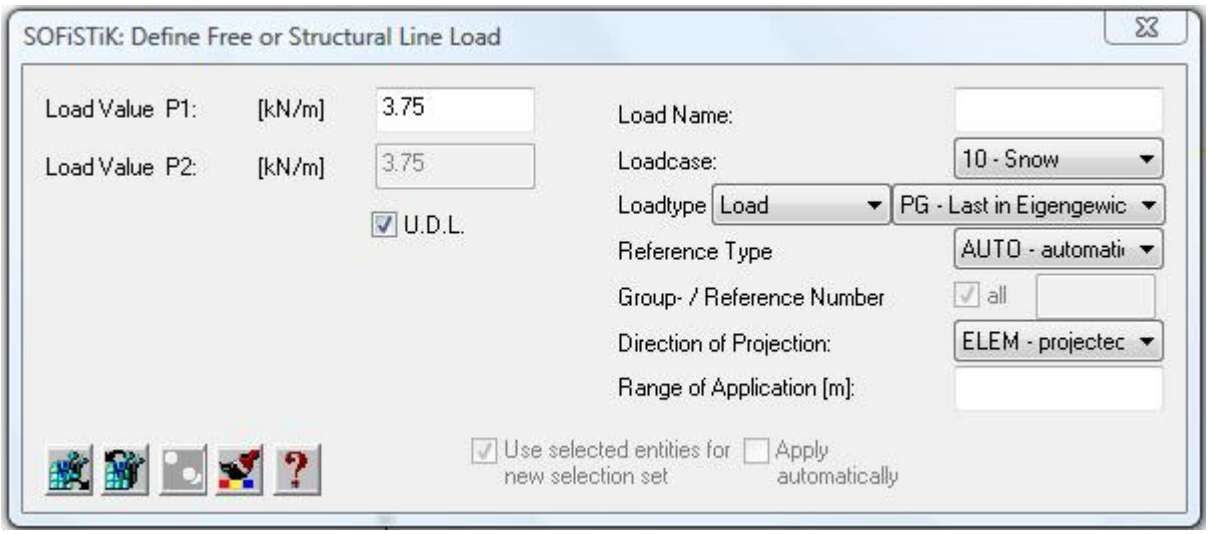


Figure 19: Define free line load – snow

Command: Modify line load

Again, correct the load values of the first and the last frame, in this case to 1.875 kN/m.

In this example, to simplify matters, we will define only one single loadcase for wind which is not according to EC3 (more information about the definition of wind loads can be found in our tutorial “3D-multistorey office building”).

Command: Free line load

On the roof beams apply a wind load in global z-direction of 1.5 kN/m (=0.3 kN/m² x 5m). On the columns on the right side define a load of 2 kN/m (=0.4 kN/m² x 5m) in global y-direction and define a load of 1.25 kN/m (=0.25 kN/m² x 5m) in global y-direction on the opposite column. The loads have to be asigned to loadcase 21.

Command: Modify line load

Again, correct the load values of the first and the last frame, in this case to 0.75 kN/m, 1 kN/m and 0.625 kN/m.

Command: Free line load

On the roof beams of the first and the last frame apply a load in x-direction of 0.96 kN/m (=0.35 kN/m² x 5.5m x 0.5), assuming that half of the windload on the gable wall goes directly into the foundation. It has to be assigned to loadcase 21 as well.

5.3.4 Definition of prestressing of cables

Now the input of the system and the loads in SOFiPLUS(-X) is done.
You can export your system, close SOFiPLUS(-X) and return to SSD.

Task: Text Interface for Loads

Next insert a new teddy task “Text interface for loads” *(select ‘insert task’ in the context menu appearing after a right click in the task tree in SSD)*. Because the new task 'Text Interface for Loads’ should be handled before ‘Linear Analysis’, you have to drag this task with the mouse between ‘GUI for Model Creation’ and ‘Linear Analysis’ *(see ‘Figure 20’)*. Here you have to define the pre-stressing of the cables. It belongs to loadcase 1 “self weight”.

```
+PROG SOFILOAD
HEAD TEXT INTERFACE FOR LOADS
LC 1 TYPE G FACD 1.0 TITL “SELF WEIGHT + PRESTRESSING OF CABLE”
CABL GRP 4 TYPE VX 2
END
```

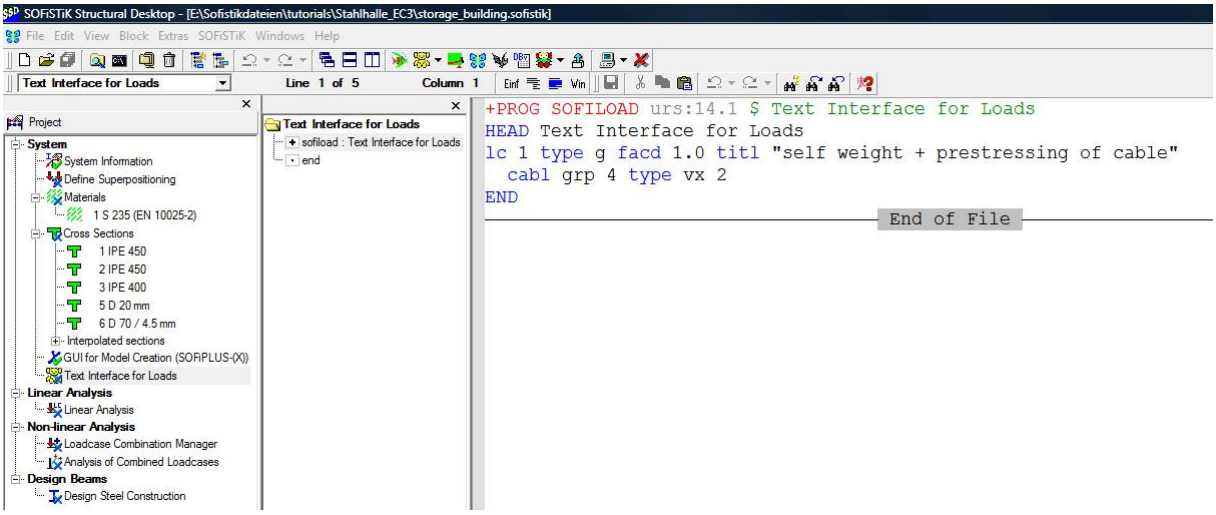


Figure 20: Text Interface for Loads

5.4 Analysis

Task: Linear Analysis

The task “Linear Analysis” calculates all loadcases according to theory 1st order. Attention: in a calculation according to theory 1st order without iteration cable elements are considered the same way as truss elements!

Task: Loadcase Combination Manager

Click on the “New” button to create a new loadcase combination. Select a loadcase on the right side and add it to the combination with the buttons in the middle of the dialog box.
We will consider the following loadcase combinations:

LC 1001: 1,35*LF1+1,35*LF2+1,5*LF10+1,5*LF21
LC 1002: 1,0*LF1 +1,0*LF2
LC 1003: 1,35*LF1+1,35*LF2

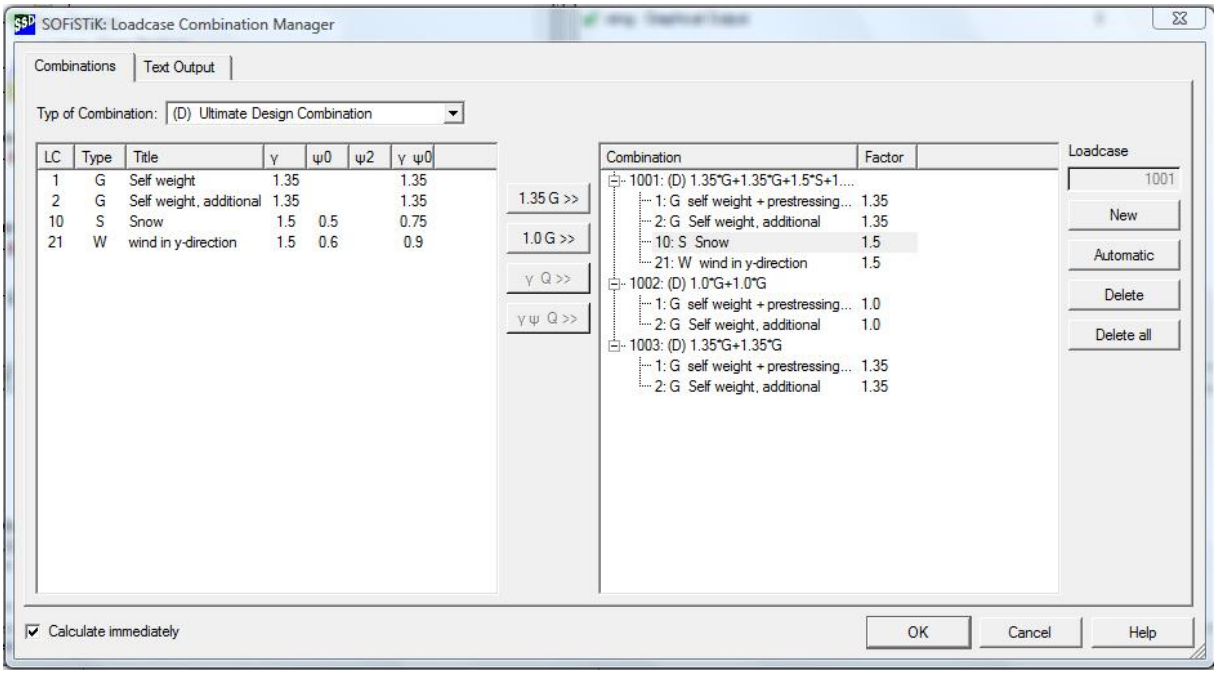


Figure 2118: Loadcase Combination Manager

Task: Analysis of Combined Loadcases

The analysis will be done according to theory 2nd order with a reduced stiffness of 1,1.

The inclination in global X and Y-direction will be set to 1/200.

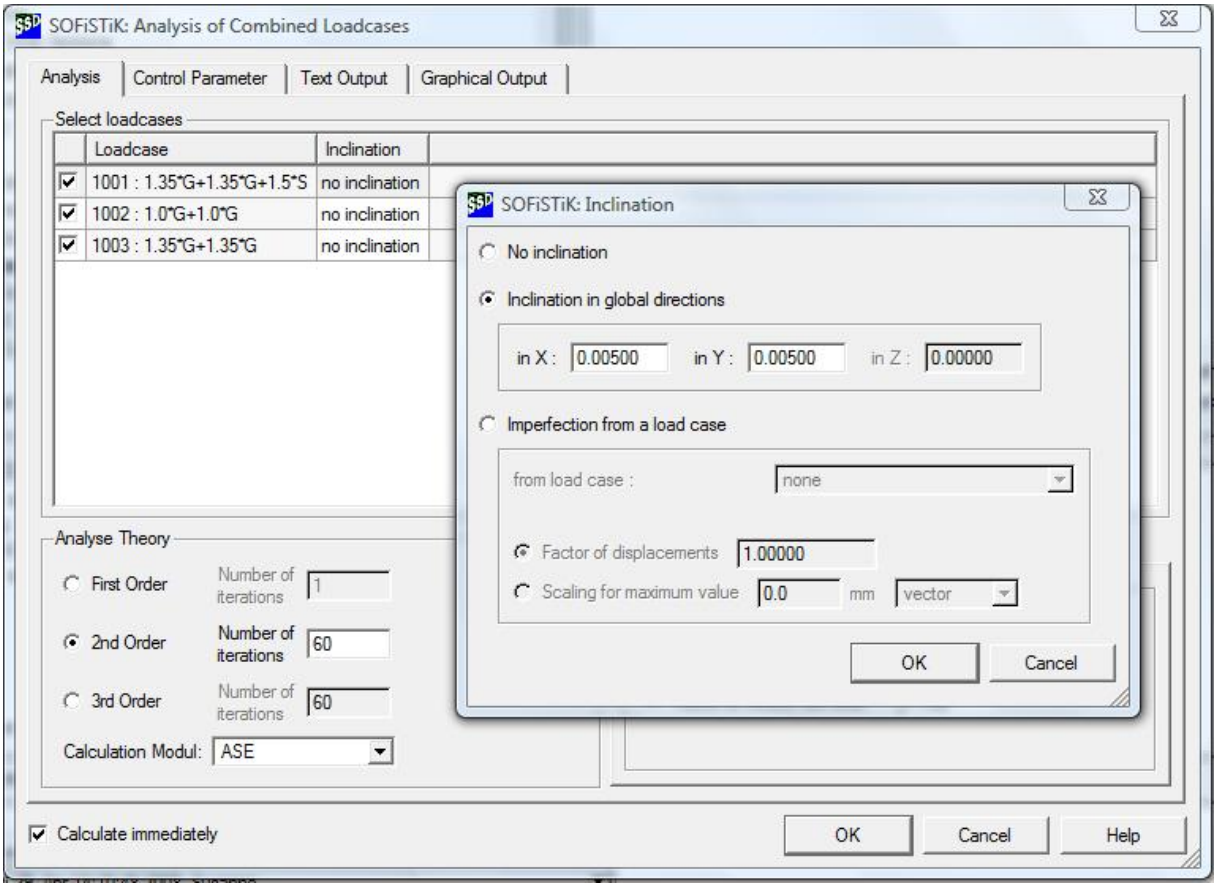


Figure 22: Analysis of Combined Loadcases

5.5 Steel Design

Task: Design Steel Construction

The design will be automatically done for all load combinations. For each element type (beam/ truss/ cable) the program will calculate the stresses and the degree of utilization.

On the tab “design” you can make sure that the buckling of the truss elements will be considered.

When using the option “Buckling” Design for Beams, you have to define the buckling length for every beam manually.

When using the option “Buckling Design for Trusses” make sure every truss element is meshed as one element.

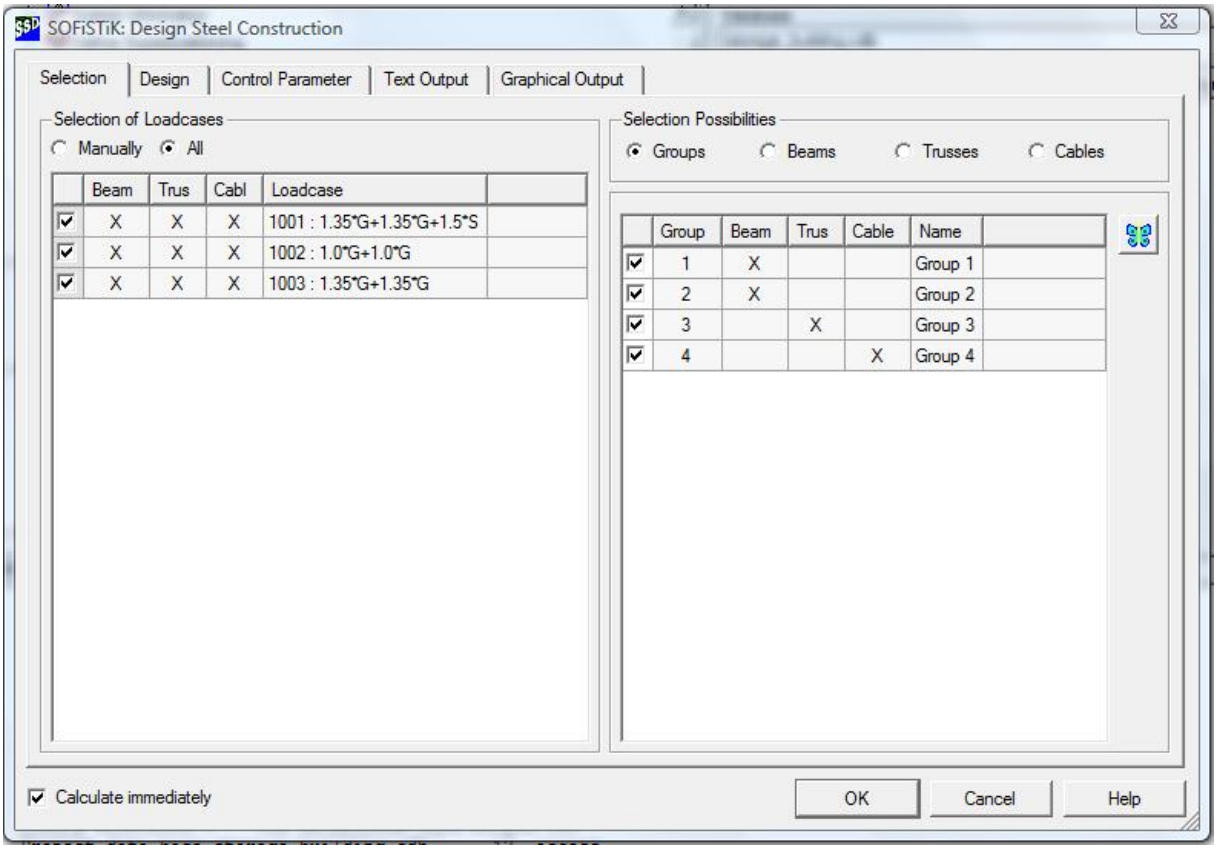


Figure 23: Design Steel Construction

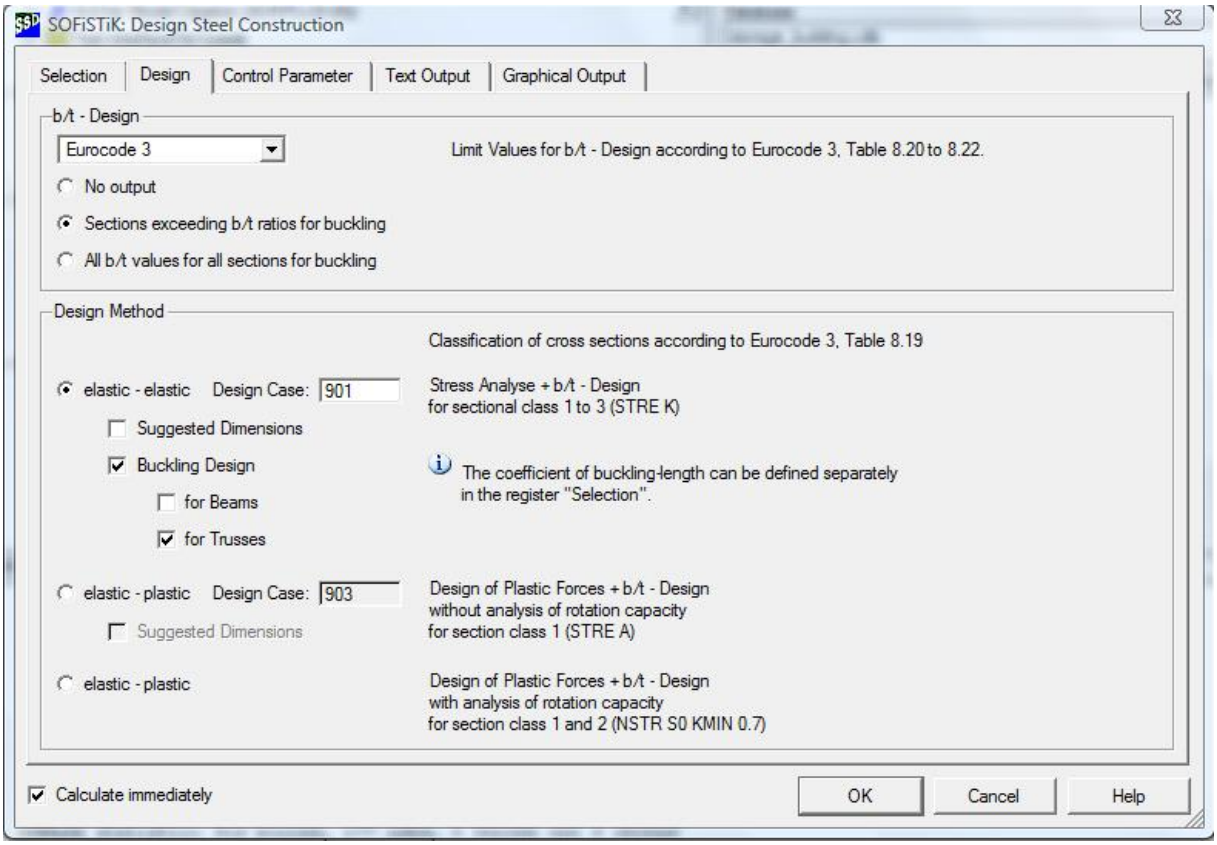


Figure 24: Design with buckling for trusses

6 Notes